Ministry of Education and Science of Ukraine
Lutsk National Technical University

CAD in Construction

Methodological instructions for laboratory classes
for applicants of the first (bachelor's) level of higher education
of the educational and professional program "Construction and Civil Engineering"
of the field of knowledge 19 Architecture and Construction of the specialty 192 Construction and Civil Engineering of full-time and part-time forms of study

Lutsk 2026

UDK 004.4 (07)
C 13

To print
Head of the educational and methodological council of the Architecture, Construction and Design faculty of LNTU _____________ O. ANDRIICHUK

An electronic copy of the printed edition has been submitted for inclusion in the repository of LNTU
Library director ___________ N. POLISHCHUK

Recommended for publication by the educational and methodological council of the Architecture, Construction and Design faculty of LNTU,
protocol № ___ from «____»____________2026_.

Considered and approved at the meeting of the Construction and Civil Engineering Department of LNTU, protocol №__ from «___»_______2026.

Head of the Construction and Civil Engineering Department
___________ O. UZHEHOVA

Compiler: __________ S. ROTKO, Ph.D., Associate Professor of the Construction and Civil Engineering Department of LNTU,

Reviewer: __________ O. UZHEHOVA, Ph.D., Associate Professor of the Construction and Civil Engineering Department of LNTU
Responsible
for the release__________ O. UZHEHOVA, Ph.D., Associate Professor, Head of the Construction and Civil Engineering Department of LNTU

C13 **CAD in Construction** [text]: methodological instructions for laboratory classes for applicants of the first (bachelor's) level of higher education of the educational and professional program "Construction and Civil Engineering" in the field of knowledge 19 Architecture and Construction, specialty 192 Construction and Civil Engineering full-time and part-time forms of study / compiled by S. Rotko. Lutsk: LNTU, 2026. 188 p.

The methodological development includes algorithms for calculating and designing building structure elements within the LIRA-FEM and MONOMAKH-SAPR software complexes.

CONTENT

INTRODUCTION

The use of advanced information technologies is key to successful calculations at the design stage of new buildings and structures, as well as when assessing the bearing capacity of existing ones. Currently, there are numerous automated software packages. The results obtained with their help allow for significantly improving the quality and speed of solving relevant engineering problems.

The adoption of technically sound and supported by appropriate calculations of structural solutions is the most important stage of design. Therefore, it is of great importance for higher education students to master modern automated design tools for calculating the strength, stability, rigidity, and design of building structure elements.

The course "CAD in Construction" of the educational and professional program "Construction and Civil Engineering" of the field of knowledge 19 Architecture and Construction, specialty 192 Construction and Civil Engineering teaches the basics of practical work with the design and computing complexes LIRA-SAPR and MONOMAKH-SAPR. This methodological development is intended to assist higher education students in completing laboratory tasks from this course and contains algorithms for calculating and designing reinforced concrete and metal structures in the environment of the LIRA-SAPR and MONOMAKH-SAPR PCs.

Course Objectives

The purpose of teaching the academic discipline "CAD in Construction" is to prepare higher education applicants for professional activity in the field of construction design in the conditions of modern information technologies.

The objectives of the course are for students to acquire the necessary knowledge about modern computer-aided design (CAD) systems in construction and to obtain practical skills in design work on the calculation and design of building structures, buildings, and structures in the environment of design and computing complexes LIRA-SAPR and MONOMAKH-SAPR.

After completing the course, the student must know:

- principles of designing elements of building structures;
- methods of selecting and checking reinforcement of reinforced concrete elements;
- basic provisions of calculating and designing steel structures.

The student must be able to:

- perform design work in the construction industry using design and computing complexes LIRA-SAPR and MONOMAKH-SAPR;
- calculate, design, investigate construction objects;
- use the necessary computer software products in work;
- apply the basic principles, theories and methods of structural mechanics to calculate elements of buildings and structures under the action of loads and influences of various nature, taking into account their interaction, using automated design systems;
- calculate and design reinforced concrete (monolithic and prefabricated), metal and wooden structures using the requirements of regulatory documents, providing reliable and economically justified design solutions;
- Calculate and design load-bearing structures of reinforced concrete, metal, and wooden structures, including using modern information technologies.

Content Module 1

Computer-aided implementation of the calculation and design of structures of buildings in the LIRA-FEM

Topic 1. Introduction to the LIRA- FEM

Laboratory Work No. 1

LIRA- FEM ribbon interface.
General algorithm for calculating structures

Purpose and plan of the session

To get acquainted with the ribbon interface and the capabilities of the software package. To get acquainted with the general algorithm for calculating structures

The ribbon interface of the LIRA- FEM

The ribbon interface of the LIRA-FEM contains the following main components: tabs, contextual tabs, panels, and menus.

A tab is an element of the graphical user interface that allows you to switch between certain sets of interface elements in one window.

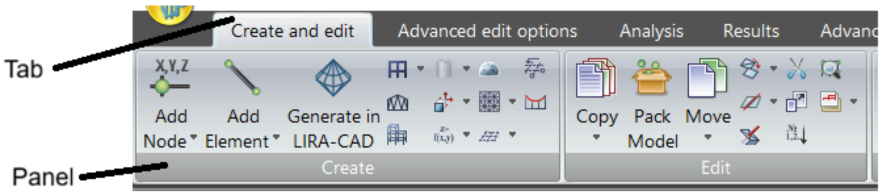


Fig. 1 – Ribbon tabs and panels

Contextual tabs (Nodes, Bars, Plates) appear when nodes and elements are selected and disappear after they are deselected (Figs. 2, 3, 4).

The **Quick Access Toolbar** (Figure 6) is located at the top of the ribbon and provides direct access to a specific set of commands. The bar is customizable and contains a set of commands that are independent of the tab currently displayed on the ribbon.

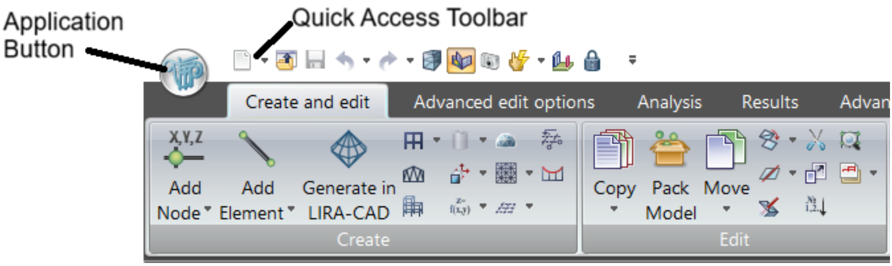


Fig. 2 – Quick Access Toolbar

Program menu (Fig. 3)—a menu for working with document files and setting units of measurement, interface language, and parameters for graphical display of the scheme and calculation results. This command is called by clicking on the button . The main page of the menu (on the right) displays a list of documents that were opened recently (Fig. 3).

The program menu contains operations that provide work with files used by VIZOR-SAPR. To manage the task as a whole, commands collected in the program menu are used. The menu consists of two panels: the left one displays groups of commands for working with files, and the right one displays a list of commands in the selected group.

Thus, the **New group** (Fig. 4)—a group of commands for creating a new problem file—contains commands for specifying the schematic characteristics:

• the first schema characteristic—two degrees of freedom in the node (displacement X, Z) XOZ;
• the second schema characteristic—three degrees of freedom in the node (displacement X, Z, Uy) XOZ;
• the third schema characteristic—three degrees of freedom in the node (displacement Z, Ux, Uy) XOY;
• the fourth schema characteristic—three degrees of freedom in the node (displacement X, Y, Z);
• the fifth schema characteristic—six degrees of freedom in the node (displacement X, Y, Z, Ux, Uy, Uz).

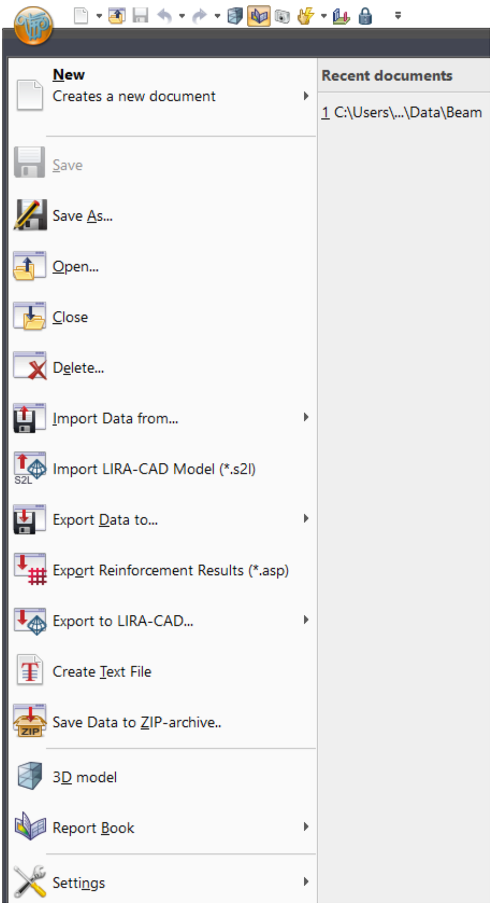


Fig. 3 – Expanded Program Menu

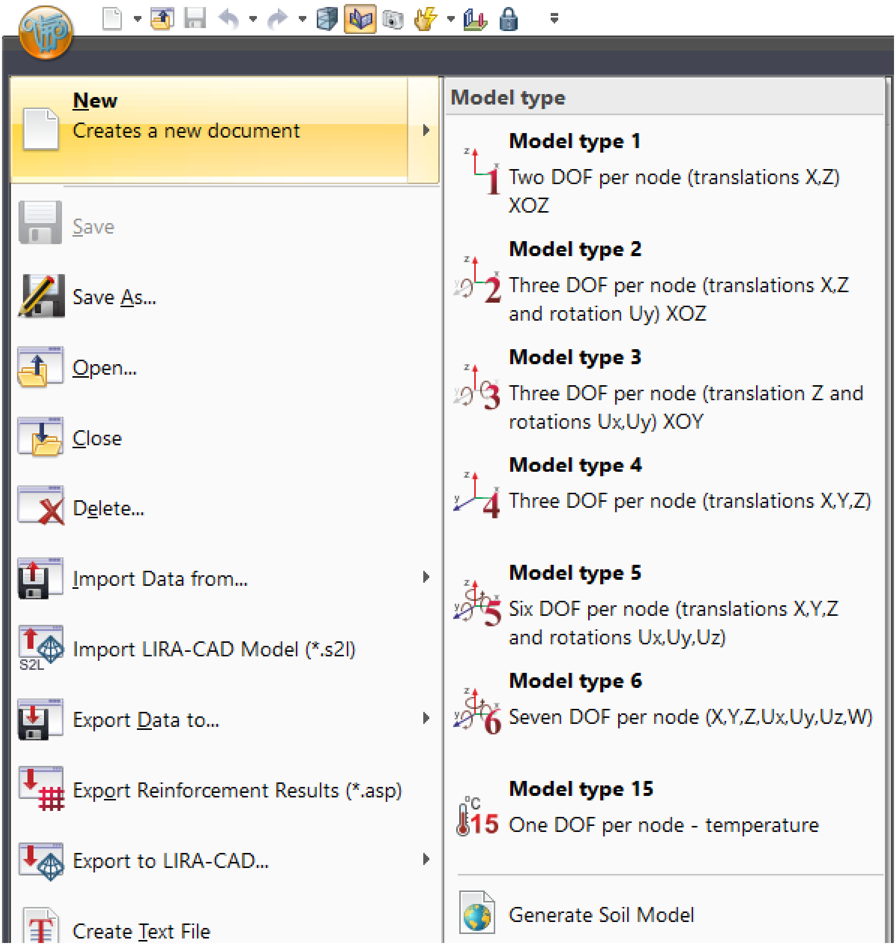


Fig. 4 – Expanded program menu, New command group

Another important group of commands – **Settings** (Fig. 5) – allows you to change the program settings (units of measurement, scale parameters, colors, calculation parameters, number and font formats, and interface language).

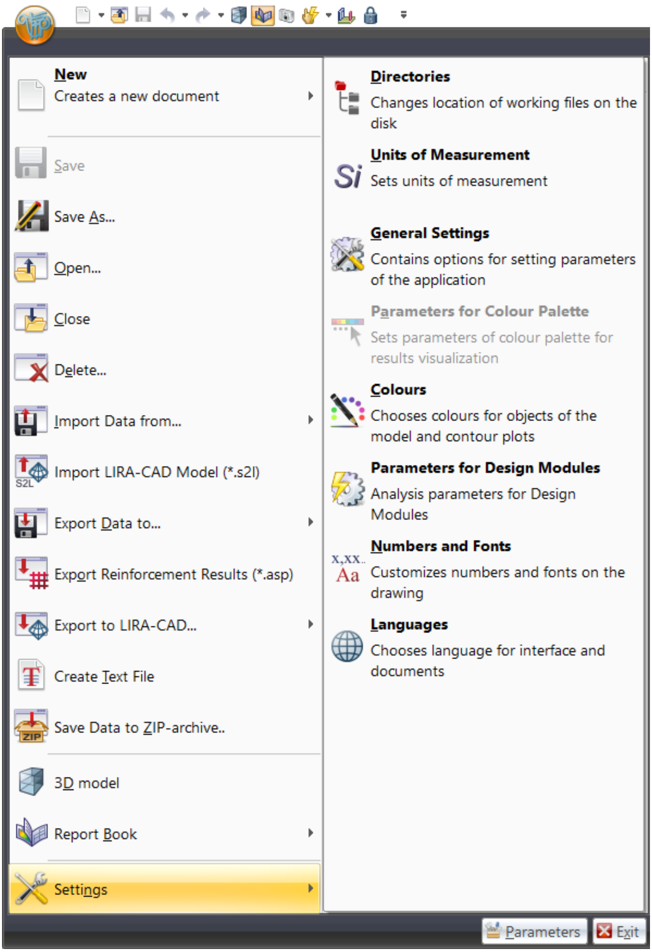


Fig. 5. Expanded program menu, Settings command group

Status bar (Fig. 6)—a panel at the bottom of the window, designed to display tips for menu items and display elements, and also contains blocks for loading the calculation scheme and loading the analysis (in the results analysis tabs) and a block for construction.

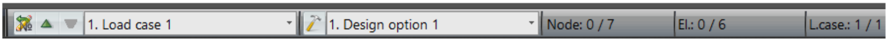


Fig. 6 – Status bar

On the **Create and Edit** tab (Fig. 7), operations are performed to create and basic edit the geometry of the scheme, assign stiffness characteristics, and generate loads.

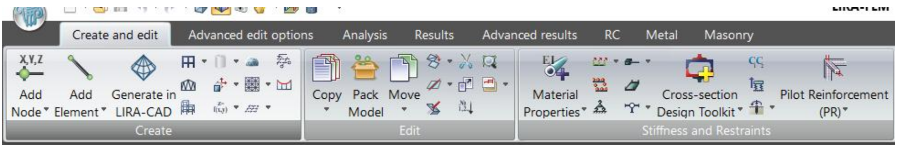


Fig. 7, a – Creation and Editing Tab

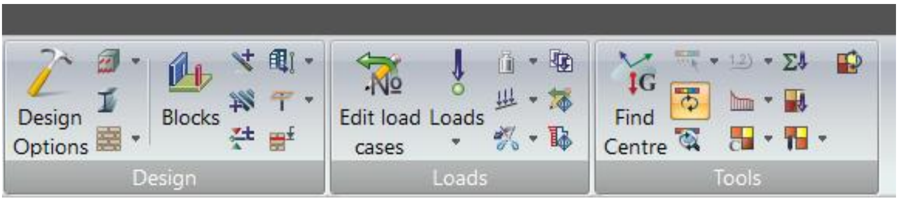


Fig. 7,b – Creation and Editing Tab (continued)

The **Create and Edit** tab contains the following panels:

• Create – here you select typical operations for creating the geometry of the scheme and triangulating the contours;

• Edit – operations for basic editing of the scheme (copying, moving, scaling), packing the scheme, and editing the triangulation mesh;

• Stiffnesses and ties – operations for assigning stiffnesses and materials to the scheme elements, modeling fasteners, etc.;

• Design – operations for creating design options and assigning materials to them, working with structural blocks, creating structural or unified elements for further calculation of reinforcement;

• Loads – operations for forming loads and assigning loads to nodes and scheme elements;

• Tools – operations for configuring the graphical display of results and output data of the scheme.

On the **Advanced Edit** tab (Fig. 8), you can perform operations for advanced editing of the scheme, assembling the scheme from subschemes, working with blocks and superelements, and connecting to the soil foundation scheme. The **Tables** panel contains commands that allow you to create, layout, and print a table, report, or explanatory note that is needed to document the current task.

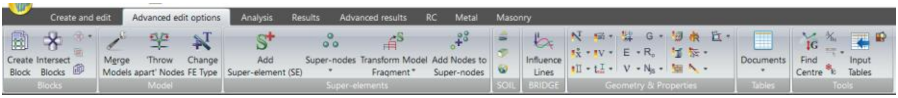


Fig. 8 – Advanced Editing Tab

On the **Analysis** tab (Fig. 9), operations are performed to specify data for static, dynamic, and additional calculations; generate tables; control parameters for calculation; and launch a calculation task.

Fig. 9 – Analysis Tab

The **Analysis** tab contains the following panels:

• Calculation – operations to change calculation parameters for the current problem, load data to the calculation processor, and perform the calculation;

• Dynamics – operations to organize the calculation for dynamic effects, specify the characteristics of each specific dynamic effect;

• DCF – operations to form calculation combinations of forces;

• Add calculations – specification of initial data for calculating displacements in nodes and forces (stresses) in elements from standard and arbitrary linear combinations of loads, for calculating loads on a fragment, for calculating principal and equivalent stresses in finite elements, for calculating stability;

• Nonlinearity – specification of parameters that determine the specificity and organization of the step process for solving nonlinear problems;

• Installation – specifying information for computer modeling of the construction process, which involves the installation and/or dismantling of elements, changing the conditions for securing the structure or connecting elements to each other.

The **Results** tab (Fig. 10) contains the most frequently used results analysis functions: displaying numerical and graphical information on the movement of nodes, forces and stresses in the elements of the design scheme.

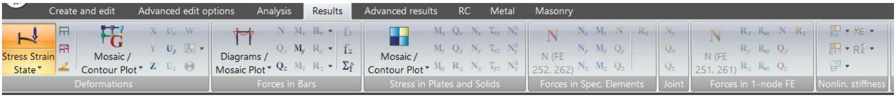


Fig. 10 – Results Tab

The **Results** tab contains the following panels:

- Deformations (Fig. 11)—operations that allow you to display deformations of a structure.

Fig. 11 – Deformation Panel

- Forces in rods (Fig. 12) – operations for displaying diagrams and mosaics of forces in rods.

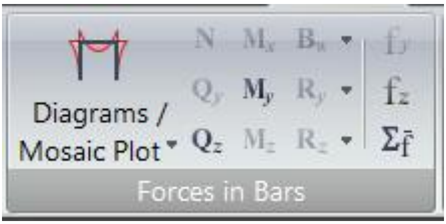


Fig. 12 – Forces in Bars

- Stresses in plates and volumetric SEs (Fig. 13) – operations for coloring plates and surfaces of volumetric elements according to the values of stresses *in them.*

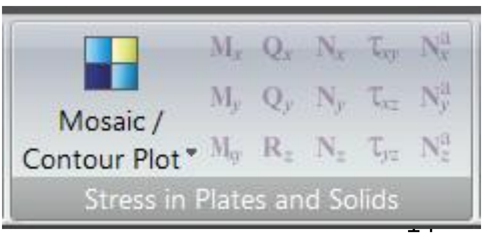


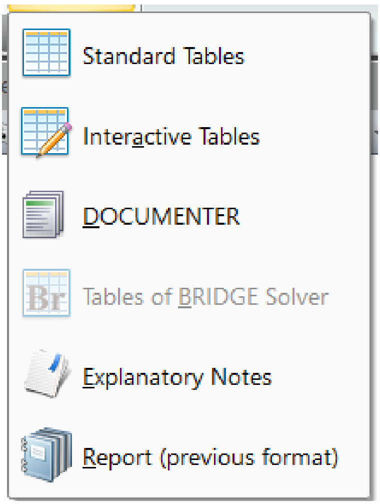
Fig. 13 – Panel Stress in plates and Solids

• Tools (Fig. 14) – the main tools for configuring the graphical display mode of the scheme and the functions for presenting the results.

Fig. 14 – Tools Panel

• Tables (Fig. 15) – displaying a numerical representation of results and launching modes for generating a report and explanatory note.

Fig. 15 – Tables Panel



Documentation drop-down list (Fig. 16) – the drop-down list contains operations for generating documentation for the current task, tables based on the results of static/dynamic calculations, creating and arranging drawings with various fixed variants of the calculation scheme and calculation results, generating a report or explanatory note.

Fig. 16 – Documentation – drop-down list

The **Design** tab (Fig. 17) contains operations for specifying initial data for design, calculating reinforcement of reinforced concrete elements, checking and selecting steel sections, displaying numerical and graphical information on the results of design system calculations, and launching local modes.

Fig. 17, a – Design Panel

Fig. 17, b – Design Panel (continued)

The **Design** tab contains the following panels:

- Calculation – operations related to the calculation of reinforcement.

Fig. 18 – Calculation Panel

• Plate reinforcement – operations for displaying the results of plate element reinforcement.

Fig. 19 – Plate Reinforcement Panel

• Bar Reinforcement – operations for displaying the results of bar element reinforcement, beam and column design.

Fig. 20 – Bar Reinforcement Panel

• Tools – operations for managing the color settings of isofields and mosaics of the initial data of the calculation scheme, the results of static/dynamic calculations, the results of checking and selecting steel sections, the results of determining areas and selecting reinforcement, operations for configuring the scale update and color display of nodes/elements of the scheme on the screen, as well as the ability to open the scheme in SAPFIR, etc.

Fig. 21 – Tools Panel

- Design – operations for changing stiffnesses and specifying initial data for design.

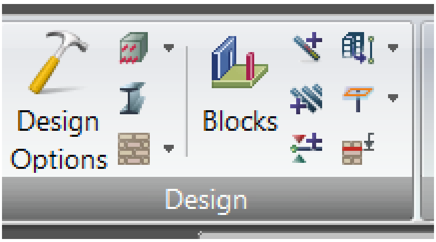


Fig. 22 – Design Panel

Steel: calculation – operations for calculating steel elements.

Fig. 23 – Analysis Panel

• *Tables – operations for displaying numerical representation of results on the screen, as well as launching modes for generating a report and explanatory note.*

Fig. 24 – Tables Panel

General algorithm for calculating structures in the LIRA-FEM

1. Creating a file (the name must contain up to 30 characters), selecting the design scheme feature, comments.
2. Setting up the program (units of measurement, interface language, main panels).
3. Selecting the cross-section calculation option (based on efforts from individual loads, RSZ or RSN). Selecting the design option:
• for reinforced concrete structures – according to DBN V.2.6-98:2009. Structures of buildings and structures. Concrete and reinforced concrete structures. Basic provisions;
• for MK – according to DBN V.2.6-198:2014. Steel structures. Design standards.
4. Creating a geometric diagram of the structure.
5. Setting boundary conditions.
6. Creating and assigning stiffnesses.
7. Visual 3D control.
8. Specifying loads.
9. Static calculation of the structure.
10. Analysis of the results of the static calculation.
11. Purpose of materials:
• for reinforced concrete – type, concrete, reinforcement;
• for reinforced concrete – material, additional characteristics, selection restrictions.
12. Full calculation (static and structural).
13. Review and analysis of the results:
• for reinforced concrete – selection of reinforcement,
• for reinforced concrete – checking and selection of sections.

Topic 2: Calculation and design of a reinforced concrete beam

Laboratory Work No. 2

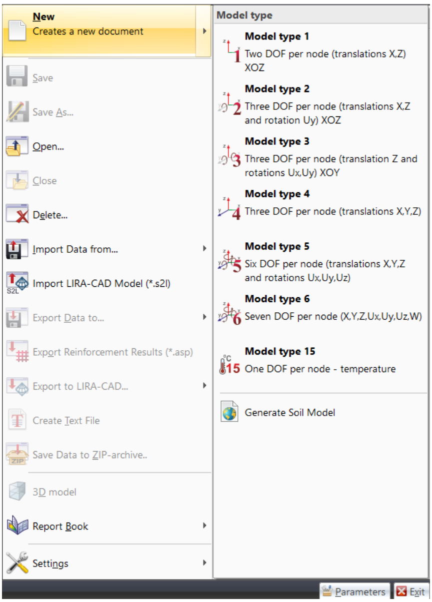
Creating a calculation model of a beam

Task. Perform the calculation of a reinforced concrete beam with a rectangular cross-section measuring 30x60 cm, a span of 6 m, made of concrete of class C25/30, reinforced with longitudinal reinforcement of class A400C, loaded with a uniformly distributed load along the length with an intensity of q=30kN/m. Analyze the results of the static calculation.

Purpose and plan of the session

Create a new project. Configure the program. Specify the calculation and design option. Create a geometric diagram. Specify boundary conditions. Specify stiffness characteristics. Specify the load. Perform the calculation.

Creating a new project



To start working with LIRA-SAPR, execute the Windows command: **Start ➤ Programs ➤ LIRA-FEM 2024.**

To create a new file, open the **Programs Menu** , select the **New** item, then – **The second feature of the scheme** (Fig. 25) – three degrees of freedom in the node (displacement X, Z, Uy), the XOZ plane.

In the Diagram **Description** dialog box (Fig. 26), specify the file name and, if desired, a comment (for example, the date the diagram was created).

Fig.25 – Programs menu, selecting a scheme feature

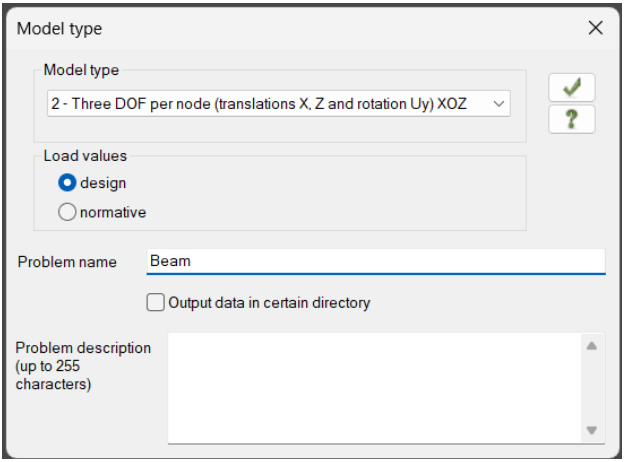


Fig. 26 – Dialog box Description of the scheme

Program settings

To set the units of measurement, you need to go to the **Programs menu** , select **Settings – Units of measurement** from the drop-down list. In the dialog box of the same name, set the necessary units: on the Scheme tab – loads and material parameters – kN, on the Results tab (stress – kPa, force – kN) – Fig. 27.

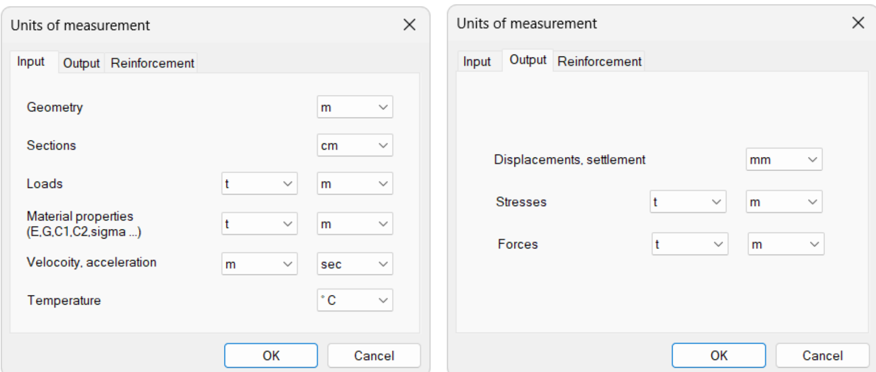
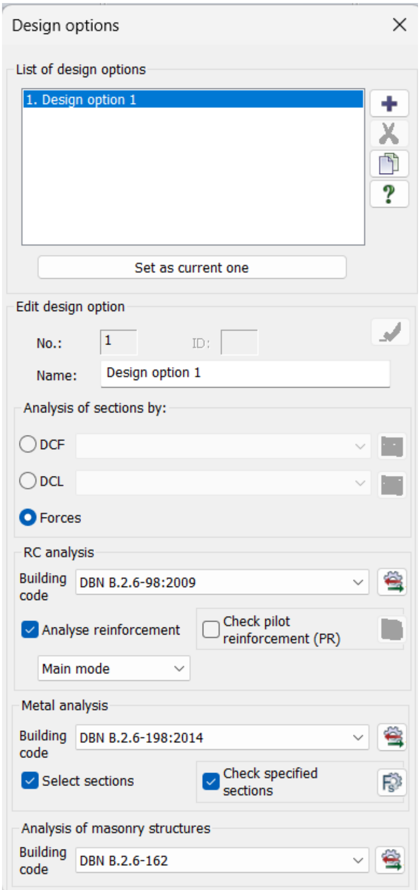


Fig. 27 – Units dialog box



In order not to clutter up the workspace with unnecessary panels, it is advisable to display the three most important ones on the screen using the right mouse button: Selection Panel, Projection Panel and Errors and Warnings.

Selecting a calculation and design option

For structural calculations according to current standards, you need to click the button  – Design options and in the Design options dialog box, select the required standard (DBN V.2.6-98:2009) – Fig. 28. In the same window, specify the calculation option (by individual efforts or design combinations of loads – DCL (or forces – DCF).

Fig. 28. Design options dialog box

Creating a geometric diagram of the structure

To create a geometric diagram of a beam, you need to call the **Create flat fragments and meshes** dialog box, on the Frame generation tab, specify six elements with a length of 1 m along the first axis (along X) (Fig. 29).

As a result, a geometric diagram of the beam will be constructed (Fig. 30).

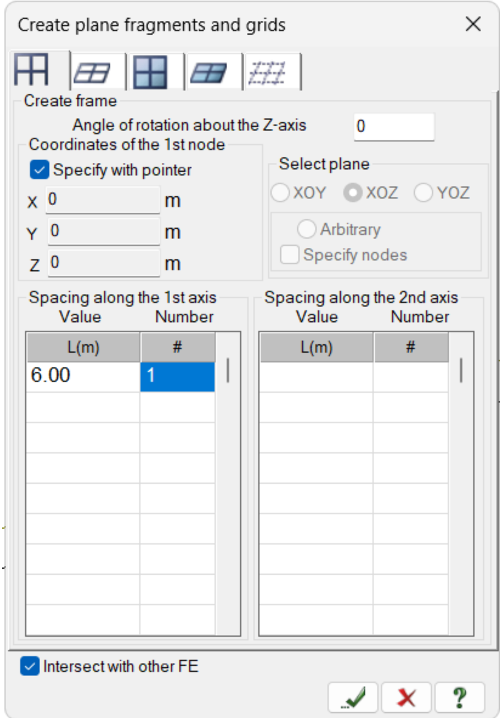


Fig. 29 – Dialog box Creating flat fragments and meshes

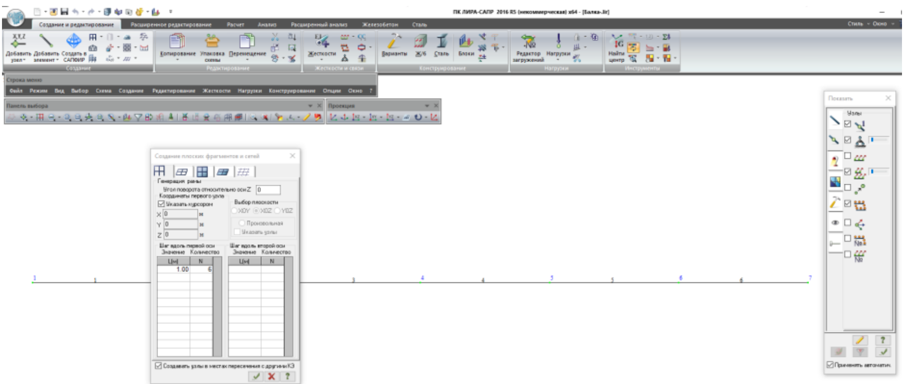
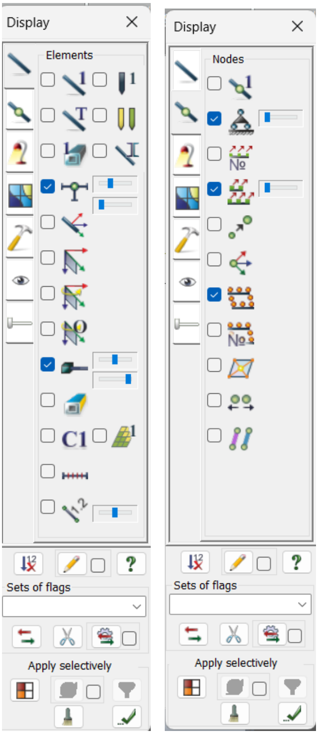


Fig. 30 – Created geometric diagram of the beam

Setting boundary conditions

The node numbers on the calculation diagram are displayed on the screen.



To display node or element numbers, you need to use the command button  – **Flags of Drowing**, which calls the **Display** dialog box (Fig. 31), where you need to check the corresponding boxes on the Elements and Nodes tabs. In this case, it is worth activating the Apply automatically option.

Fig. 31 – Display dialog box (Elements and Nodes tabs)

Activate the command button  – Mark nodes, mark node №1 (it will be displayed in red). To specify the nodes, use the button . In the **Nodes** dialog box, mark the directions in which node №1 is prohibited from moving (Z – hinged-movable support), click  – Add nodes in marked nodes. Then mark node №7. In the Nodes dialog box, mark the directions X, Z (hinged-fixed support) and click .

Creating and assigning stiffnesses

Click the button  – Element stiffnesses and materials. In the **Stiffnesses and Materials** dialog box (Fig. 32), create a list of stiffness types. On the **Stiffnesses** tab, click the Add button, select the **Standard cross-section** types tab, and activate the **Beam** cross-section.

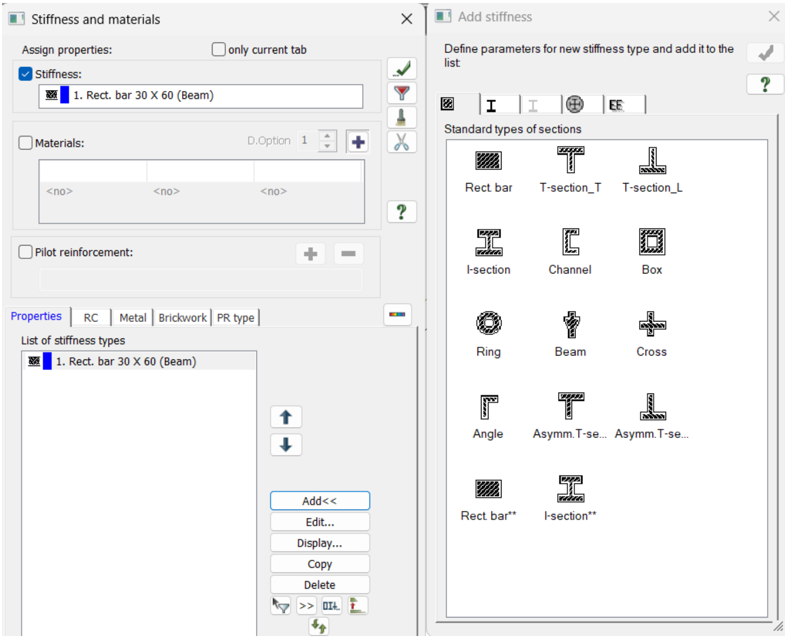


Fig. 32 – Stiffness and Materials dialog box

In the **Specify Standard Section** dialog box (Fig. 33), enter:

- modulus of elasticity for concrete of class **C25/30**

– **E** = 32500000 kN/m² (or 3.25e+007 – written in exponential form, with English keyboard layout);

- geometric dimensions – **B** = 30 cm; **H** = 60 cm;

- material density – **Ro** = 25 kN/m².

To display a sketch of the created section, click the Draw button, then confirm . After that, assign the specified stiffness to the beam elements.

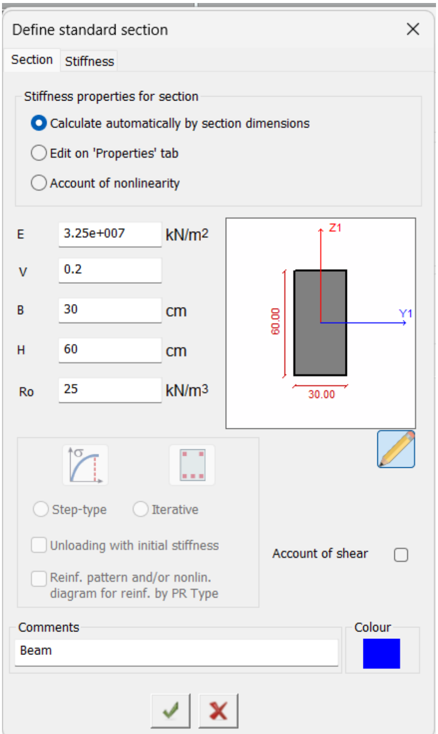


Fig. 33 – Define standard section Dialog box

Setting a load

On the Loads panel, select the Load command, and from the drop-down list, select Load on a member (Fig. 34).

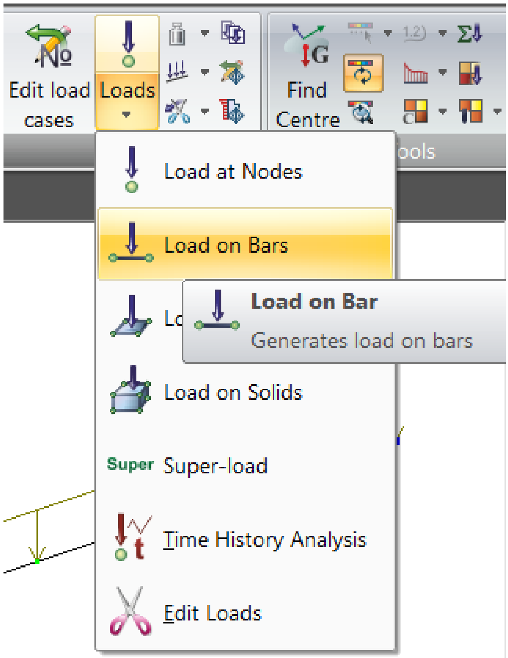
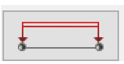


Fig. 34 – Command Load on a rod in the drop-down list

The **Load** specification dialog box will appear (Fig. 35), in which the radio buttons indicate the Global coordinate system, the direction of the load action – along the Z axis. By clicking on the uniformly distributed load button , the **Parameters** dialog box will be called. In this window, you need to enter the load intensity **P = 30 kN/m** (Fig. 35) and click the button  – Confirm.

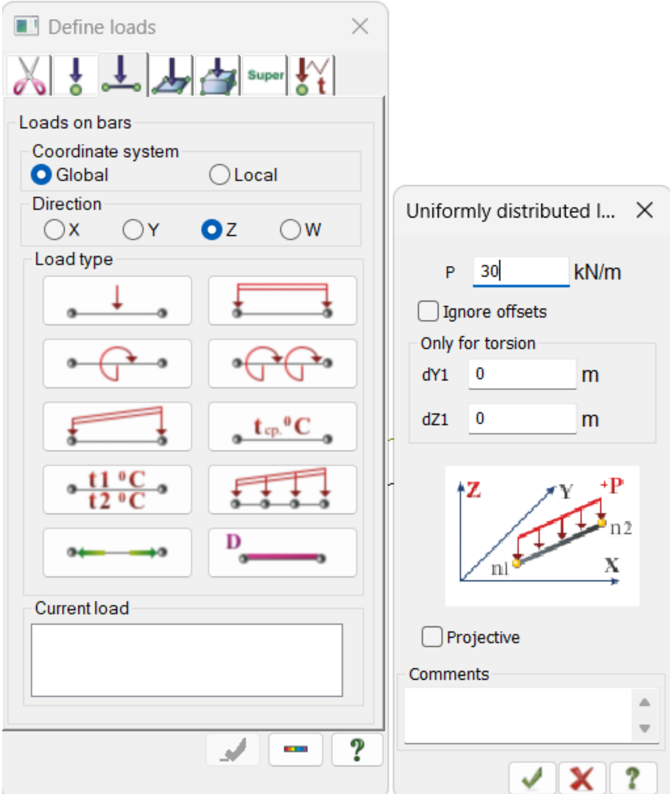


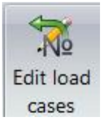
Fig. 35 – Dialog boxes for specifying loads

Using the button  – Mark elements, mark all the elements of the beam. In the dialog box **Load** definition, also click the button  – Confirm.

A linear load of 30 kN/m evenly distributed over the entire length will be applied to the beam (Fig. 36).

Fig. 36 – Applied load

To specify information about the load, you must use the button



– Edit Load cases on the **Loads** panel, Create and edit tab. The **Load Editor** dialog box will appear (Fig. 37), in which for load 1 in the Edit selected load field, you must select the Constant ribbon from the **View** list and click the button .

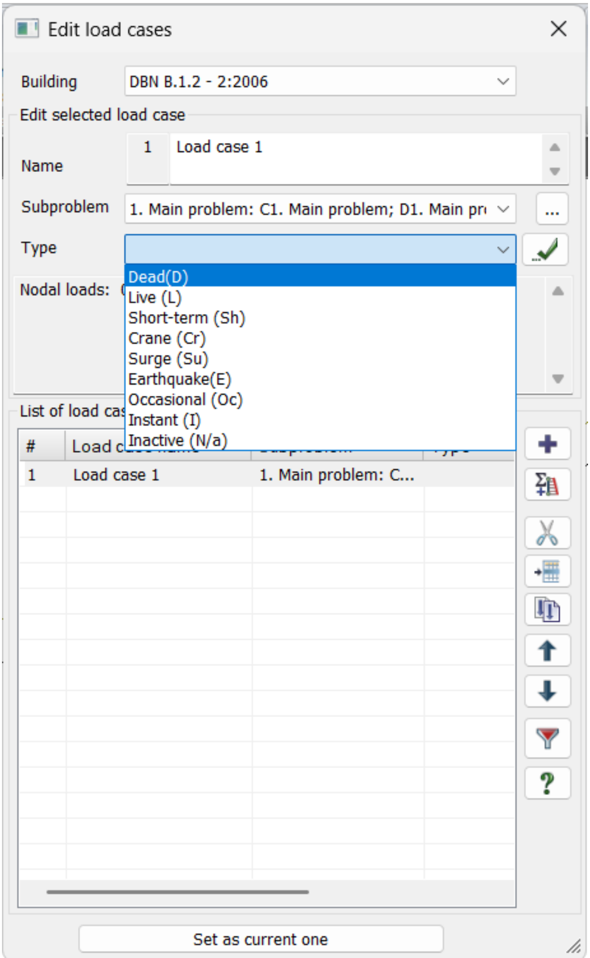


Fig. 37 – Load Editor dialog box

You can specify advanced load information before generating the load.
Full beam calculation.
Before starting the calculation, it is recommended to save the created calculation scheme. To do this, use the  – Save button on the quick access panel. In the **Save As** dialog box, give the file a name and select a folder for saving. To calculate, click the  – Perform calculation button on the **Analysis** tab.

Laboratory Work No. 3

Analysis of static calculation results and beam design

Purpose and plan of the session

Review and analyze the results of the static calculation of the beam. Display the force diagrams in the beam elements on the screen. Create tables of calculation results.

Assign material parameters for structural calculation. Analyze the results of reinforcement selection in the beam elements. Display the results on the screen in the form of a mosaic and reinforcement diagrams. Design the beam.

Viewing and analyzing static calculation results

To switch to the static calculation results visualization mode, go to the **Results** tab. In the calculation results viewing mode, the calculation diagram is displayed by default, taking into account the displacements of the nodes (Fig. 38). To display the initial diagram, click the button  (**Deformations** panel on the **Results** tab).

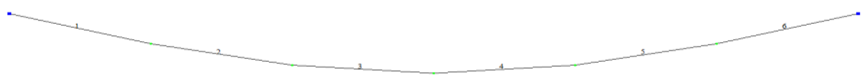
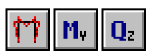


Fig. 38. Calculation diagram taking into account the displacement of nodes

To view the force diagrams, use the command buttons  (**Force in Members** panel on the **Results** tab) – Fig. 39. The force diagrams are shown in Figs. 40 and 41.

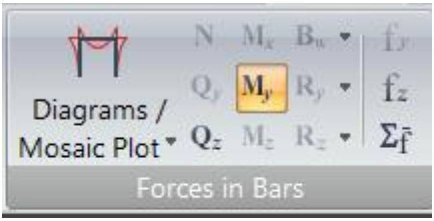


Fig. 39. Forces in Bars panel

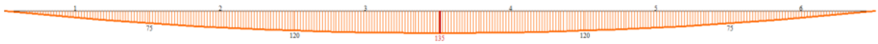


Fig. 40 – Bending moment diagram

Fig. 41 – Diagram of transverse forces

To generate and view tables of calculation results, select the **Standard Tables** command in the **Documentation** drop-down list (**Tables** panel on the **Analysis** tab) – Fig. 42.

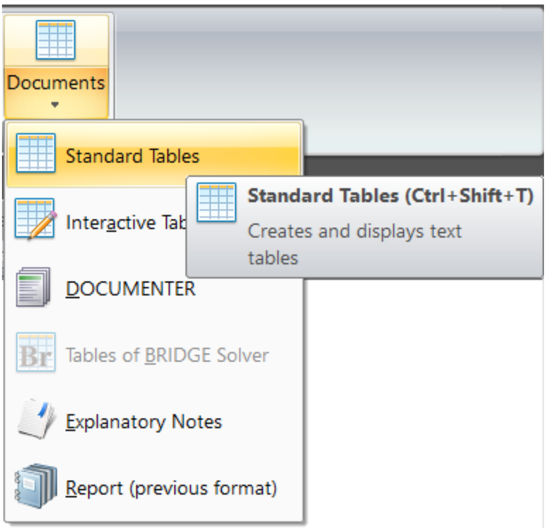


Fig. 42 – *Standard Tables* command

In the **Standard Tables** dialog box, select the **Efforts** tab (Fig. 43) and click the button .

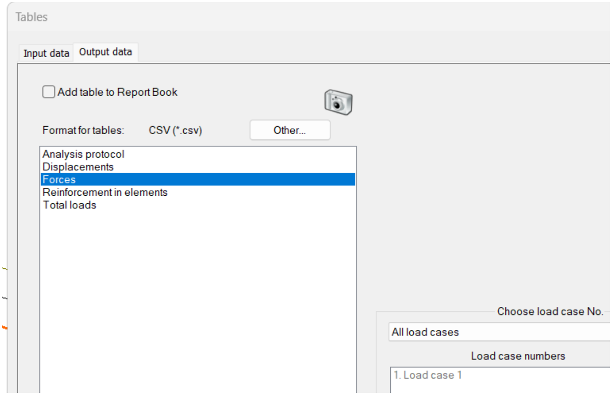
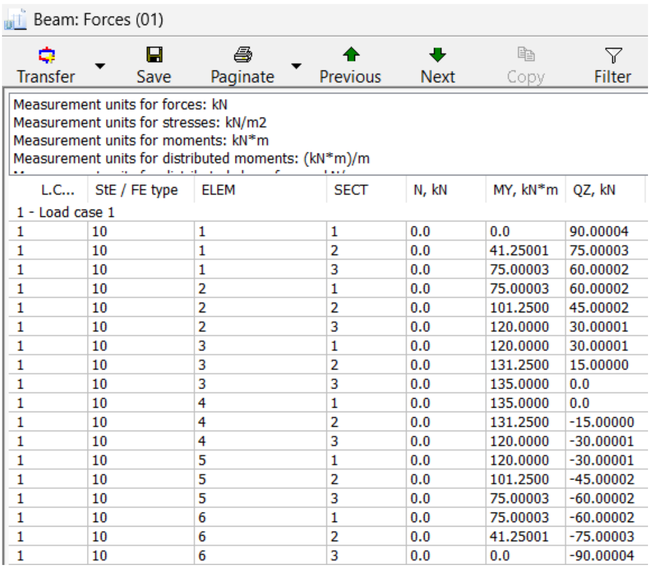


Fig. 43 – Tables dialog box

The program will generate tables with the results of static beam calculations – moments and shear forces in the elements of the calculation scheme (Fig. 44).



L.C...	StE / FE type	ELEM	SECT	N, kN	MY, kN*m	QZ, kN
1 - Load case 1						
1	10	1	1	0.0	0.0	90.00004
1	10	1	2	0.0	41.25001	75.00003
1	10	1	3	0.0	75.00003	60.00002
1	10	2	1	0.0	75.00003	60.00002
1	10	2	2	0.0	101.2500	45.00002
1	10	2	3	0.0	120.0000	30.00001
1	10	3	1	0.0	120.0000	30.00001
1	10	3	2	0.0	131.2500	15.00000
1	10	3	3	0.0	135.0000	0.0
1	10	4	1	0.0	135.0000	0.0
1	10	4	2	0.0	131.2500	-15.00000
1	10	4	3	0.0	120.0000	-30.00001
1	10	5	1	0.0	120.0000	-30.00001
1	10	5	2	0.0	101.2500	-45.00002
1	10	5	3	0.0	75.00003	-60.00002
1	10	6	1	0.0	75.00003	-60.00002
1	10	6	2	0.0	41.25001	-75.00003
1	10	6	3	0.0	0.0	-90.00004

Fig. 44 – Table of forces in beam elements

Setting material parameters for structural calculation

On the Reinforced Concrete (RC) tab in the **Stiffness and Materials** dialog box, you need to set the parameters for designing reinforced concrete structures: type, concrete, reinforcement (Fig. 45). To do this, select the Type radio button and click the Edit button.

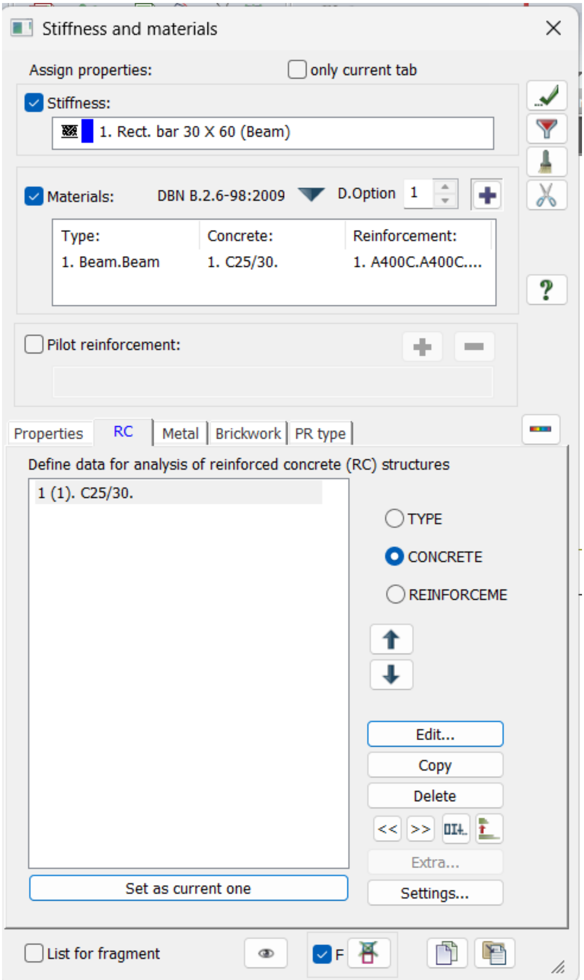


Fig. 45 – Stiffness and Materials dialog box, Reinforced Concrete tab

The **Materials for Calculating Reinforced Concrete Structures** dialog box will appear (Fig. 46), in which you should left-click in the Bar field. In the upper right corner, specify: calculation type – Beam, reinforcement – Asymmetric, system – Statically determinate, uncheck the Select corner bars command (for beams, use the discrete reinforcement algorithm). In the Concrete field, select concrete class C25/30. In the Reinforcement field, specify: longitudinal – class A400C, transverse – A240C. To save the entered data, click the  – Confirm button.

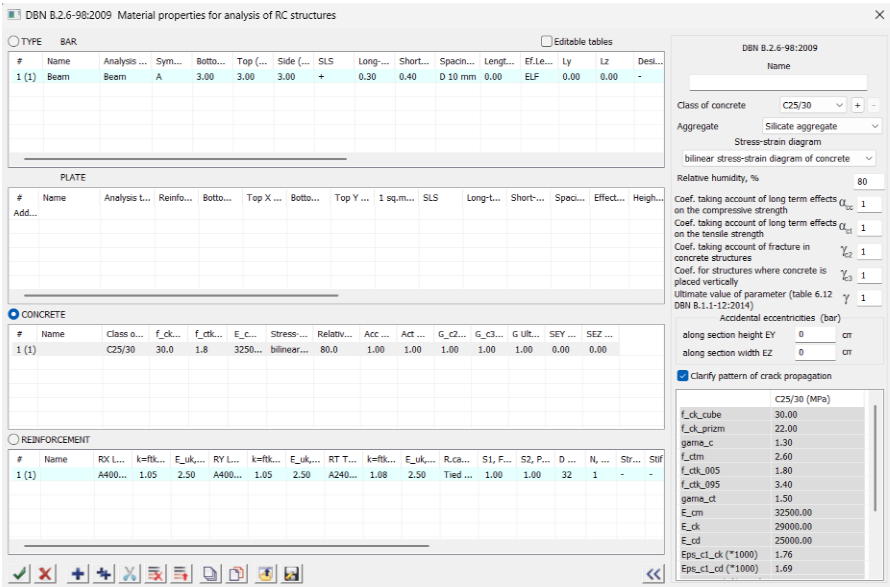


Fig. 46 – Materials dialog box for calculating reinforced concrete structures

Use the  button to select all beam elements. The elements are highlighted in red. In the **Stiffness and Materials** dialog box, click the button  to confirm. Close the **Stiffness and Materials** dialog box. To view the assigned stiffnesses, click the  button to open the **Display** dialog box, and on the Elements tab, select the Stiffness Types check box. The program will display the assigned stiffness type (1) for each finite element (FE).

Reviewing and analyzing the results of beam reinforcement

On the **RC** tab, click the 

– Reinforcement Calculation button and review and analyze the reinforcement results. To do this, select the Asymmetry command on the **Bar Reinforcement** panel to set the display mode for asymmetric reinforcement (Fig. 47).

Fig. 47 – **RC** tab

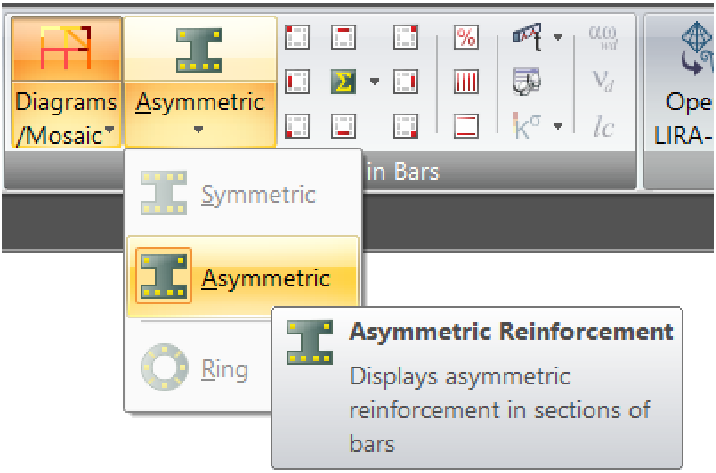


Fig. 48 – Asymmetric reinforcement command

The program displays the selected reinforcement either in a color corresponding to the scale (on the scale, this reinforcement is presented as the area of reinforcement in cm^2) or in the form of a reinforcement diagram. The user selects the desired display mode – in the first case, select the Reinforcement mosaic in bars command, in the second case – Reinforcement diagrams in bars (Fig. 50). To display the reinforcement at the bottom edge of the beam, press the button  – Distributed reinforcement AU1.

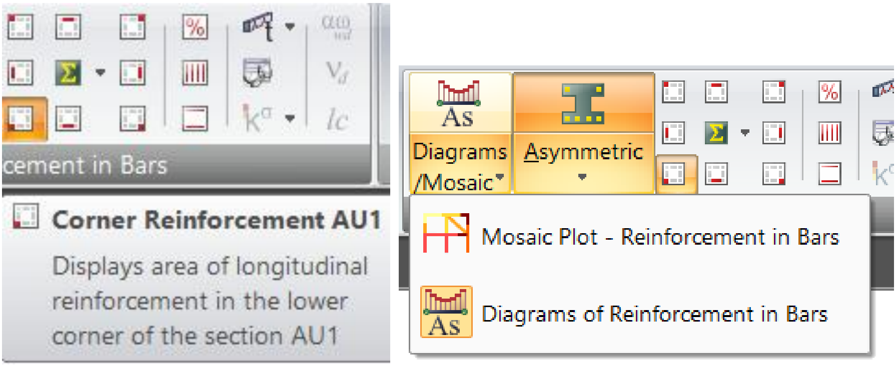


Fig. 49 – Reinforcement display modes

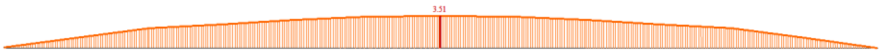
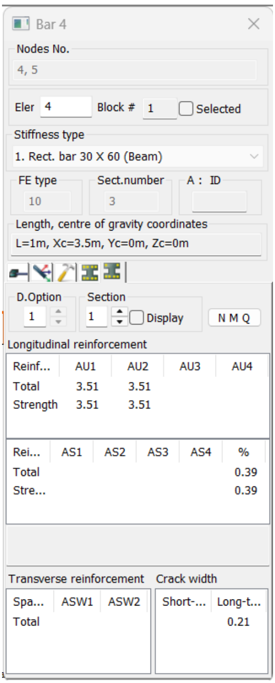


Fig. 50 – Diagram of selected reinforcement in beam elements (longitudinal reinforcement AU1 near the bottom edge)



To view information about the selected reinforcement in one of the SEs, click the  – Information about the node or element button on the **Selection Panel** toolbar and point the cursor at the desired beam element (currently No. 4). In the dialog box that appears (Fig. 51), go to the **Longitudinal reinforcement** tab (this window contains complete information about the selected element, including the results of reinforcement selection).

According to the calculation results, the area of longitudinal reinforcement in the lower (tension) zone of the beam is 702 mm^2. The required number of rods is selected according to the assortment—for example, 2∅14+2∅16 class A400C (the actual cross-sectional area will be: 307.7+401.9=709.6 mm^2).

Fig. 51 – Element Information dialog box

Topic 3. Calculation and design of reinforced concrete slabs

Task. Calculate a monolithic reinforced concrete flat slab with dimensions BxL=3×6 m and thickness h=160 mm. The slab is freely supported by its shorter sides on the walls. Perform the calculation for a 6x12 grid under the following loads:

- ✓ load 1 – dead weight of the slab;
- ✓ load 2 – quasi-permanent, evenly distributed over the entire area of the slab p2 = 0.35 kPa;
- ✓ load 3 – short-term, evenly distributed over the entire area of the slab p3 = 1.15 kPa.

Laboratory Work No. 4

Creating a calculation model of a slab, static calculation

Purpose and plan of the session

Create a new task. Create a geometric diagram of the slab. Set boundary conditions and calculation options. Simulate two loads and set detailed information about them. Generate a table of calculated load combinations (DCL). Perform a complete calculation of the model.

Creating a file

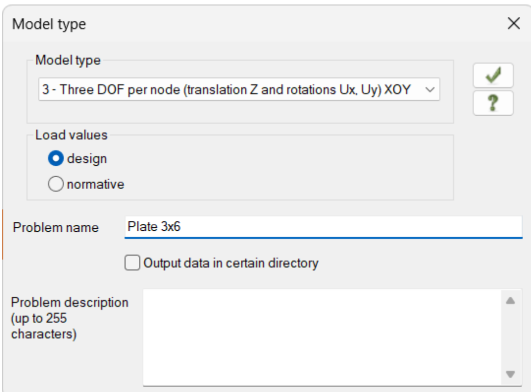
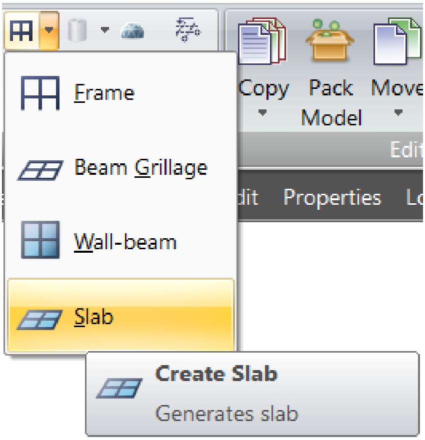


Fig. 52 – Model type dialog box

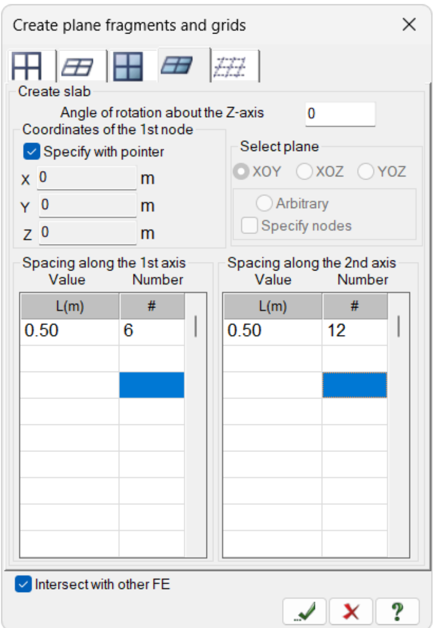
To create a new file, click the New button on the Quick Access Toolbar. In the Diagram Description dialog box, specify the Diagram Type – 3 – three degrees of freedom in the node (Z, Ux, Uy displacement) XOY, the file name, and, if desired, a comment (Fig. 52). Click the  – Confirm button.

Creating a geometric diagram of the slab



On the **Creation** and **Editing** tab, click the - Generation of Flat Fragments and Grids button to open a drop-down list, where you can select the Slab Generation command.

Fig. 53 – Calling the Slab Generation command



The **Create Flat Fragments and Grids** dialog box will appear. In the table, set the step along the first axis to 0.5 m and the number of steps to 6; along the second axis, set the step to 0.5 and the number of steps to 12 (Fig. 54). Leave the rest of the parameters as default and click the Confirm button .

To save the information about the created model, click the button on the quick access toolbar.

Fig. 54 – Create Flat Fragments and Meshes dialog box.

Setting boundary conditions

To set boundary conditions, display the node numbers on the calculation diagram by activating the command button  – Node Marking, select the nodes supporting the slab along two short edges with the cursor – Nos. 1-7 and 85-91 (they will be displayed in red). To set boundary conditions at the support nodes, open the dialog box **Connections at nodes** by clicking the button  – Connections (**Stiffness and Connections** panel on the **Create and Edit** tab).

In the dialog box, select the directions in which node movement is prohibited (Z – hinged support). To execute the command, click the  – Add restraints at selected nodes button. The color of the nodes will change from red to blue, indicating that the command has been executed (Fig. 55).

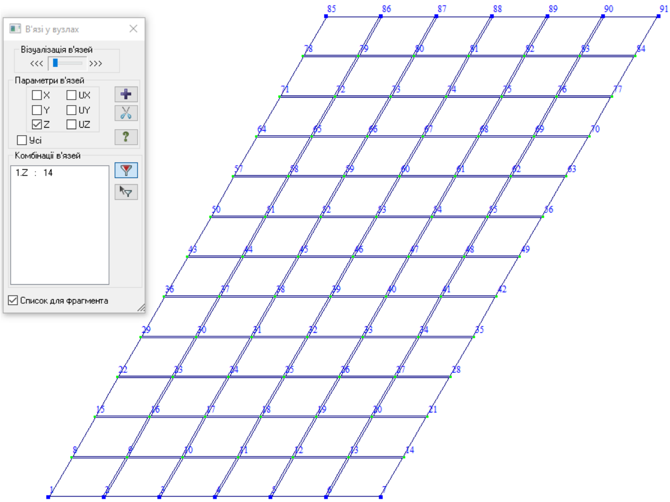


Fig. 55 – Setting constraints in the support nodes of the slab

Setting calculation and design options

For design calculations, click the button  – Design Options (**Design** panel on the **Create and Edit** tab). In the **Design Options** dialog box, specify the parameters for the first option design (selection of reinforcement according to Karpenko's theory) – Fig. 56. In the same window, select the calculation option (according to the design combinations of forces - DCF) and the applicable standards. Click the **Apply** button.

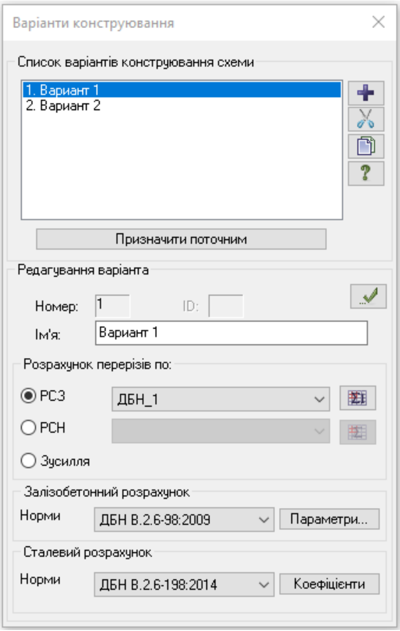


Fig. 56 – Design Options dialog box

To create the ***second*** design option (selection of reinforcement according to Wood's theory), click the button  - Create a new design option. Then specify the following parameters: design option name – Option 2; standards for reinforced concrete and steel calculations; type of cross-section calculation – according to RC. Enter the data for the design option by clicking the  - Apply button.

Double-clicking on the **Design Options List ribbon** activates the selected option in the graphical environment. Select the Option 1 ribbon and click the Set as Current button. Close the **Design Options** window by clicking the Close button .

The materials for the design option are selected in the **Stiffness and Materials** dialog box (Fig. 57).

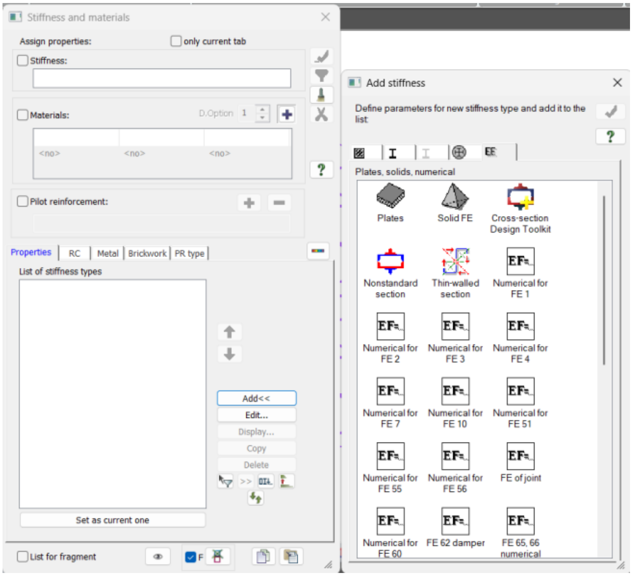
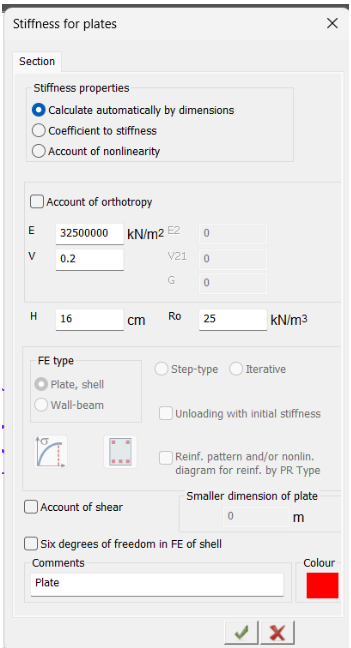


Fig. 57 – Stiffness and Materials dialog boxes, Add Stiffness



Creating and assigning stiffnesses and material parameters

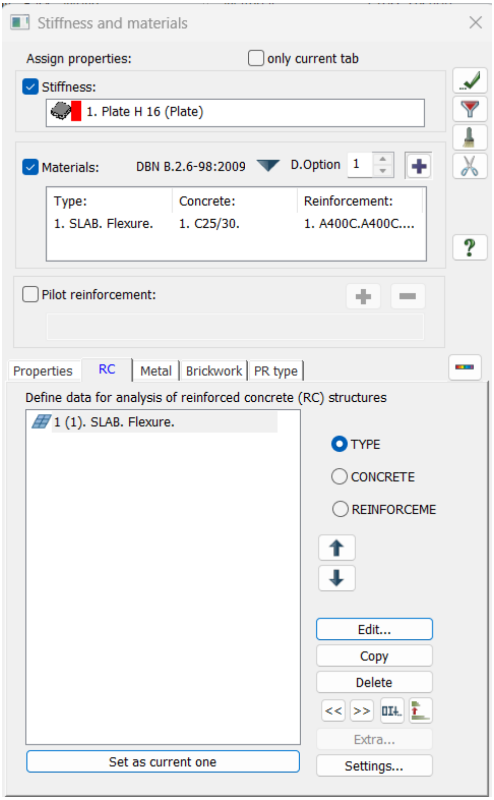
To set the stiffness type, click the button  to open the **Stiffness and Materials** dialog box (**Stiffness and Joints** panel on the **Create and Edit** tab). On the **Stiffness** tab, click the Add button. In the Add **Stiffness** dialog box, select the third tab – Plate, Volume, Numerical – and activate the Plate cross-section.

In the **Set Stiffness for Plates** dialog box (Fig. 58), enter:

the modulus of elasticity for C25/30 concrete – E=3.25e7 kN/m2;

- Poisson's ratio – V=0.2;
- thickness H = 16 cm;
- material density - Ro = 25 kN/m3.

Fig. 58 – Stiffnesses for plates Dialog box



To close the stiffness characteristics library, click the Close button  in the **Add Stiffness** dialog box or the Add button in the **Stiffness and Materials** window.

On the Reinforced Concrete (RC) tab of the **Stiffness and Materials** dialog box, you need to set the parameters for reinforced concrete structures: type, concrete, reinforcement (Fig. 59).

Fig. 59 – Stiffness and Materials dialog box, Reinforced Concrete tab

Select the Type radio button and click Edit.

The **Materials for reinforced concrete structures** calculation dialog box appears (Fig. 60), in which you left-click in the Plate field. In the upper right corner, specify: calculation type – Slab (bending), reinforcement bar spacing – 200 mm. In the Concrete field, select concrete class C25/30, in the Reinforcement field: longitudinal – class A400C, transverse – A240C. To save the entered data, click the – Confirm button .

Using the  – Mark elements button, mark all elements of the slab, and the elements will be highlighted in red. In the Stiffness and Materials dialog box, click the  – Confirm button. Close the **Stiffness and Materials** dialog box. To view the assigned stiffnesses, click the  button to open the **Display** dialog box, and on the Elements tab, select the

Stiffness Types check box. The program will display the assigned stiffness type (1) for each FE (finite element).

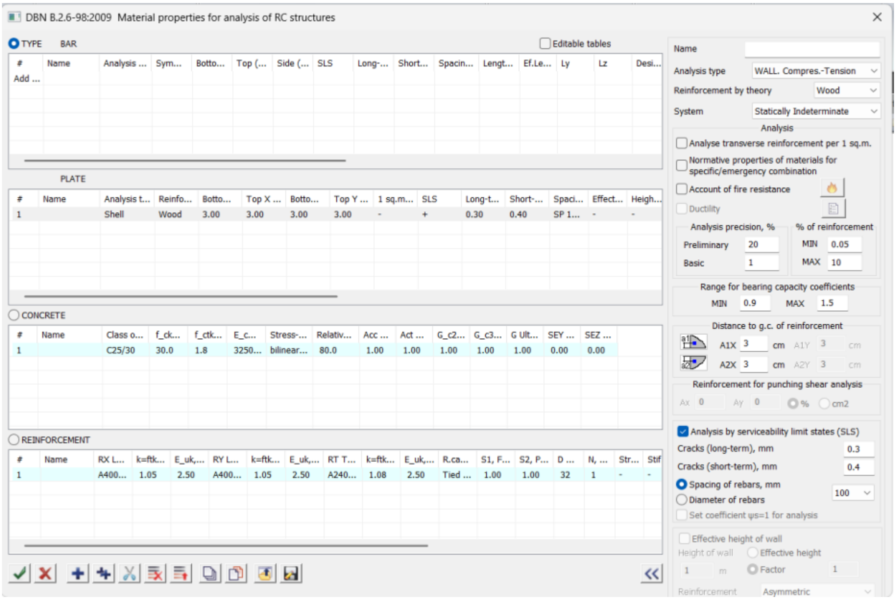


Fig. 60 – Materials dialog box for calculating reinforced concrete structures

To specify materials for the second option of reinforced concrete structures design, in the **Stiffness and Materials** dialog box (Fig. 59), use the counter to switch to option No. 2. Then select the **Type** radio button and click the **Edit** button. The **Materials for reinforced concrete structures** calculation dialog box (Fig. 60) appears on the screen, in which you specify the following parameters for plate elements:

- In the **Plate** field

- In the Calculation type line, select Plate (bending);

- Select the Select reinforcement according to Wood's theory checkbox;

- In the **Concrete characteristics** field

- In the Concrete class line, select C25/30;

- in the Stress-strain diagram line - 2-linear;

- **in the Reinforcement characteristics** field

- in the Classes field - longitudinal - A400C, transverse - A240C;

- in the Reinforcement cage type field – knitted;

- maximum diameter – 25 mm;

- leave the rest of the parameters as default.

After that, click on the button – Confirm.

To assign stiffnesses and materials to slab elements, click on the button – Mark elements on the **Selection Panel** toolbar. At the same time, the stiffness – 1 should be set in the list of the current stiffness type. Plate H 16, and in the list of current materials – Type 2. Slab (bending), Concrete – 1. C25/30, Reinforcement – 1. A400C. Use the cursor to select all elements of the diagram by stretching the "rubber window" around the group of elements. In the **Stiffness and Materials** dialog box, click the Apply button . The color of the elements will change from red to blue, which means that the elements have been assigned the current stiffness.

To switch to the first design option, in the **Stiffness and Materials** dialog box, use the **Current Design Option Number** counter to select design option 1. To assign materials to the first design option, clear the **Stiffness** check box in the Assign to Scheme Elements field. In the **Stiffness and Materials** dialog box, select line 1 in the list of general properties of materials for reinforced concrete elements with the cursor. Plate (bending), click the Assign as Current button. Select all elements of the diagram with the cursor and click the Apply button .

Loading tasks

Creating load No. 1

To set the load from the slab's own weight, click the – Add own weight button (**Load** panel, on the **Create and edit** tab). The Add own weight dialog box will appear (Fig. 61).

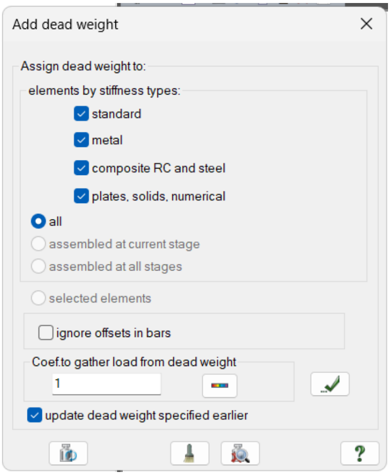


Fig. 61 – Add Custom Weight dialog box

In this window, with the All circuit elements radio button selected and a load reliability factor of 1 specified, click the Apply button  (depending on the specified volumetric weight Ro, the elements of the design circuit are loaded with their own weight) – Fig. 62.

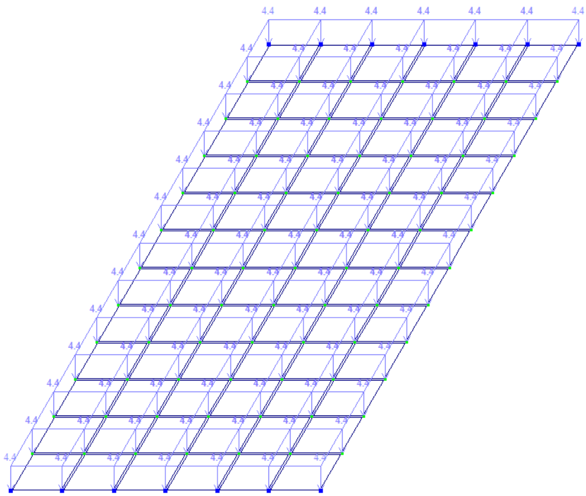


Fig. 62 – Load 1 – dead weight of the slab (constant)

Creating load No. 2

Change the number of the current load by clicking the Next load button  in the status bar (in the lower area of the working window).

On the **Load** panel (**Create and Edit** tab), select the Load command, and from the drop-down list, select the **Load to Plates** command (Fig. 63).

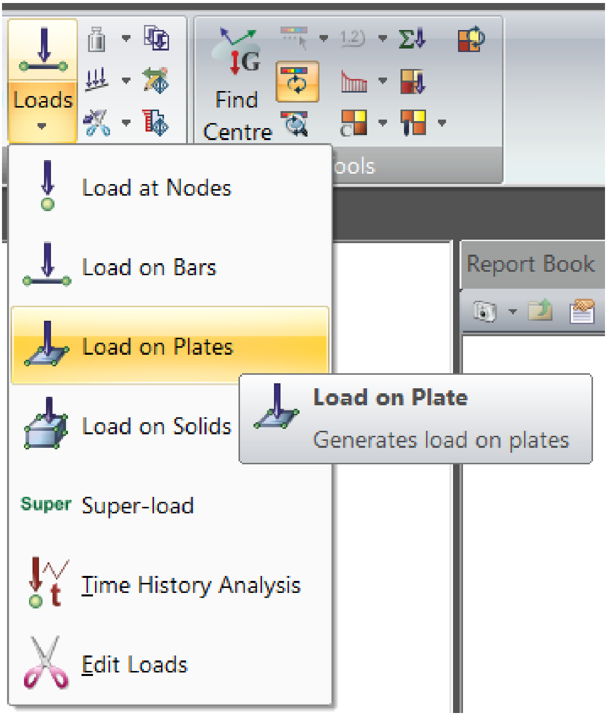
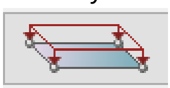


Fig. 63 – Load drop-down menu

In the **Load Assignment** dialog box (Fig. 69), specify the coordinate system – Global, the direction of load action – along the Z axis.

To specify a uniformly distributed variable quasi-steady load, click the command button  , the **Parameters** window appears, in which you enter the load intensity – P = 0.35 kN/m^2 (Fig. 64) and click the button  – Confirm.

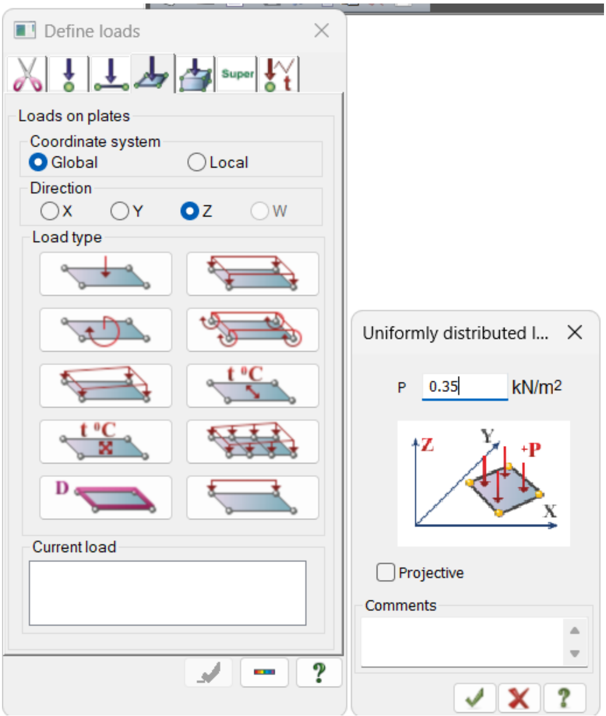


Fig. 64 – Load Assignment and Parameters dialog boxes

Use the button  to select all elements of the slab. In the **Load Assignment** dialog box, click the button  – Confirm. A uniformly distributed quasi-steady load with an intensity of 0.35 kN/m² will be applied to the slab (Fig. 65).

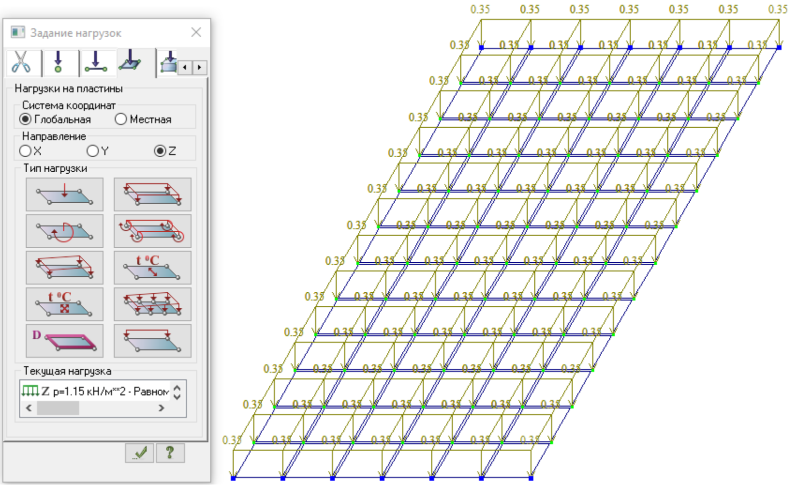
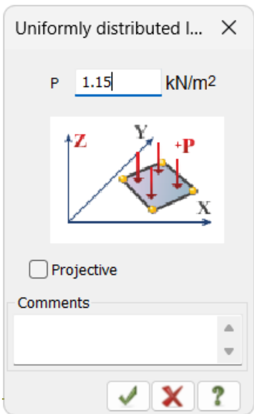


Fig. 65 – Load 2 – quasi-steady

Loading formation No. 3

Change the current loading number by clicking the button  – Next loading in the status bar (in the lower area of the working window).

On the **Loading** panel (**Creation and Editing** tab) select the **Loading** command, and from the drop-down list – the **Loading on plates** command.



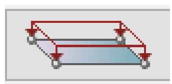
To set a uniformly distributed variable short-term load, click the command button , the Parameters window appears (Fig. 66), in which you specify the load intensity – P = 1.15 kN/m^2 and click the button  – Confirm.

Fig. 66 – Options dialog box

Use the button to select all elements of the slab. In the **Load Assignment** dialog box, click the button . A uniformly distributed short-term load with an intensity of 1.15 kN/m² will be applied to the slab (Fig. 67).

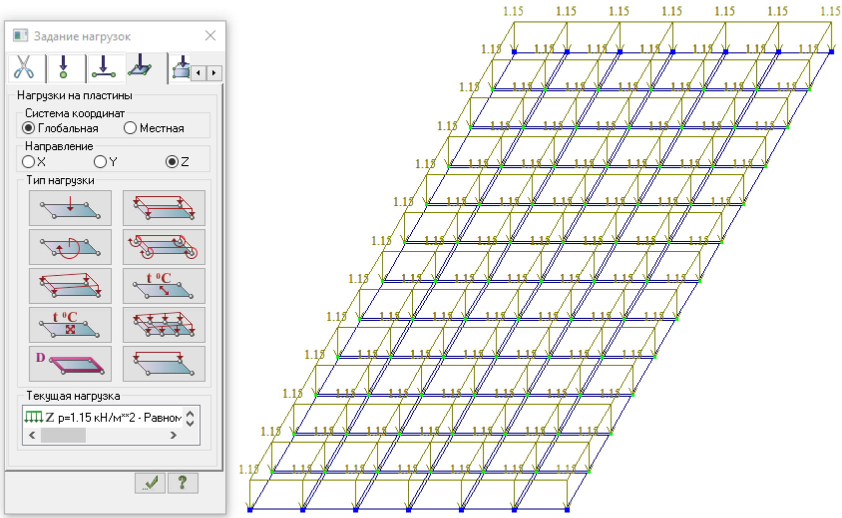


Fig. 67 – Load 3 – short-term

To specify additional information about the upload, use the – Upload Editor button on the **Upload** panel. The **Download Editor** dialog box will appear (Fig. 68), in which, for download 1, select Permanent from the View list in the **Edit selected download field** and click the – Confirm button.

For download 2, in the **Edit Selected Download field**, select Permanent from the View list and click the – Confirm button.

For download 3, in the **Edit Selected Download field**, select Short-term from the View list and click the – Confirm button.

You can also set advanced download information before creating downloads.

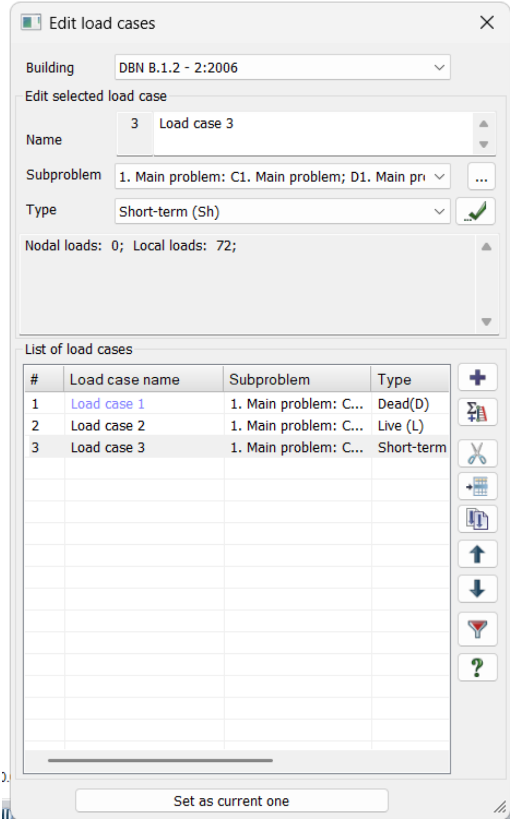


Fig. 68 – Download Editor dialog box

Generating the DCF table

Before starting the calculation, it is necessary to create a table of design load combinations. This is done when several loads of different durations are specified. According to building codes, reinforcement calculations must be performed for the most dangerous combinations of forces. Therefore, for further work in the **Reinforced Concrete Structures** mode, calculations must be performed for DCF or DCL.

The calculation of design force combinations (DFC) is performed according to the criterion of extreme stress values at characteristic points of element cross-sections based on the rules established by regulatory documents (unlike the calculation of DFC, where the latter are performed by directly summing the corresponding values of node displacements and forces in the elements).

The dialog box is opened by clicking the button  – DCF Table (DCF panel on the Calculation tab) – Fig. 69.

In the **Building Standards** field, select DBN V 1.2-2:2006, which is valid in Ukraine, from the drop-down list. For each load, specify the type of load and click the button  – Confirm.

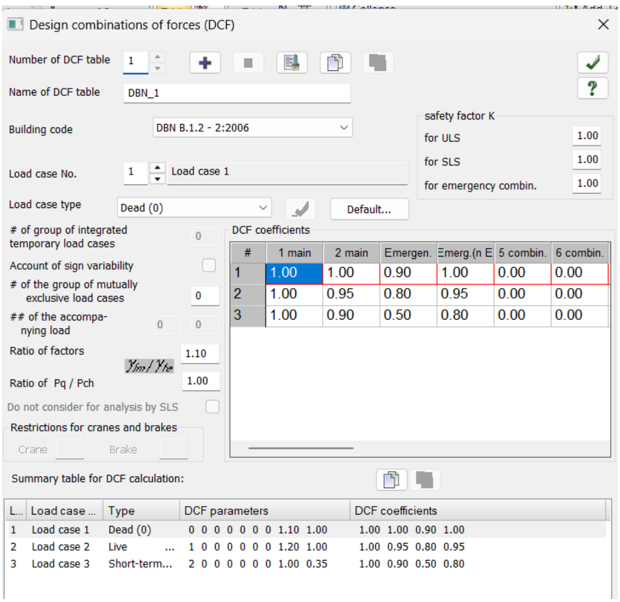


Fig. 69 – Design Combinations of Forses dialog box

Complete slab calculation

Before starting the calculation, it is recommended to save the created calculation scheme. To perform the calculation, click the button .

Laboratory Work No. 5

Analysis of static calculation results and slab design

Purpose and plan of the session

Perform an analysis of the static calculation results. Display the isopoles of displacements and stresses from each load on the screen. Create tables of calculation results for the DCF.

Select reinforcement for the plate elements of the calculation scheme. Review the results of the reinforcement selection on the screen using the appropriate commands. Create tables with the results of the selected reinforcement. Analyze the results for different design options.

Review and analysis of static calculation results

Static calculation results are reviewed and analyzed on the **Results** tab. By default, in the results review mode, the calculation diagram is displayed taking into account node displacements (Fig. 70).

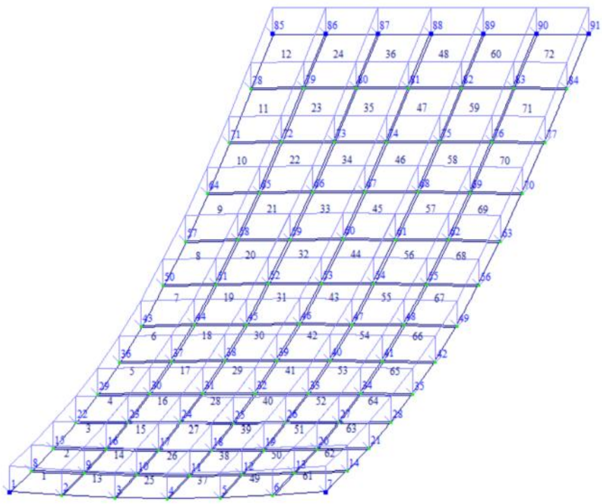


Fig. 70 – Calculation diagram of the slab taking into account the displacement of nodes

To display the diagram without numbering elements and nodes, applied loads in the **Display** dialog box with the **Elements** tab active, you need to uncheck the **Element numbers** option, the **Node numbers** option on the **Nodes** tab, and the **Loads** option on the **General** tab.

Displaying displacement isopoles

To display displacement isopoles in the Z direction (Fig. 71), select the command  – Displacement isopoles in the global system, click the button  – Displacement isopoles along Z (**Deformations** panel on the **Results** tab).

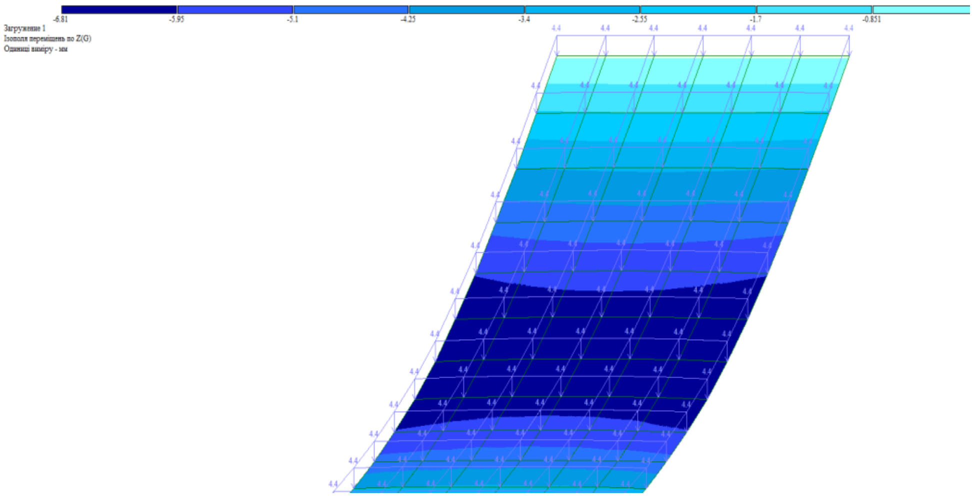
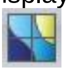


Fig. 71 – Isopole of displacements along Z

Displaying stress isopoles

To display the stress isopoles along Mu (Fig. 72), select the command  – Stress isopoles, then click the button  – Stress isopoles along Mu (**Stress in plates and solid FE** panel on the **Analysis** tab).

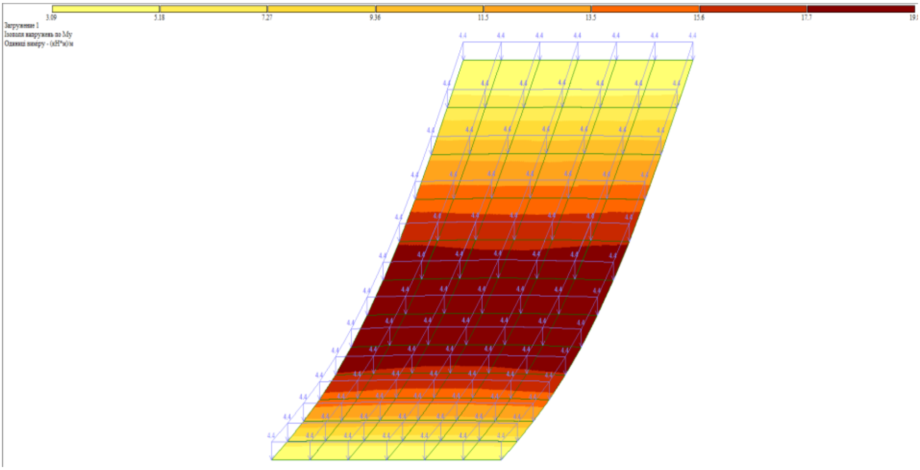


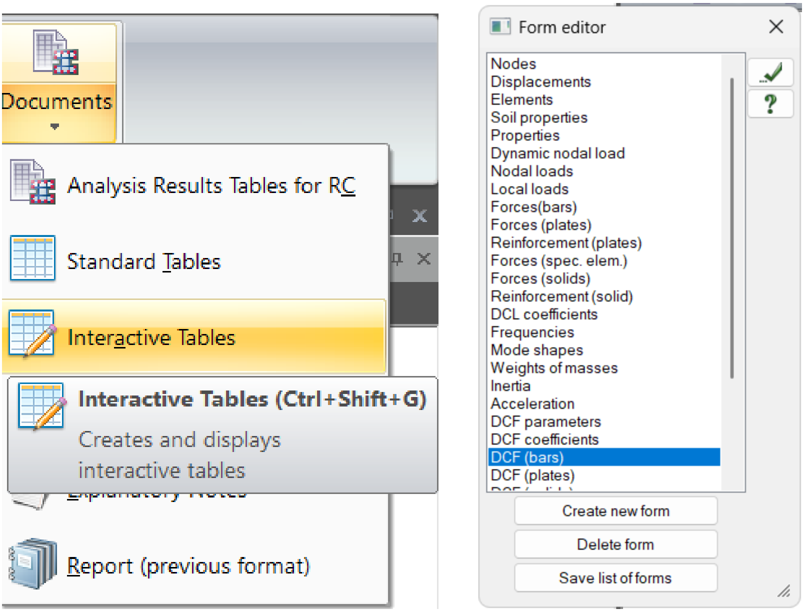
Fig. 72 – Stress isofields along Mu

Changing the current load number

In the status bar (by default, it is located in the lower area of the working window), select load No. 2 in the Change load number list, then No. 3, analyzing the calculation results for each.

Generating and viewing tables of static calculation results

To display tables with the values of calculated force combinations in the elements of the diagram, select the command – Interactive Tables from the drop-down list **Documentation** (**Tables** panel on the **Analysis** tab) – Fig. 73, a. In the **Form Editor** dialog box (Fig. 73, b), select the RCS (bars) row and click the – Confirm button.



a) b)

Fig. 73: a – command to open the Interactive Tables dialog box; b – Forms Editor dialog box

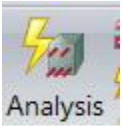
The program will generate tables with the results of static slab calculation – design load combinations (DLC) in FE (finite elements) of the calculation scheme (Fig. 74).

РСЗ (пластини)

№ елем	№ стовпця	Кран/сейсм	Група РСЗ	Критерій	Зусилля (напруження) Nx (кПа)	Ny (кПа)	Nz (кПа)	Txy (кПа)	Txz (кПа)	Mx (кН)	My (кН)	Mxy (кН)	Qx (кН/м)	Qy (кН/м)	Rz (кПа)	№№ завант
18	2	-	A1	1	0.000	0.000	0.000	0.000	0.000	1.441	25.660	0.141	0.588	1.238	0.000	1 2 3
19	2	-	A1	1	0.000	0.000	0.000	0.000	0.000	1.441	25.660	- 0.141	0.588	- 1.238	0.000	1 2 3
30	2	-	A1	1	0.000	0.000	0.000	0.000	0.000	1.821	25.512	0.046	0.194	1.247	0.000	1 2 3
31	2	-	A1	1	0.000	0.000	0.000	0.000	0.000	1.821	25.512	- 0.046	0.194	- 1.247	0.000	1 2 3
42	2	-	A1	1	0.000	0.000	0.000	0.000	0.000	1.821	25.512	- 0.046	- 0.194	1.247	0.000	1 2 3
43	2	-	A1	1	0.000	0.000	0.000	0.000	0.000	1.821	25.512	0.046	- 0.194	- 1.247	0.000	1 2 3
54	2	-	A1	1	0.000	0.000	0.000	0.000	0.000	1.441	25.660	- 0.141	- 0.588	1.238	0.000	1 2 3
55	2	-	A1	1	0.000	0.000	0.000	0.000	0.000	1.441	25.660	0.141	- 0.588	- 1.238	0.000	1 2 3

Fig. 74 – Table of DCF in slab elements

Slab design
Review and analysis of reinforcement calculation results

After calculating the task, review and analyze the reinforcement results on the **Design** tab. To select reinforcement in the diagram elements, click the  button on the Calculate panel.

To view information about the selected reinforcement in one of the plate elements, click the  – Node or Element Information button on the **Selection Panel** toolbar and point the cursor at any plate element. In the dialog box that appears, go to the **Longitudinal reinforcement** tab (this window contains complete information about the selected element, including the results of reinforcement selection). Close the dialog box by clicking on the button  – Close.

Viewing selected reinforcement

To view the area of the bottom reinforcement in the plates in the X1 axis direction, click the button  – Bottom reinforcement in plates along the X1 axis (Plate **Reinforcement** panel on the **Design** tab) – Fig. 75.

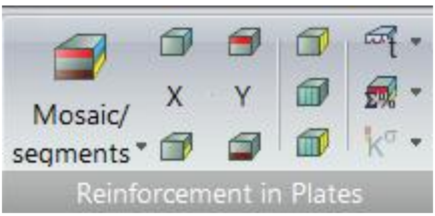


Fig. 75 – Plate reinforcement panel

To view the area of the lower reinforcement in the plates in the Y1 axis direction, press the button  – Lower reinforcement in plates along the Y1 axis (Fig. 76).

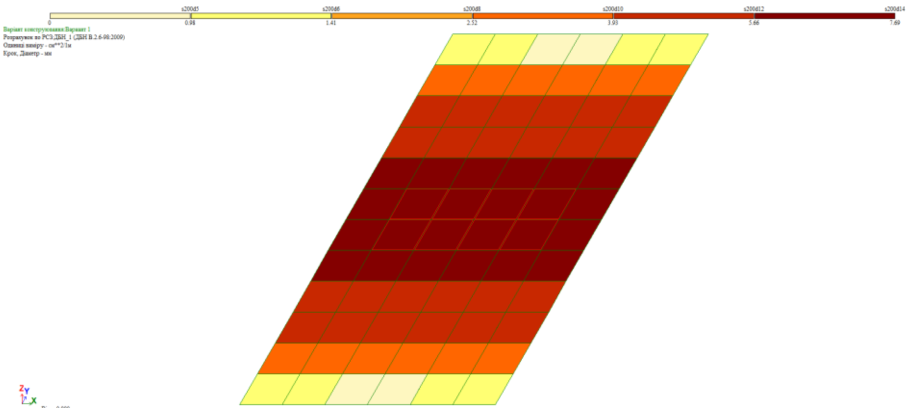
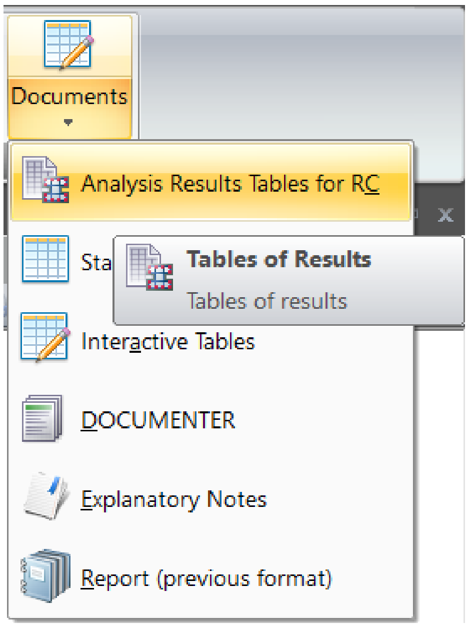


Fig. 76 – Bottom reinforcement in plates along the Y1 axis

Forming and viewing tables of reinforcement selection



From the drop-down list **Documentation** (Fig. 77), select the command **Tables** of results (Tables panel, **Design** tab), call the dialog box **Tables of results** (Fig. 78). In this window, in the Elements field, click the Plates button.

Fig. 77 – Calling the Table of Results command for RC

Changing the design option number

To view and analyze the results for other design options, open the Design Options dialog box by clicking the Design Options button  (**Design** tab on the **Design** panel). To switch to another design option, select the corresponding line in the List of Design Options for the Schematic in the **Status Bar**.

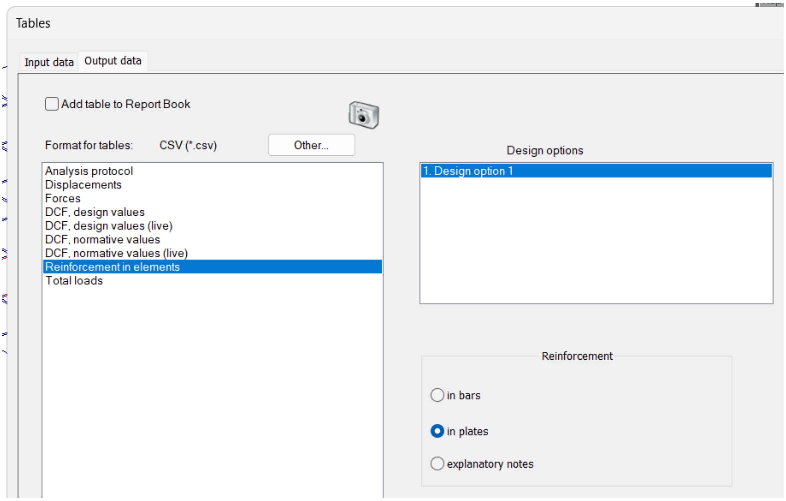
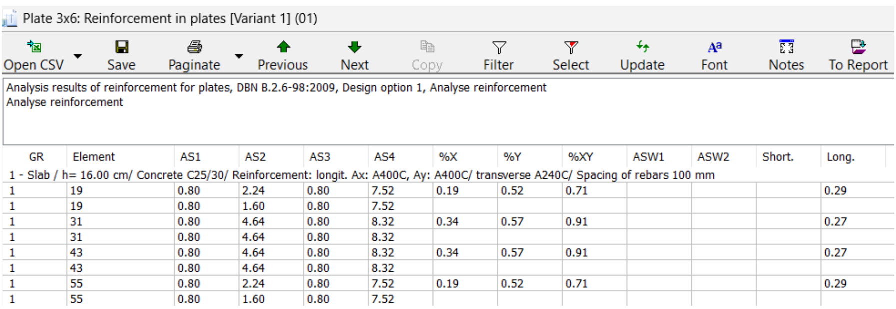


Fig. 78 – Tables dialog box

By default, the radio button is enabled for all items in the Create Table field (if necessary, you can select the items that interest you and enable the radio button for the selected items). When you click the Table button, the command to add the created table to the Report Book will be executed on the screen. Fig. 79 shows a table with the results of the selected reinforcement in the plates (for the first design option).



Analysis results of reinforcement for plates, DBN B.2.6-98:2009, Design option 1, Analyse reinforcement
Analyse reinforcement

GR	Element	AS1	AS2	AS3	AS4	%X	%Y	%XY	ASW1	ASW2	Short.	Long.
1 - Slab / h= 16.00 cm/ Concrete C25/30/ Reinforcement: longit. Ax: A400C, Ay: A400C/ transverse A240C/ Spacing of rebars 100 mm												
1	19	0.80	2.24	0.80	7.52	0.19	0.52	0.71				0.29
1	19	0.80	1.60	0.80	7.52							
1	31	0.80	4.64	0.80	8.32	0.34	0.57	0.91				0.27
1	31	0.80	4.64	0.80	8.32							
1	43	0.80	4.64	0.80	8.32	0.34	0.57	0.91				0.27
1	43	0.80	4.64	0.80	8.32							
1	55	0.80	2.24	0.80	7.52	0.19	0.52	0.71				0.29
1	55	0.80	1.60	0.80	7.52							

Fig. 79 – Table of reinforcement results in plates

Topic 4. Calculation of a reinforced concrete frame of a multi-story building. Design of columns and beams

Task. Perform calculation of transverse reinforced concrete frame (Fig. 80) for static loads (Figs. 82-85). Perform the calculation for the following loads:

- first – from the dead weight of the structures;
- second – snow (evenly distributed over the roof) – q_2= 8.48 kN/m;
- third – from the weight of equipment (evenly distributed on the floors in the extreme spans) – q_3= 30 kN/m;
- fourth – from the weight of equipment (evenly distributed on the floors in the middle spans) – q_4= 30 kN/m;
- fifth – wind from the right – P_1 = 2.88 kN/m; P_2 = 3.3 kN/m.

Each load should be formed separately.

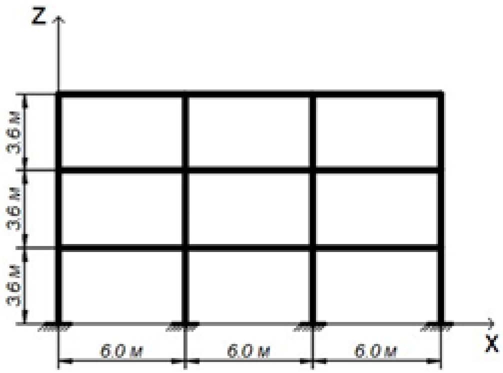


Fig. 80 – Frame diagram

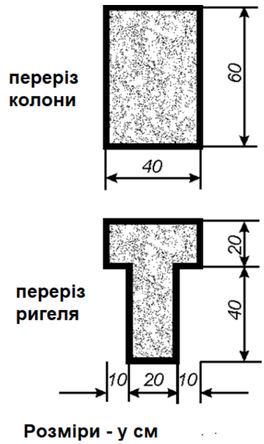


Fig. 81 – Cross-sections of frame elements

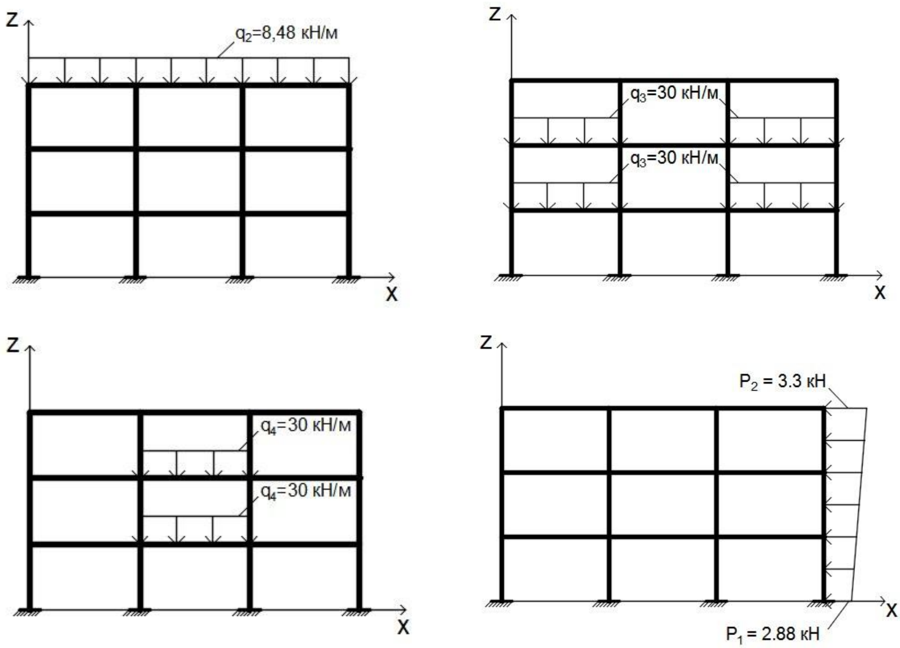


Fig. 82-85 – Frame element load diagrams

Laboratory Work No. 6

Creating a calculation model

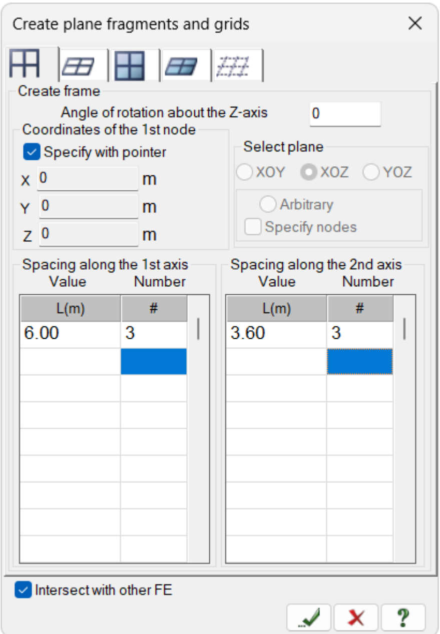
Purpose and plan of the session

Creating a new task. Generating a geometric diagram of the frame. Setting boundary conditions. Selecting a design option. Setting stiffness parameters and material parameters for the elements of the diagram. Forming loads on the frame elements. Generating a table of calculated force combinations. Setting calculated cross-sections for the beams. Complete calculation of the frame.

Creating a new task

On the Quick Access Toolbar, click the – New button. In the **Diagram Description** dialog box, enter the name of the task: “Flat Frame,” and in the **Diagram Type** list, select the line 2 – Three degrees of freedom in a node (X, Z, Uy displacement). Click the – Confirm button.

Creating a geometric frame diagram



On the **Creation** panel (**Creation and Editing** tab), select the – **Frame Generation** button and **Frame Generation** in the drop-down list.

The **Create Flat Fragments and Meshes** dialog box appears (Fig. 86), in which you can set the frame parameters:

- step along the first axis (X) – L= 6 m;
- number – N=3;
- step along the second axis (Z) – L= 3.6 m;
- number – N= 3.

Fig. 86 – Create plane fragments and grids dialog box

After clicking the Apply button , the created frame will appear (Fig. 87).

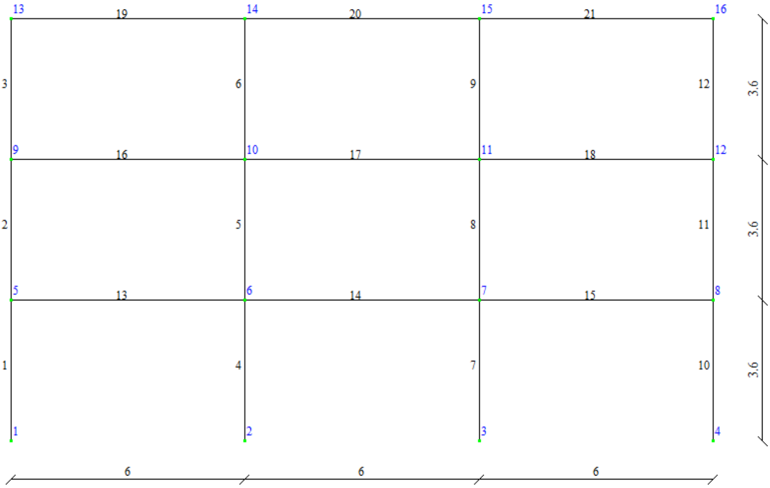
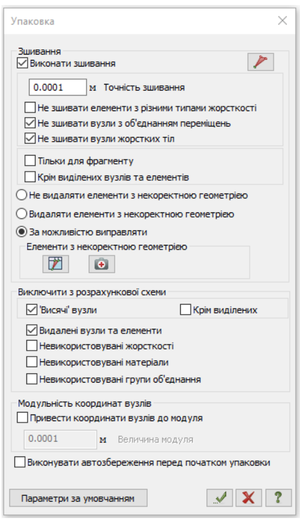
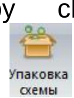


Fig. 87 – Geometric diagram of the frame with dimensions, numbering of nodes and elements



After creating a diagram (especially if it has been edited by adding or deleting nodes and/or elements), it is recommended to perform repackaging by clicking the Repackage Diagram button 

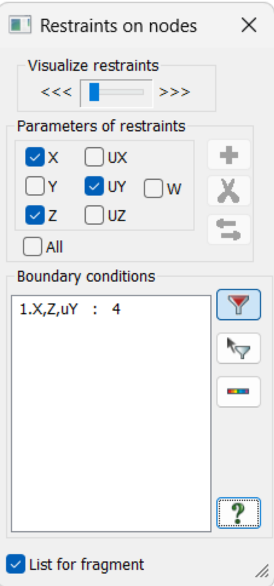
on the **Edit** panel.

In the **Pack** dialog box (Fig. 88), click the Apply button , and the program will renumber the nodes and elements of the diagram.

You can pack the diagram multiple times at any stage of creating and editing the calculation diagram.

Fig. 88 – Packaging dialog box

Setting boundary conditions



To mark nodes №№1-4 (nodes connecting columns to foundations), click the button  – Mark nodes on the **Selection Panel** (by default, it is located in the lower area of the working window). The nodes will be colored red. Next, click the command button  – Restraints (**Rigidity and Restraints** panel on the Create and Edit tab), and the Restraints in Nodes dialog box will appear (Fig. 89).

In this window, activate the constraint parameters, i.e., the directions in which node movement is prohibited (X, Z, UY), and click the Add Constraints to Selected Nodes button . The nodes will turn blue, which means that the boundary conditions (constraints) have been set.

Fig. 89 – Constraints at Nodes dialog box

Selecting a calculation and design option

For structural calculations in accordance with current standards, click the button  – Design options and select the required standard in the **Design options** dialog box (Fig. 90) – DBN V.2.6-98:2009. In the same window, specify the calculation option (by individual forces or by design load combinations – DCL (or forces – DCF). In this task, it is necessary to perform calculations for several loads, so select DCF.

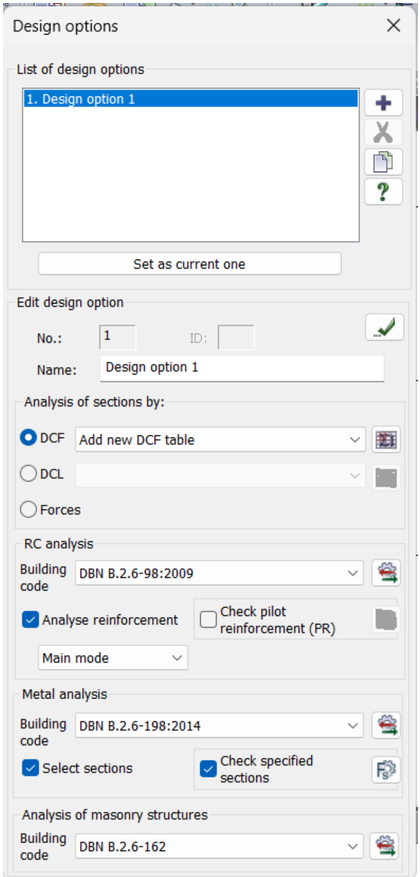
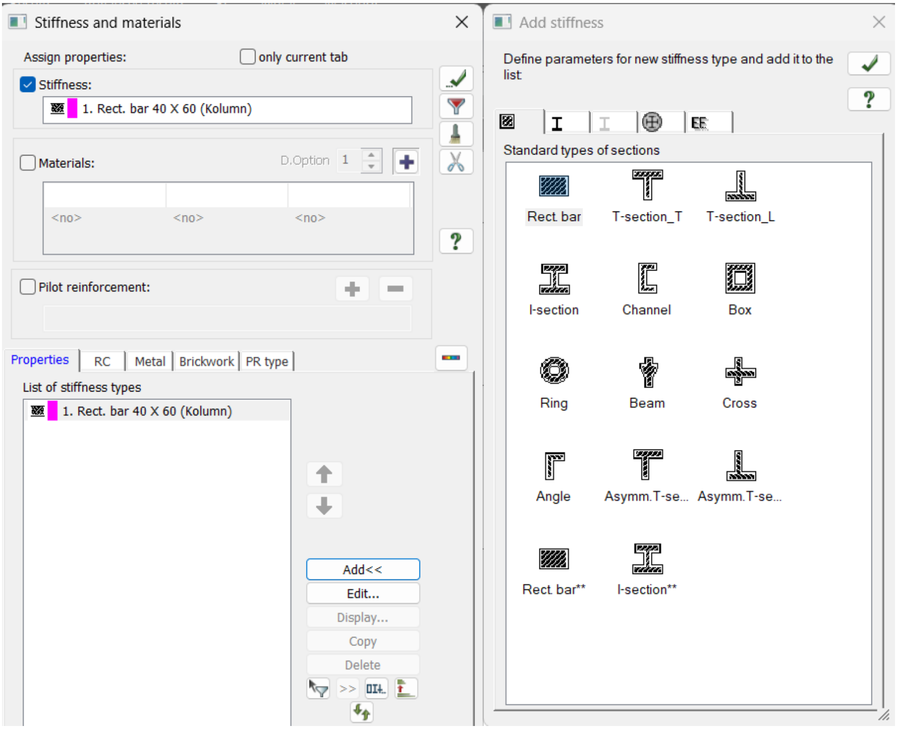


Fig. 90 – Design Options dialog box

Setting stiffness parameters and material parameters for diagram elements

Click the Stiffness and Materials button  (**Stiffness and Connections** panel on the **Create and Edit** tab) to open the **Stiffness and Materials** dialog box (Fig. 91).

In this window, click the **Add** button and in the **Add Stiffness** dialog box, select the first tab – **Standard Cross-Sections**, activate the **Beam** cross-section.



a *b*

Fig. 91 – Dialog boxes: a – Stiffnesses and materials,
b – Add stiffness

In the Standard Cross-Section Settings dialog box (Fig. 92), enter the following data for the column:

- modulus of elasticity of concrete class C16/20 – E = 2.7e7 kN/m^2 (for English keyboard layout);

- geometric dimensions – W = 40 cm; H = 60 cm;

- specific weight of the material - Ro = 25 kN/m^2.

To see a sketch of the cross-section being created with all dimensions, click the Draw button.

Press the Apply button  to confirm the data entry.

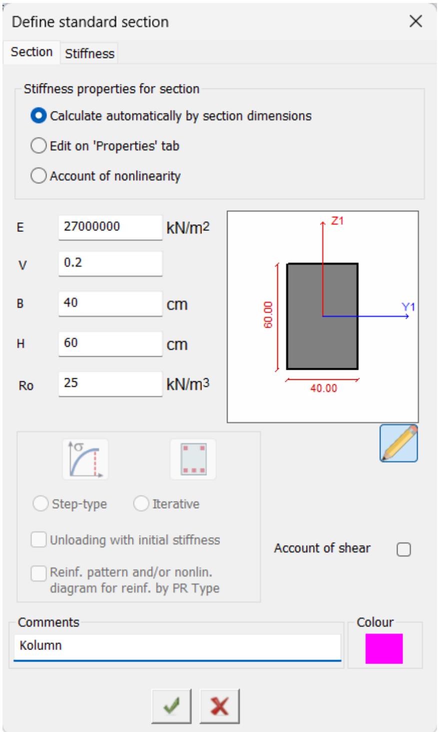


Fig. 92 – Dialog box for specifying a standard cross-section (rectangle)

Next, in the **Stiffness and Materials** dialog box, go to the **Reinforced Concrete** tab to set the parameters for reinforced concrete columns. To do this, select the **Type** radio button, then click the Edit button. This will bring up the **Materials for Reinforced Concrete Structures Calculation** dialog box.

In the **Materials for reinforced concrete structures calculation** dialog box (Fig. 93), specify the following parameters for columns:

- in the Bar field:
- name – Column;
- reinforcement – symmetrical (since the column is centrally compressed);
- select corner bars;
- design length – specify the coefficient – 0.7;
- in the Concrete field – select class C16/20 from the list;
- in the Reinforcement field – select class A400C for longitudinal reinforcement and A240C for transverse reinforcement from the list.

Press the Apply button  to confirm the data entry.

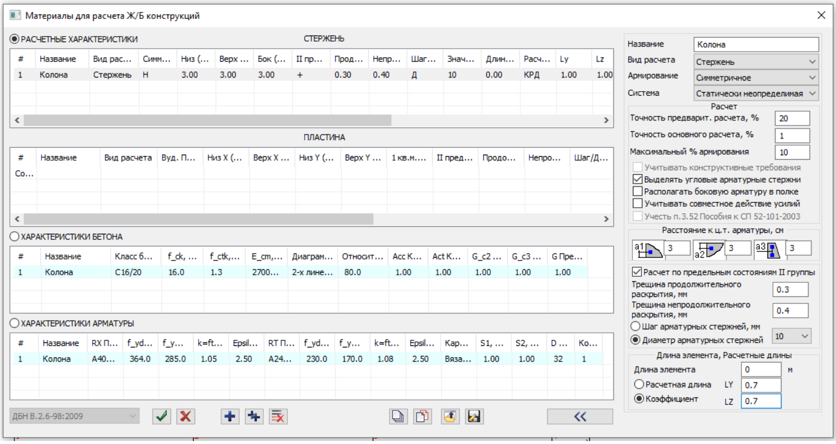
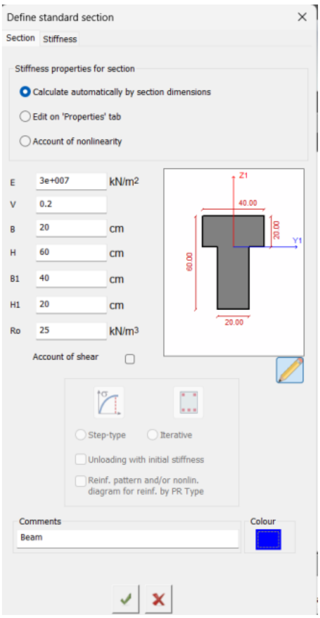


Fig. 93 – Materials dialog box for calculating reinforced concrete structures (for columns)



To assign stiffnesses and materials to columns, mark all vertical frame elements. The latter will be colored red. In the **Stiffnesses and Materials** dialog box, click the Apply button . The selection is removed from the elements, which means that they are assigned the first stiffness type – 1. 40x40 beam (columns).

To set the second type of stiffness (for beams), activate the Tavr_T cross-section in the **Add Stiffness** dialog box. In the Set Standard Cross-Section dialog box (Fig. 94), enter:

- elastic modulus –
$E = 3e7$ kN/m^2;
- geometric dimensions –
B = 20 cm; H = 60 cm;
B1 = 40 cm; H1 = 20 cm;
- density - Ro = 25 kN/m^2.

Fig. 94 – Dialog box Setting the standard cross-section (tavr)

To see a sketch of the cross-section being created with all dimensions, click the Draw button.

Press the Apply button  to confirm the data entry.

To create stiffnesses and materials for beams, double-click on the second type of stiffness (tavr), assigning it as current (Fig. 95). On the Z/b tab, click the Edit button.

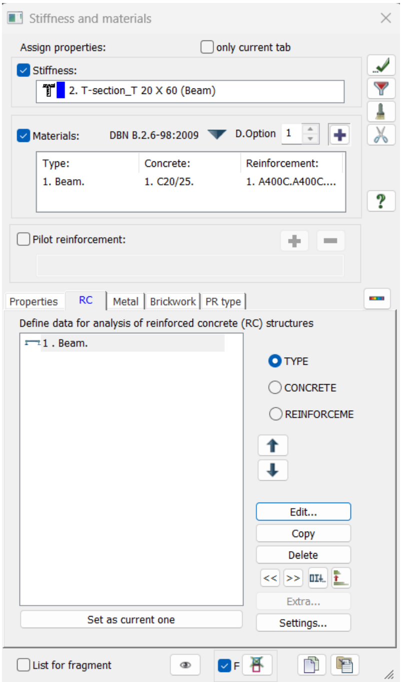


Fig. 95 – Created list of stiffnesses for frame elements

In the **Materials for reinforced concrete structures** calculation dialog box (Fig. 96), click the Add button  to create parameters for beams:

• In the Bar field:

- Name – beam;

- reinforcement – asymmetric;

- do not select corner members;

- design length coefficient – 0;

• In the Concrete field, select class C20/25 from the list.

• In the Reinforcement field, select longitudinal class A400C and transverse class A240C from the list.

Click the Apply button  to confirm the data entry.

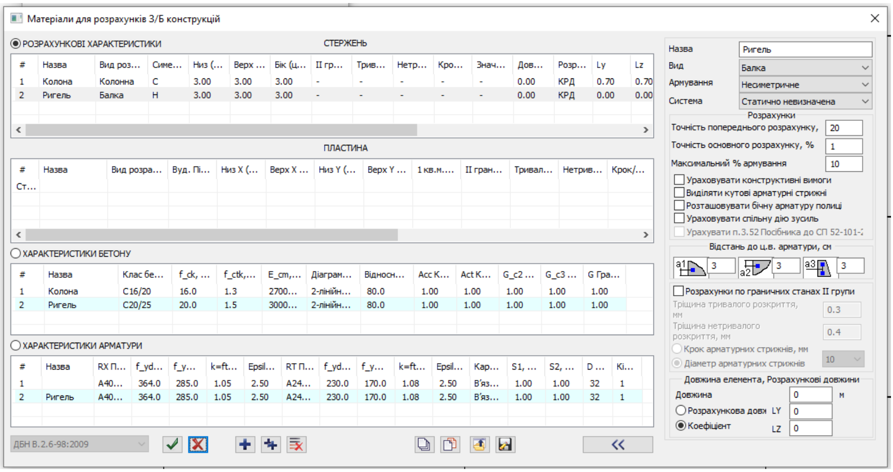


Fig. 96 – Materials dialog box for calculating reinforced concrete structures (for beams)

Select horizontal elements (beams). In the **Stiffness and Materials** dialog box, the second stiffness type should be current – Tavr¬¬_T 20x60 (beams) with the appropriate materials for construction. Click the Apply button  .

Forming loads

According to the task, you need to calculate the frame for five loads. For convenience, it is recommended to display the element numbers using the button  button to open the **Display** dialog box, and check the Element numbers command on the Elements tab.

Creating load No. 1

To set the load from the frame elements' own weight, click the  – Add own weight button (**Load** panel on the **Create and edit** tab). The Add Own Weight dialog box will appear, in which you click the  – Apply button (depending on the specified volumetric weight, the elements of the calculation scheme are loaded with the load from their own weight) – Fig. 97.

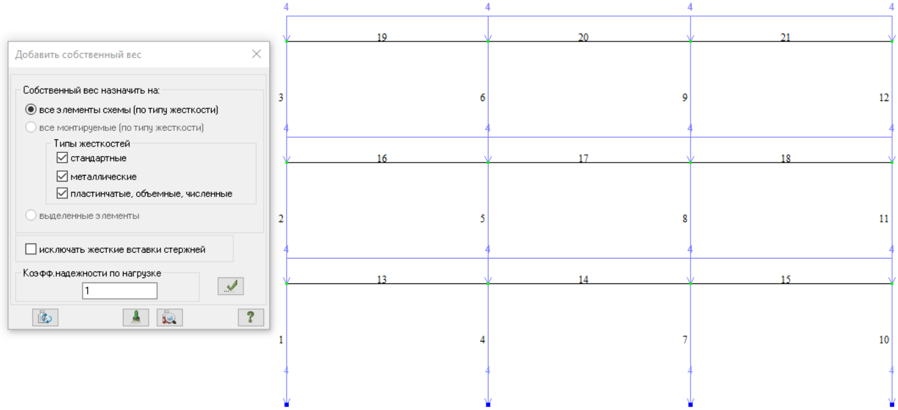


Fig. 97 – Load 1 – dead weight (constant)

Creating load No. 2

According to the task, the second load is variable short-term (snow) with an intensity of 8.48 kN/m. Change the number of the current load by clicking the Next Load button  in the status bar (in the lower area of the working window).

On the **Load** panel (**Create and Edit** tab), select the **Load** command. From the drop-down list, select the **Load on Bars** command (Fig. 98).

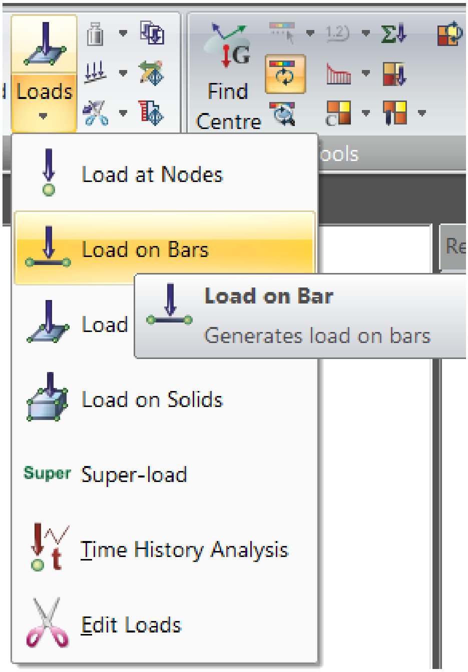
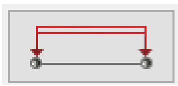


Fig. 98 – Load on bars command in the Load drop-down list

In the **Load Assignment** dialog box (Fig. 99), specify the coordinate system – Global, the direction of load action – along the Z axis.

To specify a linear load uniformly distributed along the length of the beam, click the command button , the **Parameters** window appears, in which you specify the load intensity – P = 8.48 kN/m (Fig. 99) and click the button – Confirm.

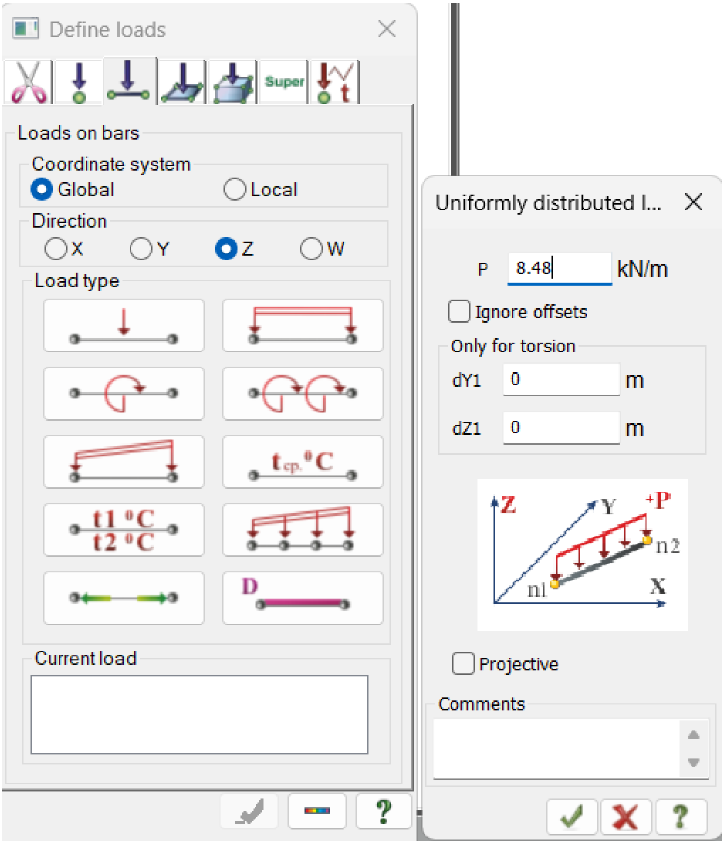


Fig. 99 – Load Assignment and Parameters dialog boxes

Use the button  to select horizontal elements Nos. 19-21 (roof purlins). In the **Load Assignment** dialog box, click the button . A snow load with an intensity of P=8.48 kN/m will be applied to the frame (Fig. 100).

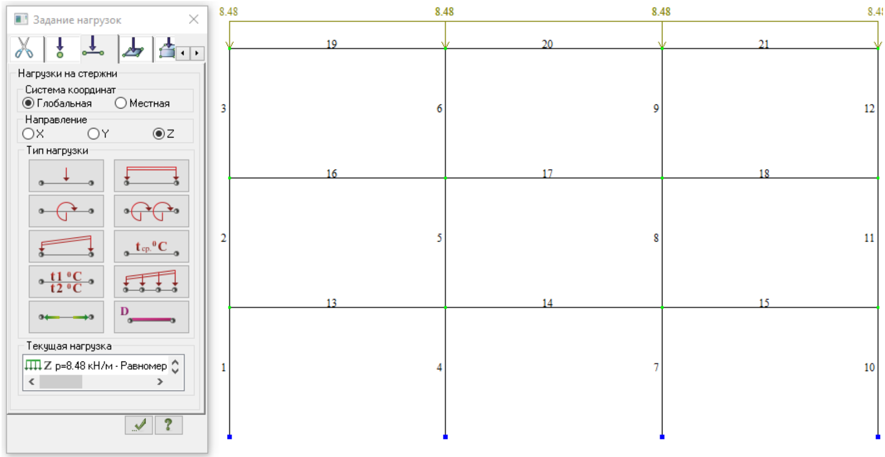
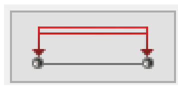


Fig. 100 – Load 2 – snow (short-term)

Forming loads No. 3 and 4

According to the task, the third and fourth loads are variable quasi-constant loads depending on the weight of the equipment, with an intensity of P=30 kN/m. Change the number of the current load by clicking on the button  – Next load in the status bar (in the lower area of the working window).

On the **Load** panel (**Creation and Editing** tab), select the **Load** command, and from the drop-down list, select the **Load on Bars** command.

To specify the linear load, click the command button  specify the load intensity in the **Parameters** window – P=30 kN/m, and click the button  – Confirm.

Use the button  to select horizontal elements Nos. 13, 15, 16, and 18 (for the third load). In the **Load Assignment** dialog box, click the button . A long-term load from the weight of equipment with an intensity of P=30 kN/m will be applied to the beams of the extreme spans of the 2nd and 3rd floors of the frame (Fig. 101).

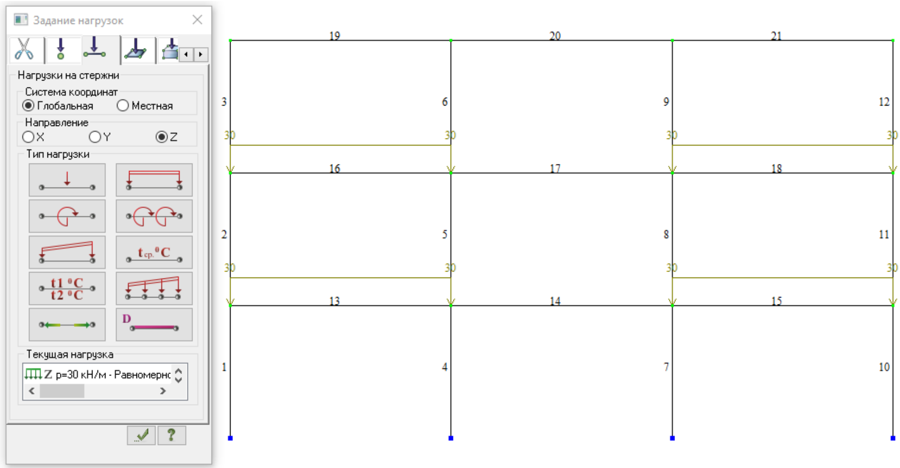


Fig. 101 – Load 3 – from equipment (long-term)

For the fourth load, repeat the steps described above, but apply the load to elements Nos. 14 and 17 (Fig. 102).

Formation of load No. 5

The fifth load is a variable short-term (wind) load with an intensity of 2.88 and 3.3 kN/m. Change the number of the current load by clicking on the button  – Next load in the status bar (in the lower area of the working window).

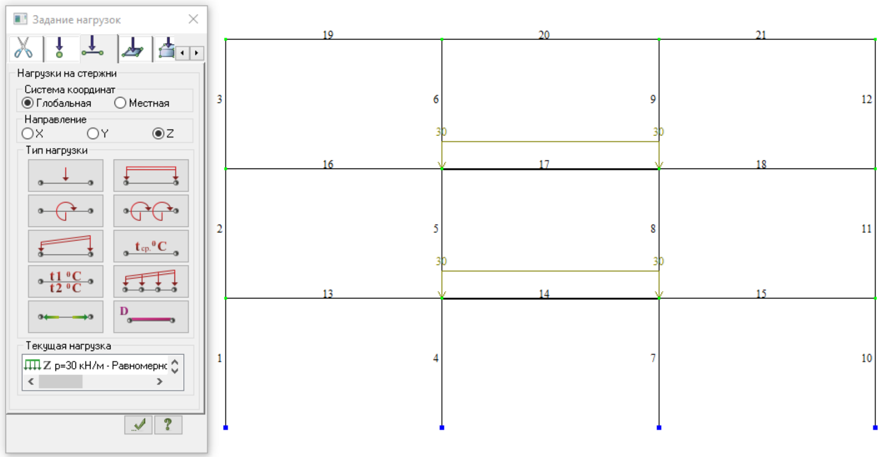
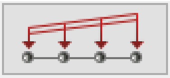


Fig. 102 – Load 4 – from equipment (long-term)

On the **Load** panel (**Create and Edit** tab), select the **Load** command, and from the drop-down list, select the **Load on Rods** command.

In the **Load Assignment** dialog box, specify the coordinate system – Global, the direction of load action – along the X axis. To create a load, click the command button  – On a group of elements, the **Uneven Load** window appears, in which you specify with a radio button along which axis (Z) the load (wind pressure) will increase and the load intensity – P1=2.88 kN/m, P2=3.3 kN/m and click the button  – Confirm.

Use the button  to select vertical elements Nos. 10-12 (columns on the far right axis for the entire height of the frame). In the **Load Assignment** dialog box, click the button  . A wind load will be applied to the frame on the right (Fig. 103).

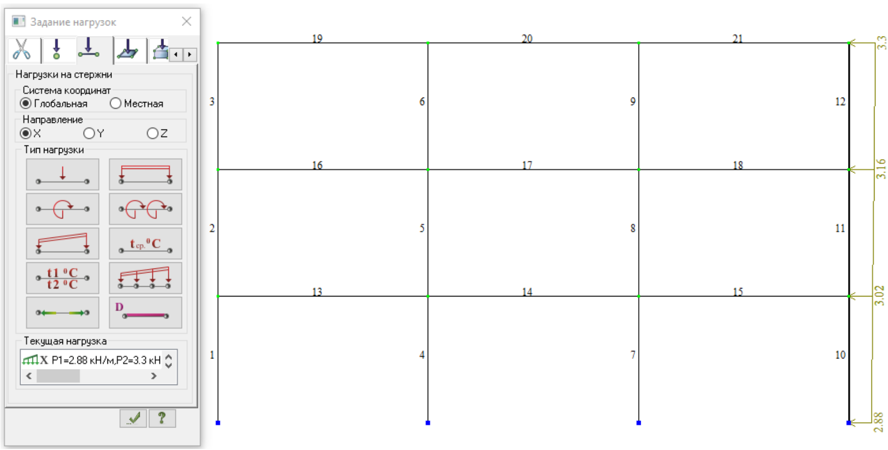
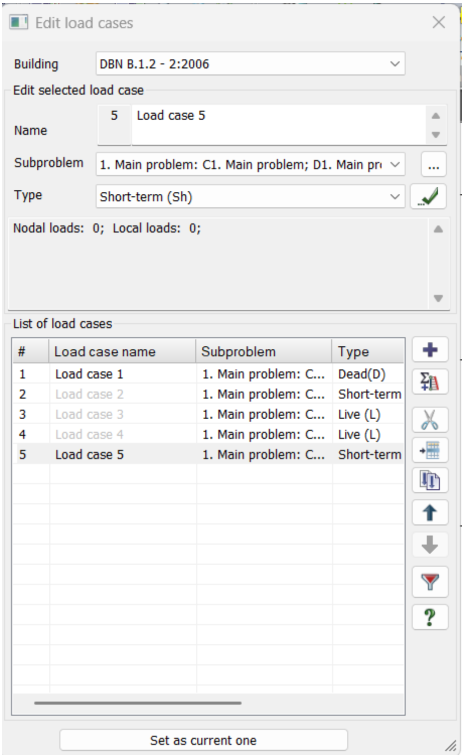
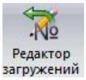


Fig. 103 – Load 5 – wind from the right (short-term)

Request for additional information about the load



To specify additional information about the load, use the button 

on the **Load** panel. The **Download Editor** dialog box will appear (Fig. 104), in which, for download 1, in the **Edit selected download field**, specify the name – Own weight, select Constant from the Type list, and click the button .

Fig. 104 – Download Editor dialog box

For download 2, in the **Edit Selected Download** field, enter the name Snow, select Short-term from the **Type** list, and click the button .

For downloads 3 and 4, enter the name Equipment Weight in the **Edit Selected Download** field, select Long Term from the Type list, and click .

To load 5, in the **Edit Selected Load** field, enter the name – Wind from the right, select Short-term from the **View** list, and click the button .

You can also enter additional information about the load before generating the loads.

Generating the RSD table

Since several loads of different durations have been specified, it is necessary to create an DCF table for further calculation of reinforcement for the most dangerous combinations of forces.

The dialog box of the same name is called up by clicking on the button  – DCF table (DCF panel on the **Calculation** tab) – Fig. 105.

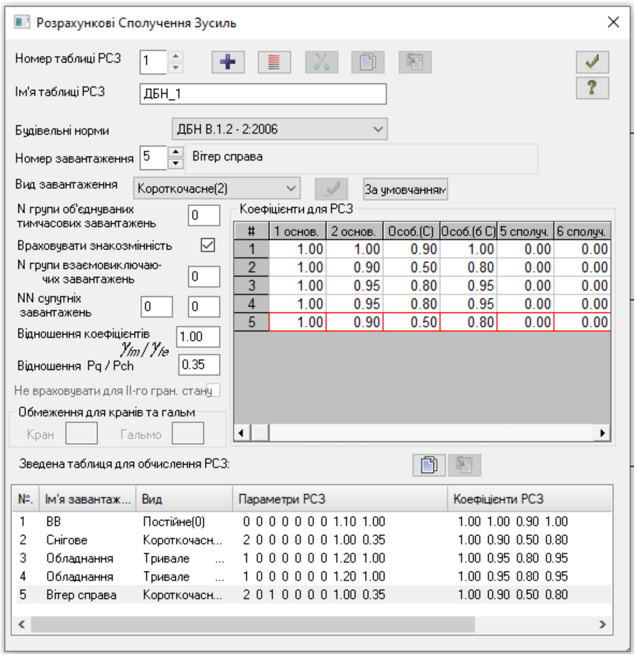


Fig. 105 – Calculated force combinations dialog box

In this window, for the selected building codes DBN V.1.2-2:2006, you need to specify the relevant data for each load.

For loads 3 and 4, enter 1 in the text field No. of mutually exclusive loads, and for wind load, check the box Consider sign change. Save the table by clicking the button .

Specifying design cross-sections for beams

Mark all horizontal elements on the diagram. On the Members context tab, on the Edit Members panel, click the button  – Design cross-sections of members. The **Design Cross-Sections** dialog box appears (Fig. 106), in which you specify the number of design cross-sections N = 5 (to design a bending element, you need to calculate the forces in three or more cross-sections). Click the  – Apply button.

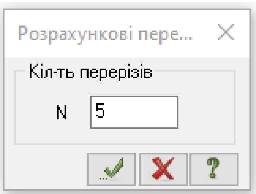
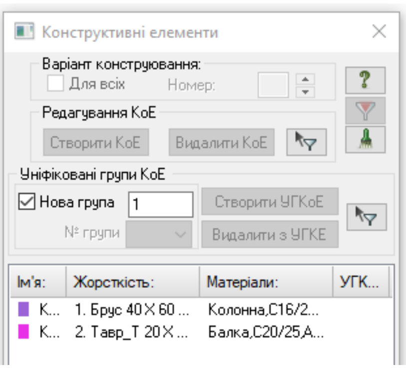


Fig. 106 – Calculated Cross-Sections dialog box



Assigning structural elements

To create a **BEAM** structural element, select the horizontal elements of the beams, for example, on the top floor.

Fig. 107 – Structural Elements dialog box

Open the **Structural Elements** dialog box by clicking the Structural Elements button  on the **Beams** context tab.

In the dialog box, in the **Edit CoE** field, click the Create button (the BEAM structural element is used to indicate that this is a continuous beam).

To create a **COLUMN** structural element, select vertical elements to a height of three floors, for example, along the axis on the left. In the **Structural Elements** dialog box, in the **Edit CoE** field, click the Create button (the COLUMN structural element is used to indicate that this is a continuous column).

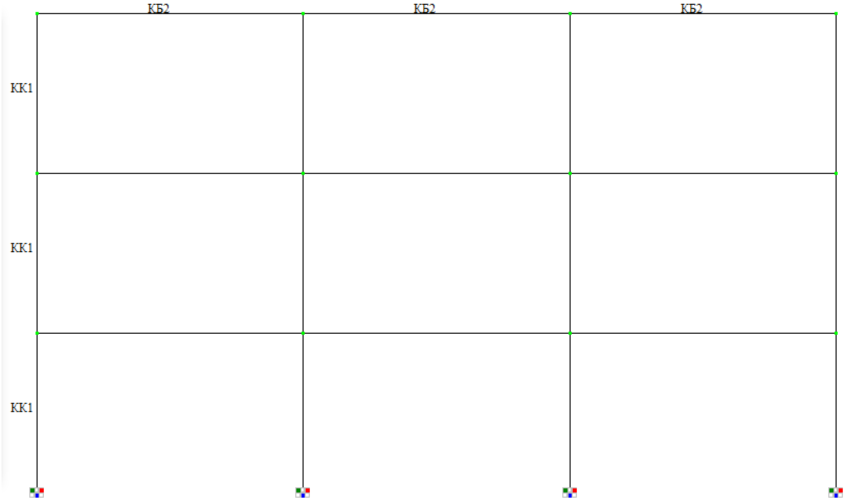


Fig. 108 – Frame with created structural elements

Complete frame calculation

Before starting the calculation, you need to save the task (button  on the quick access panel). To calculate, click on the button  – Perform calculation.

Laboratory Work No. 7

Analysis of static calculation results and frame element design

Purpose and plan of the session

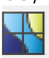
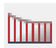
Display the displacement and force diagrams in the frame elements for each load. Analyze the results of static calculation. Create PCS tables. Analyze the selected reinforcement for vertical and horizontal elements of the calculation scheme (columns and beams). Create tables of results for reinforced concrete elements. Design columns and beams.

Review and analysis of static calculation results

After calculating the problem, review and analyze the static calculation results on the **Analysis** tab.

In the calculation results review mode, the calculation diagram is displayed by default, taking into account node displacements. To display the diagram without taking into account node displacements, click the – Initial Diagram button  (**Deformations** panel on the **Analysis** tab).

Displaying internal force diagrams on the screen

To display the longitudinal force diagram in the frame elements N, click the button  – Longitudinal force diagrams N (**Force in members** panel on the **Analysis** tab). To shade the diagrams and display the force values, use the button  button to open the **Display** dialog box. On the tab  – Values, check the boxes next to the commands  – Show shaded diagrams and  – Values on diagrams.

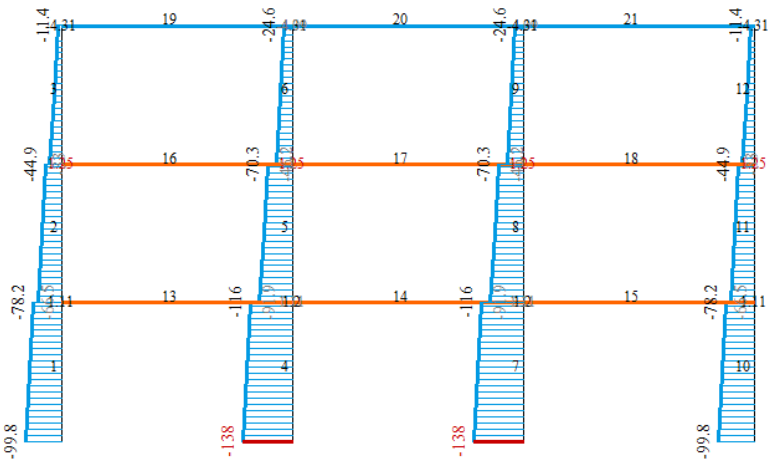


Fig. 109 – Diagram of longitudinal forces in frame elements (dead load)

To display the My diagrams, click the button My – My bending moment diagrams (**Force in members** panel on the **Analysis** tab) – Fig. 110.

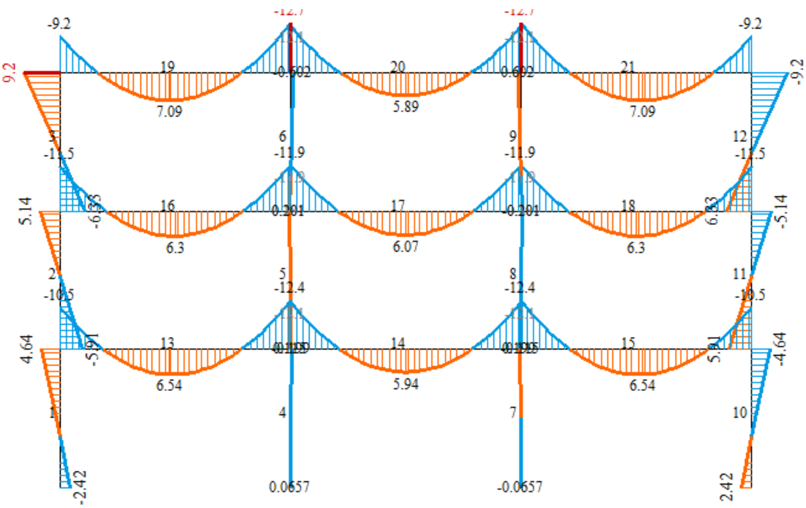


Fig. 110 – Diagram of bending moments in frame elements (loading with own weight)

To display the Qz diagrams, click the button Qz – Qz shear force diagrams (**Force in members** panel on the **Analysis** tab) – Fig. 111.

Sometimes it is more convenient to analyze the forces in the elements of the calculation scheme in the form of a mosaic (in color). To do this, select the command – **Mosaic of forces in members** in the **Diagrams/mosaic** list (the **Forces in members** panel on the **Analysis** tab) – Fig. 112.

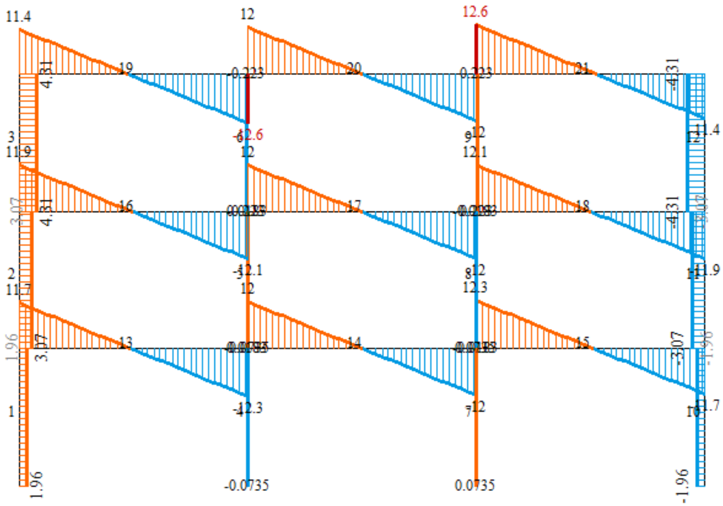


Fig. 111 – Diagram of transverse forces in frame elements (loading by own weight)

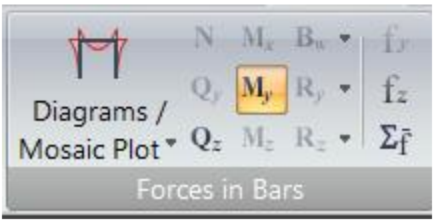


Fig. 112. Forces in Bars panel

Changing the current load number

To view the force diagrams for the next load in the status bar, select the row corresponding to the second load in the **Change load number** list and click the Apply button .

Generating and viewing calculation result tables

To view the force diagrams from the next load in the status bar in the **Change load number** list, select the line corresponding to the second load and click the Apply button  .

Creating and viewing tables of calculation results

To display a table with the values of calculated forces in the elements of the diagram, click the button  – Documentation (**Tables** panel on the **Analysis** tab), select the Interactive Tables command from the **Documentation** drop-down list (Fig. 113).

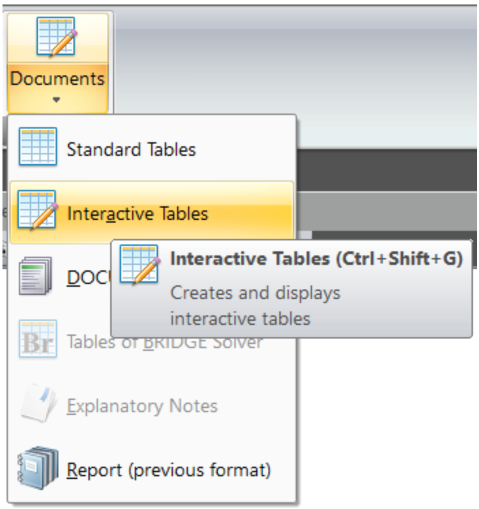


Fig. 113 – Calling the Interactive Tables command

The Form Editor dialog box will appear, in which you select the **DCF bars** ribbon (Fig. 120), and in the **Create DCF Table** window, select the desired option – for all elements or only for selected ones (in which case these elements must be selected in advance). Confirm the operation.

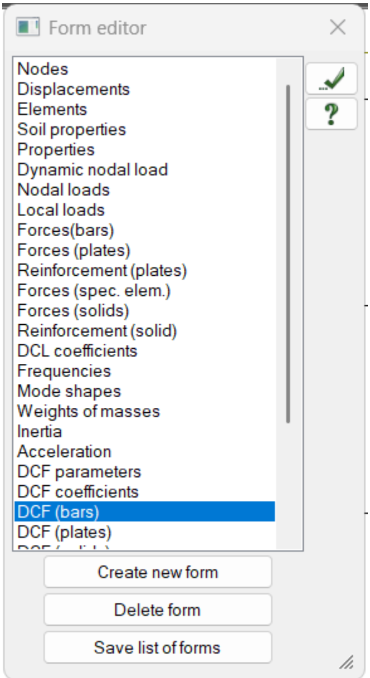


Fig. 114 – Form Editor and Create Table dialog boxes

The program will generate a table of calculated force combinations (Fig. 115).

РСЗ(стержні)

Файл Редагувати Опції

РСЗ(стержні)

№ елем	№ перер	№ стовпця	Кран/сейсм	Група РСЗ	Критерій	Зусилля N (кН)	Mk (кН*м)	My (кН*м)	Qz (кН)	Mz (кН*м)	Qy (кН)	№№ завант
1	1	1	-	A1	1	- 93.171	0.000	12.627	- 4.019	0.000	0.000	1 -5
1	1	2	-	A1	2	- 298.348	0.000	- 32.932	21.150	0.000	0.000	1 2 3 5
1	1	1	-	A1	5	- 106.489	0.000	- 17.460	7.940	0.000	0.000	1 5
1	1	2	-	A1	6	- 286.362	0.000	- 5.854	10.387	0.000	0.000	1 2 3 -5
1	1	2	-	A1	32	- 91.115	0.000	11.254	- 3.650	0.000	0.000	1 4 -5
1	2	2	-	A1	1	- 254.882	0.000	42.835	21.009	0.000	0.000	1 3 5
1	2	2	-	A1	2	- 91.382	0.000	- 1.511	- 3.508	0.000	0.000	1 2 4 -5
1	2	2	-	A1	6	- 276.748	0.000	43.210	21.150	0.000	0.000	1 2 3 5
1	2	1	-	A1	14	- 71.571	0.000	- 1.843	- 4.019	0.000	0.000	1 -5
1	2	2	-	A1	32	- 69.515	0.000	- 1.885	- 3.650	0.000	0.000	1 4 -5
2	1	2	-	A1	2	- 176.631	0.000	- 55.091	31.184	0.000	0.000	1 2 3 5
2	1	2	-	A1	5	- 154.849	0.000	- 55.292	31.849	0.000	0.000	1 3 5
2	1	2	-	A1	6	- 84.922	0.000	0.836	- 2.034	0.000	0.000	1 2 4 -5

Fig. 115 – Table of reinforcement in frame elements (fragment)

Review and analysis of reinforcement selection results

Reinforcement calculation

After calculating the problem, go to the **RC** tab to view and analyze the reinforcement results. To select reinforcement in the diagram elements, click the Calculate button 

(**Analisis** panel on the **RC** tab). In the **Reinforcement calculation** dialog box, click the Perform calculation button.

Viewing reinforcement results

To view information about the selected reinforcement in one of the elements, click the  – Node or Element Information button on the **Selection Panel** toolbar, pointing the cursor at the desired element. In the dialog box that appears, go to the Longitudinal reinforcement tab (this window contains all the information about the selected element, including the reinforcement selection results).

To set the display mode for symmetrical reinforcement in column cross-sections, select the **Symmetry** command from the drop-down list on the **Bar Reinforcement** panel in the **RC** tab (Fig. 48).

The user can customize the reinforcement display mode at their discretion—as a mosaic or reinforcement diagram (Fig. 48).

An algorithm for selecting corner bars was set up for the columns. To view the areas of longitudinal reinforcement in the lower left corner of the AU1 bar cross-section, click the  – AU1 corner reinforcement button.

To view the areas of longitudinal reinforcement in the lower right corner of the AU2 bar cross-section, click the button  – AU2 corner reinforcement.

To set the display mode for asymmetrical reinforcement in beam cross-sections in the form of reinforcement diagrams, select the **Asymmetry** command from the drop-down list on the **Reinforcement** panel in the **Design** tab. Since the distributed reinforcement mode was set for beams, to display the results of the lower reinforcement (in spans), click the Distributed reinforcement AS1 button  (Fig. 116).

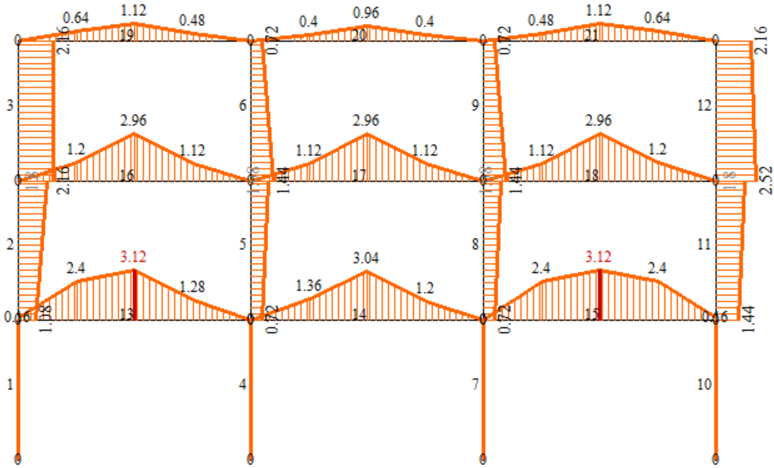


Fig. 116 – Distributed reinforcement AS1 (near the lower edge of the section) in beams

To display the results of the upper reinforcement (on the supports), click the button  – Distributed reinforcement AS2 (Fig. 117).

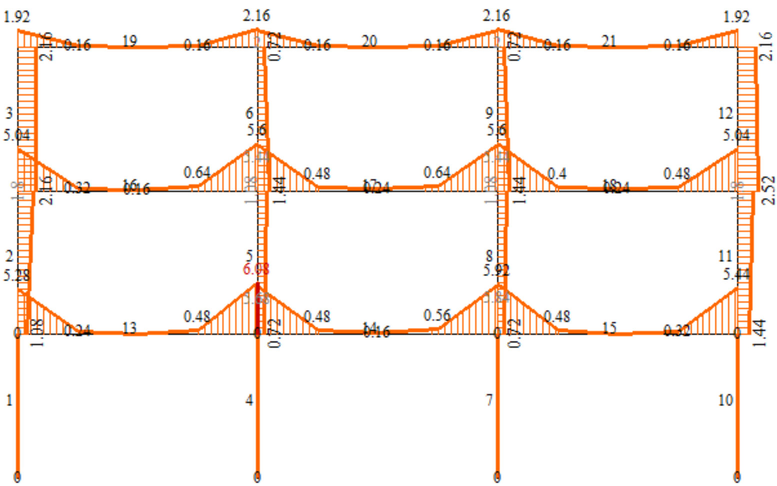
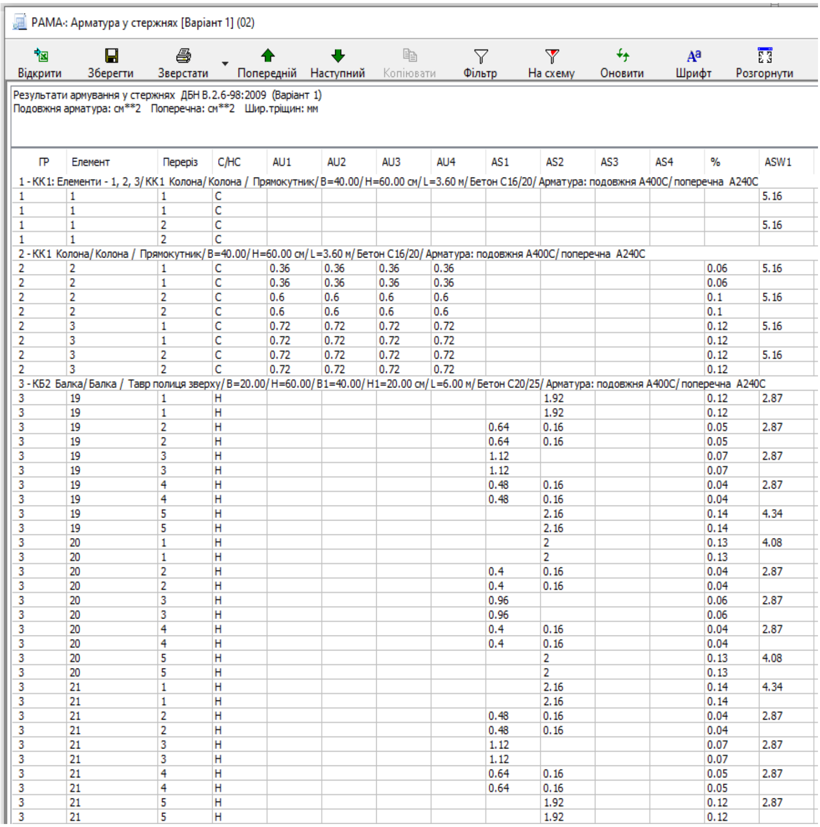


Fig. 117 – Distributed reinforcement AS2 (near the upper edge of the cross-section) in beams

Forming and viewing tables of reinforcement selection results

On the **Design** tab, from the **Tables** panel 

– **Analisis**, activate the **Results tables for reinforced concrete** command from the drop-down list. In the Results tables dialog box, a table with the results of the selected reinforcement is displayed on the screen (Fig. 118).



Результати армування у стержнях ДБН В.2.6-98:2009 (Варіант 1)
Подовжня арматура: см**2 Поперечна: см**2 Шир.тріщин: мм

ГР	Елемент	Переріз	С/НС	AU1	AU2	AU3	AU4	AS1	AS2	AS3	AS4	%	ASW1
1 - КК1: Елементи - 1, 2, 3/ КК1 Колона/ Колона / Прямокутник/ B=40.00/ H=60.00 см/ L=3.60 м/ Бетон C16/20/ Арматура: подовжня A400C/ поперечна A240C													
1	1	1	С										5.16
1	1	1	С										
1	1	2	С										5.16
1	1	2	С										
2 - КК1 Колона/ Колона / Прямокутник/ B=40.00/ H=60.00 см/ L=3.60 м/ Бетон C16/20/ Арматура: подовжня A400C/ поперечна A240C													
2	2	1	С	0.36	0.36	0.36	0.36					0.06	5.16
2	2	1	С	0.36	0.36	0.36	0.36					0.06	
2	2	2	С	0.6	0.6	0.6	0.6					0.1	5.16
2	2	2	С	0.6	0.6	0.6	0.6					0.1	
2	3	1	С	0.72	0.72	0.72	0.72					0.12	5.16
2	3	1	С	0.72	0.72	0.72	0.72					0.12	
2	3	2	С	0.72	0.72	0.72	0.72					0.12	5.16
2	3	2	С	0.72	0.72	0.72	0.72					0.12	
3 - КБ2 Балка/ Балка / Тавр полиця зверху/ B=20.00/ H=60.00/ B1=40.00/ H1=20.00 см/ L=6.00 м/ Бетон C20/25/ Арматура: подовжня A400C/ поперечна A240C													
3	19	1	Н						1.92			0.12	2.87
3	19	1	Н						1.92			0.12	
3	19	2	Н					0.64	0.16			0.05	2.87
3	19	2	Н					0.64	0.16			0.05	
3	19	3	Н					1.12				0.07	2.87
3	19	3	Н					1.12				0.07	
3	19	4	Н					0.48	0.16			0.04	2.87
3	19	4	Н					0.48	0.16			0.04	
3	19	5	Н						2.16			0.14	4.34
3	19	5	Н						2.16			0.14	
3	20	1	Н						2			0.13	4.08
3	20	1	Н						2			0.13	
3	20	2	Н					0.4	0.16			0.04	2.87
3	20	2	Н					0.4	0.16			0.04	
3	20	3	Н					0.96				0.06	2.87
3	20	3	Н					0.96				0.06	
3	20	4	Н					0.4	0.16			0.04	2.87
3	20	4	Н					0.4	0.16			0.04	
3	20	5	Н						2			0.13	4.08
3	20	5	Н						2			0.13	
3	21	1	Н						2.16			0.14	4.34
3	21	1	Н						2.16			0.14	
3	21	2	Н					0.48	0.16			0.04	2.87
3	21	2	Н					0.48	0.16			0.04	
3	21	3	Н					1.12				0.07	2.87
3	21	3	Н					1.12				0.07	
3	21	4	Н					0.64	0.16			0.05	2.87
3	21	4	Н					0.64	0.16			0.05	
3	21	5	Н						1.92			0.12	2.87
3	21	5	Н						1.92			0.12	

Fig. 118 – Table with the results of the selected reinforcement

Topic 5. Design a monolithic beam floor slab. Design a slab and secondary beam

Task. Calculate the static load on a monolithic floor slab. Analyze the calculation results. Select the reinforcement.

The structural diagram is shown in Fig. 119. The floor material is C16/20 concrete, reinforced with separate A400C reinforcement bars.

Perform the calculation for a 27x24m slab (36x12 grid) in the xOy plane for two loads:

- first load – constant uniformly distributed $g = 2.4$ kN/m^2, including dead weight;
- second load – variable (long-term) uniformly distributed $p = 7.5$ kN/m^2.

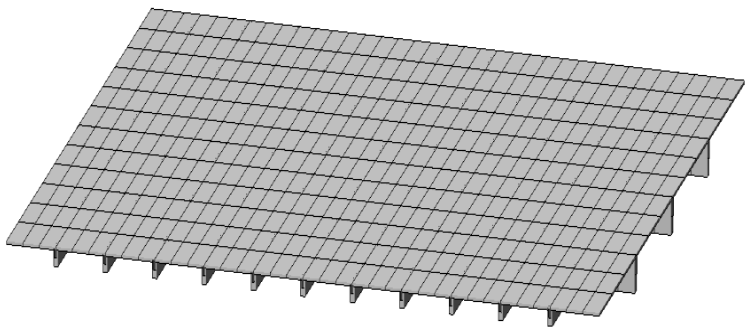


Fig. 119 – Scheme of a monolithic ribbed floor with beam slabs

Design Guidelines

The design of reinforced concrete floors is carried out in the following sequence:

1. Layout of the building/structure. At this stage, a structural diagram is developed, the main load-bearing elements are determined, their dimensions are specified, etc.
2. Static calculation. Based on the approved structural diagram, a calculation diagram is selected, all loads and influences are determined, and the forces in the system elements are determined.
3. Calculation of cross-sections. Based on the acting forces, the dimensions of the cross-sections of the elements are selected or the

previously assigned dimensions are checked to see if they are sufficient to withstand the forces.

4. Design. Determine the reinforcement schemes for the elements and create drawings of reinforcement and embedded products.

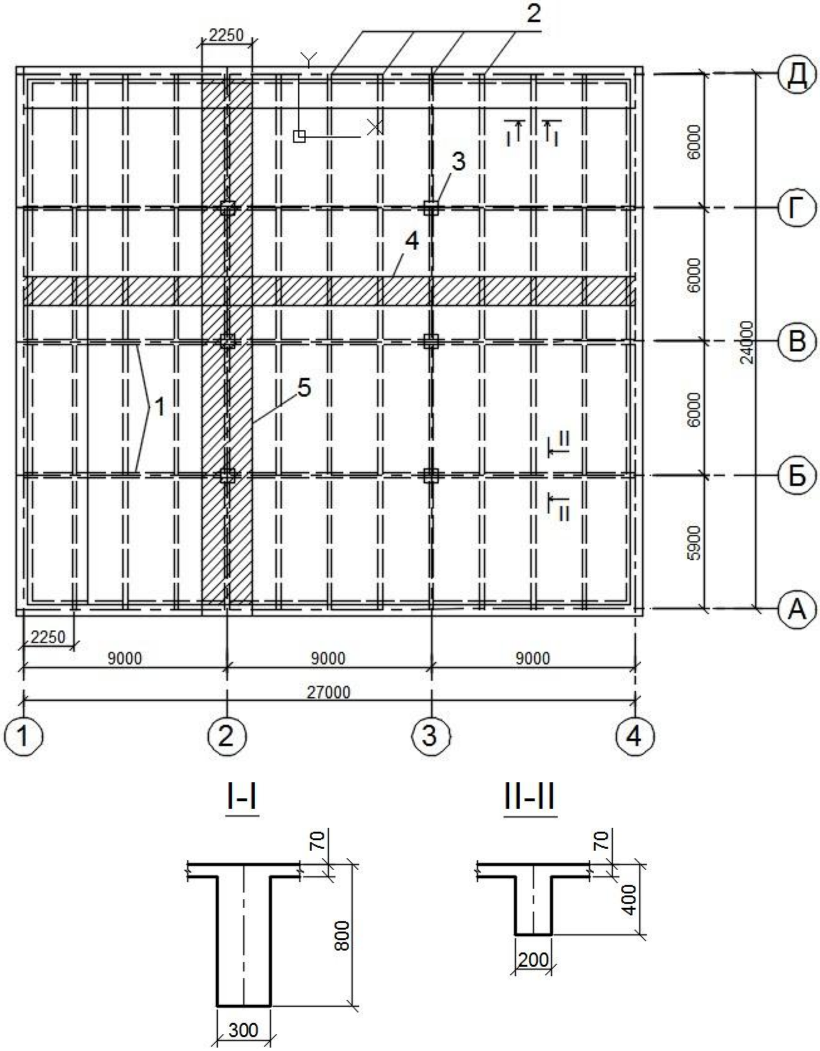


Fig. 120 – Calculation of a monolithic ribbed floor with beam slabs:
1 – main beam Bm-1; 2 – secondary beam Bm-2; 3 – column Km; 4 – design strip of slab Pm; 5 – load strip of secondary beam

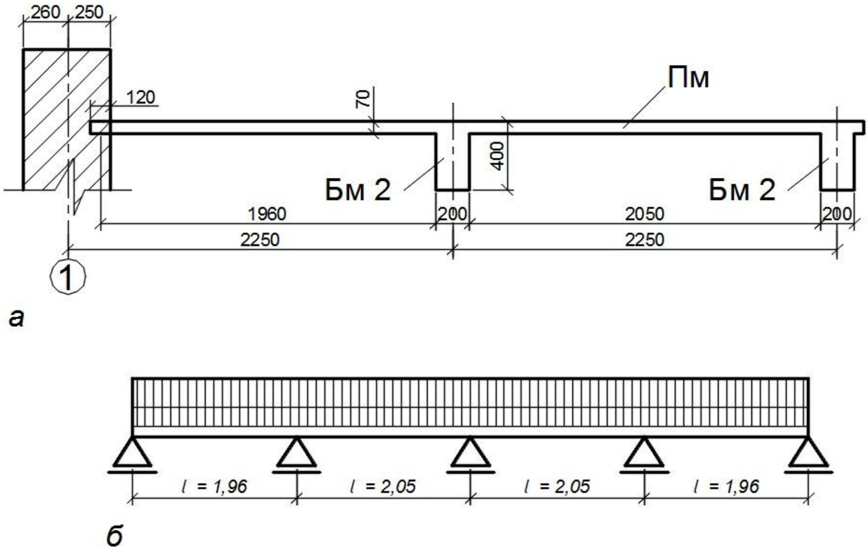


Fig. 121 – Calculation of the slab:

a – structural diagram; b – calculation diagram

Laboratory Work No. 8

Creating a calculation model of a beam floor slab

Purpose and plan of the session

Create a new task. Generate a slab, main and secondary floor beams. Set boundary conditions (support conditions). Create calculation and design options. Set stiffness characteristics and material parameters. Review the spatial model of the floor. Check the types of finite elements. Set the load. Perform a complete calculation of the slab.

Creating a new task

To create a new file, click the  – New button on the quick access toolbar. In the Diagram Description dialog box, specify the **Diagram Type – 5** – six degrees of freedom in the node (X, Y, Z, Ux, Uy, Uz displacement), the task name: “Monolithic beam floor” and, if desired, a description of the task (Fig. 122). Press the button – Confirm.

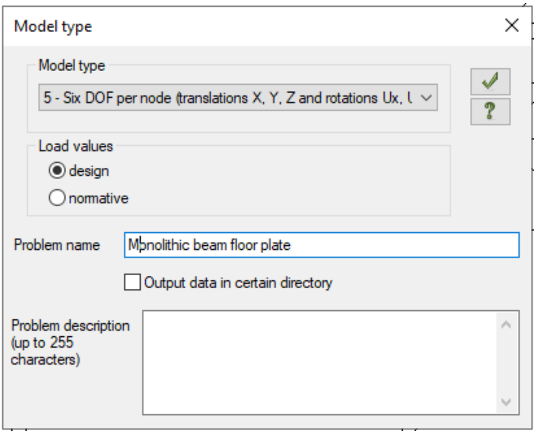
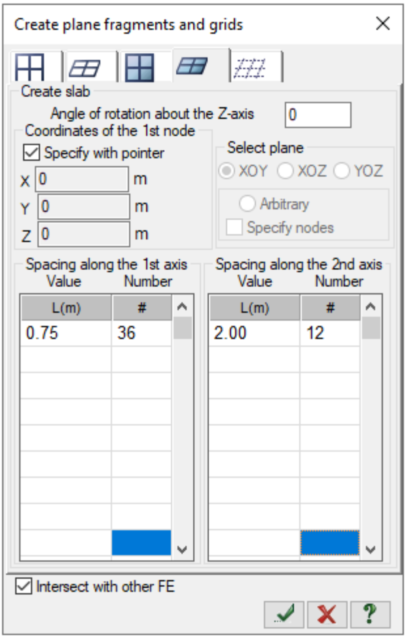


Fig. 122 – Model type dialog box

Creating a geometric slab schematic

On the **Create and Edit** tab, click – Generation of flat fragments and meshes in the drop-down list and select the Slab Generation command (Fig. 123).



In the **Create Flat Fragments and Meshes** dialog box, create a 36x12 finite element model by specifying the following parameters:

step along the first axis: L(m) – 0.75; N – 36;

step along the second axis: L(m) – 2; N – 12.

Leave the rest of the parameters as default and click – Confirm.

Fig. 123 – Create Planar Fragments and Meshes dialog box, Slab Generation tab

To model beams, on the **Create and Edit** tab, click the Add Element button 

and select **Add Member** from the drop-down list. A dialog box with the same name will appear (Fig. 124). On the diagram of the newly created slab, add members in the places where secondary and main beams should be (the step of secondary beams is 2.25 m, main beams – 6 m). The geometric diagram of a monolithic beam floor is shown in Fig. 125.

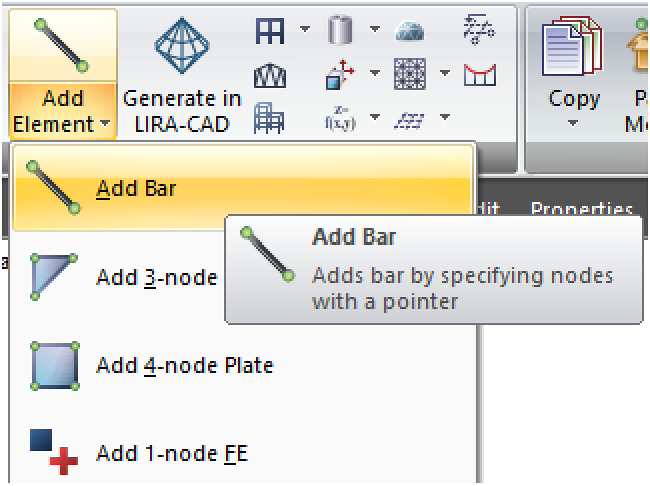


Fig. 124 – Drop-down list Add Bar

To save information about the created model, click the button  on the quick access toolbar.

Setting boundary conditions (constraints)

After activating the command button  – Node Marking, select the nodes supporting the slab on the two shorter sides (those supported by the main beams) with the cursor; the nodes will be displayed in red. To set boundary conditions, call the Restraints in nodes dialog box by clicking the **Restraints** button  (**Stiffness and restraints** panel on the **Create and edit** tab).

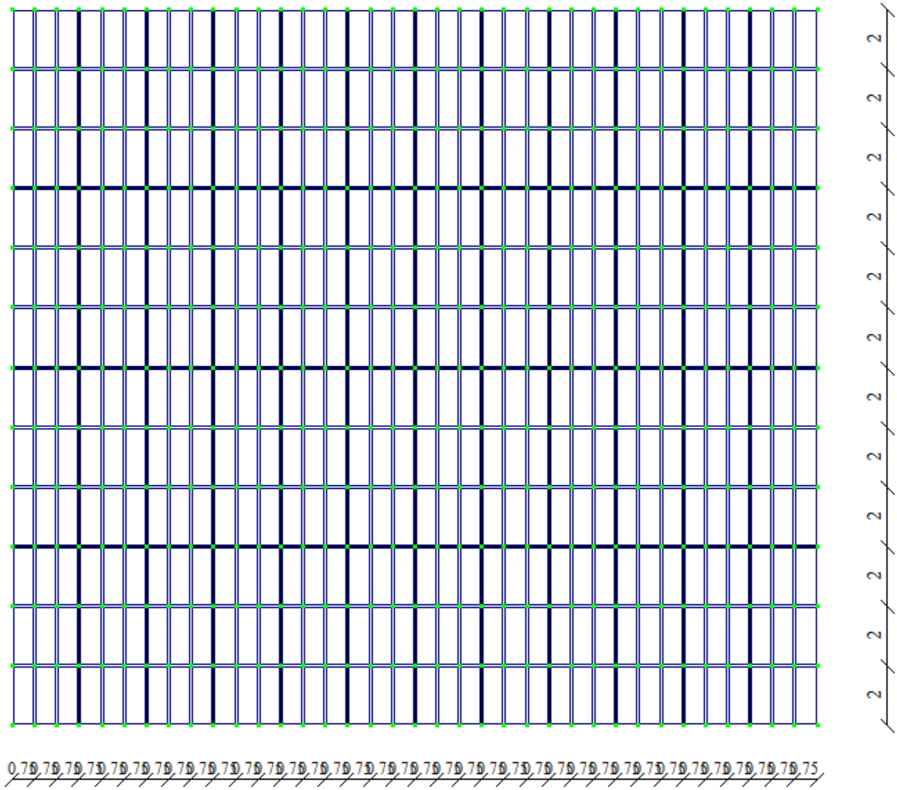


Fig. 125 – Geometric diagram of a monolithic beam floor

In the dialog box, select the directions in which node movements are prohibited (X, Y, Z, UX, UY, UZ – pinching) and click the button  – Add ties at the selected nodes. The color of the nodes will change from red to blue, indicating that the command has been executed (Fig. 126).

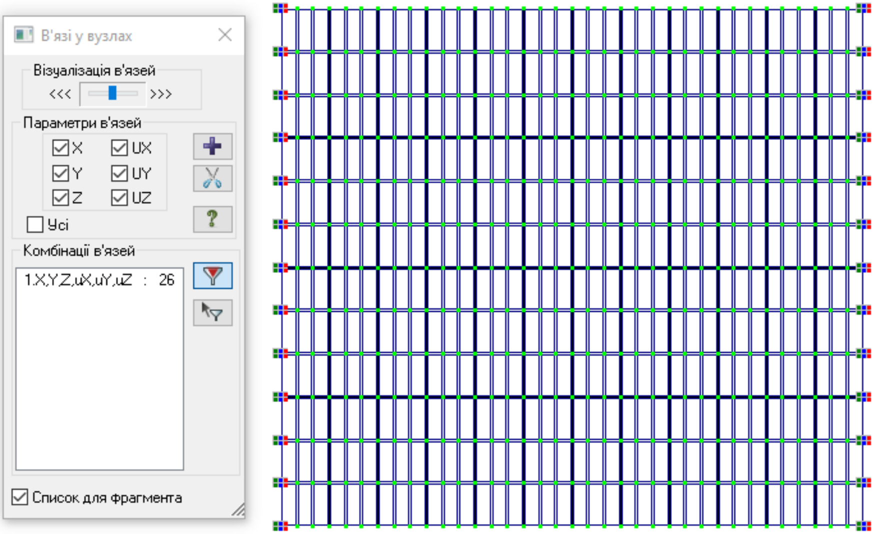


Fig. 126 – Setting the joints on the shorter sides of the slab

After that, select the slab support nodes on the longer sides and the nodes at the intersections of the longitudinal beams with the columns, prohibit movement in the Z direction (hinged support) – Fig. 127 – and press the button . All ties are set.

Setting calculation and design options

For structural calculations, click the button  – Design options (**Design** panel on the **Create and Edit** tab). In the **Design Options** dialog box, set the parameters for design (selection of reinforcement based on the strength of normal sections). In the same window, select the calculation option (based on the calculated combinations of forces - PCF) and the applicable standards. Click the button  - Apply.

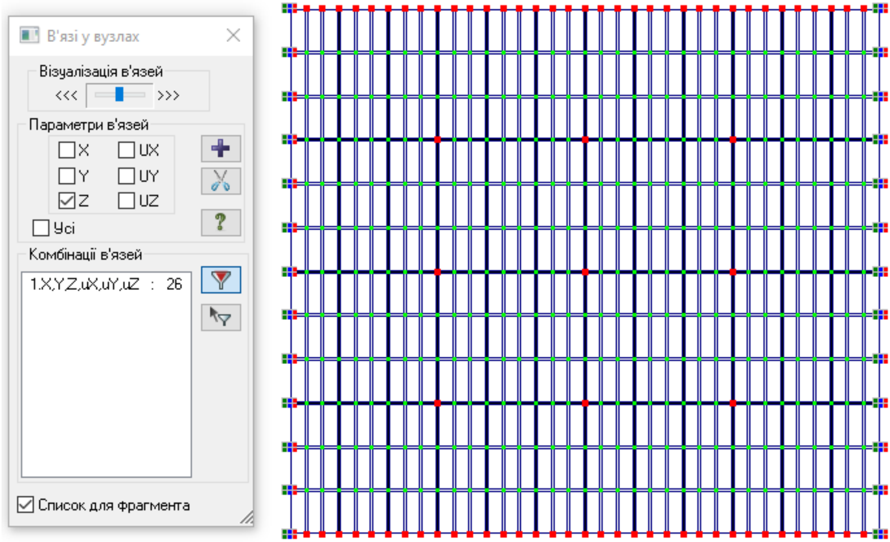


Fig. 127 – Specifying joints in the supporting nodes of the floor slab

Creating and assigning stiffnesses and material parameters

Click the button . In the **Stiffnesses and Materials** dialog box, form a list of stiffness types. Assign the stiffness type ***Plate H7*** to the slab elements. Longitudinal (main) beams have the cross-section type ***Beam 30x80***, transverse (secondary) beams – ***Beam 20x40*** (Fig. 128).

On the Reinforced concrete (RC) tab of the **Stiffness and materials** dialog box, you need to specify the parameters for reinforced concrete structures: type, concrete, reinforcement.

Select the Type radio button and click Edit.

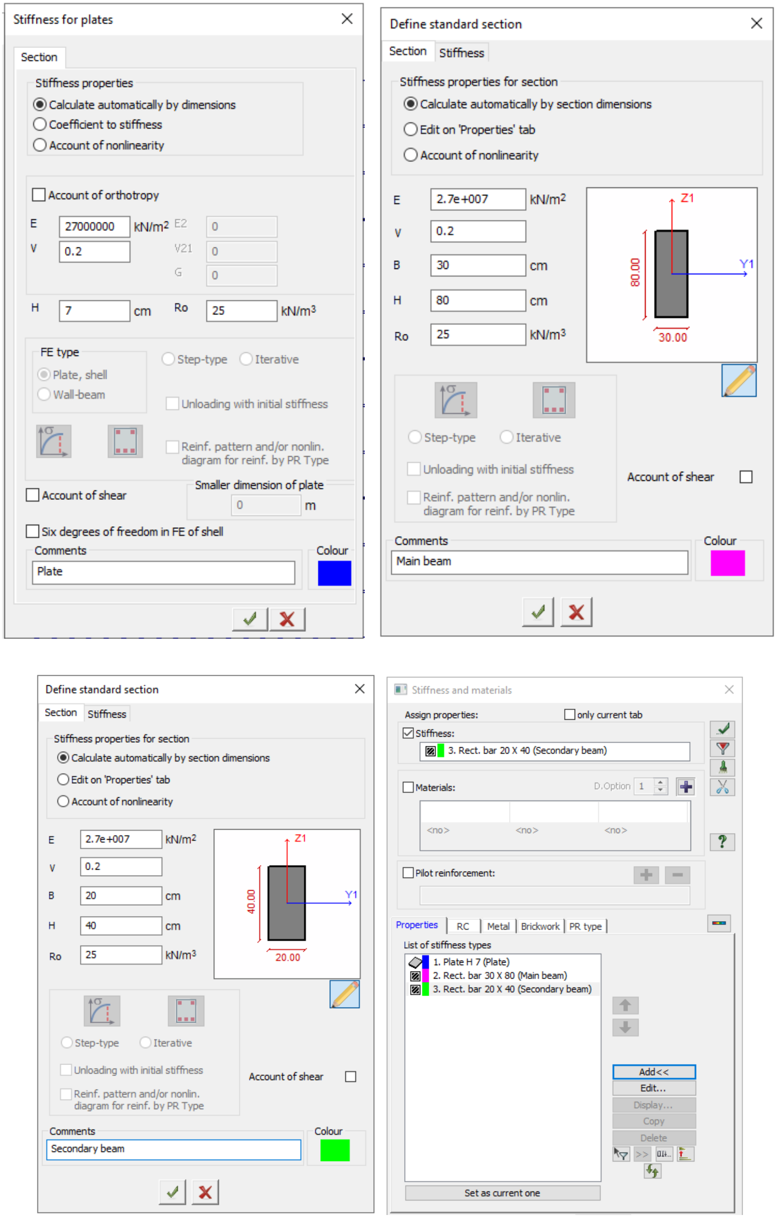


Fig. 128 – Specifying stiffnesses for slabs and beams

The **Materials for Calculating Reinforced Concrete Structures** dialog box will appear, in which you should left-click in the Plate field. In the upper right corner, specify: calculation type – Slab (bending), set the reinforcement bar spacing – 200 mm. In the Concrete field, select concrete class C16/20, in the Reinforcement field: longitudinal – class A400C, transverse – A400C. To save the entered data, click the  – Confirm button.

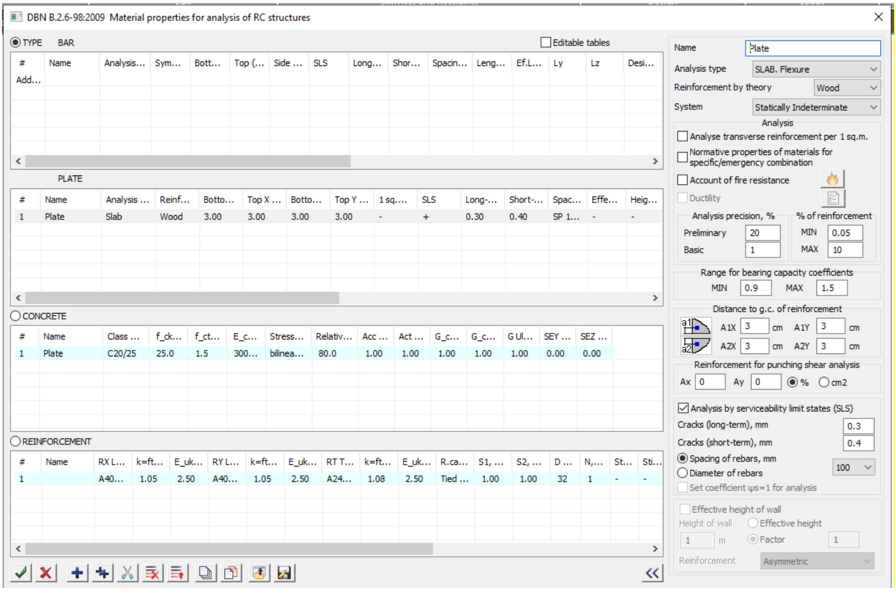
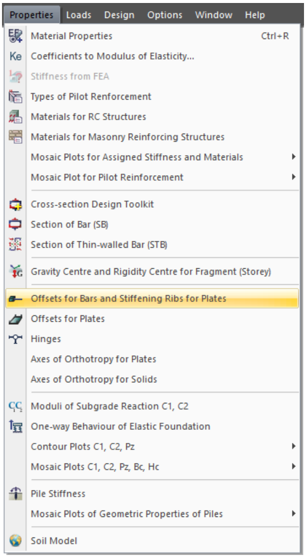


Figure 129 – Dialog box Material properties for analysis RC structures

After assigning stiffnesses for correct beam placement, **rigid inserts** must be entered. Rigid inserts are entered along the local Z-axis with a size equal to half the height of the cross-section of the bar element. That is, for main beams it is 0.4 m, and for secondary beams it is 0.2 m.

To specify stiffeners, first select the main beams and then select the **Stiffener** command from the **Stiffness drop-down** menu (Fig. 130). In the dialog box for assigning rigid inserts (Fig. 131), enter the value -0.4 in the first and second nodes and press . Perform the same operation for secondary beams (enter the value -0.2).



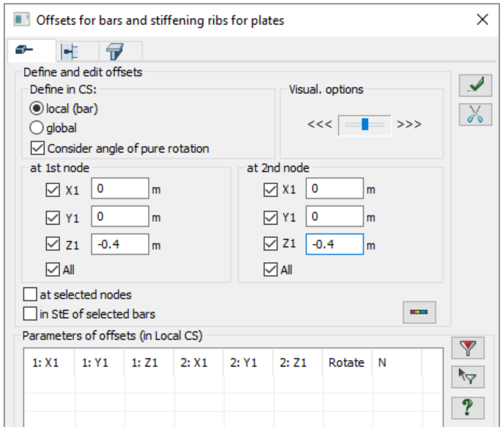


Fig. 131 – *Offsets for bars* dialog box

Fig. 130 – Setting the Rigid Rod Inserts command

To check the correctness of the calculation model, activate the **Spatial model** (3D graphics) command in the program menu. A spatial image of the beam floor will appear on the screen (Fig. 131).

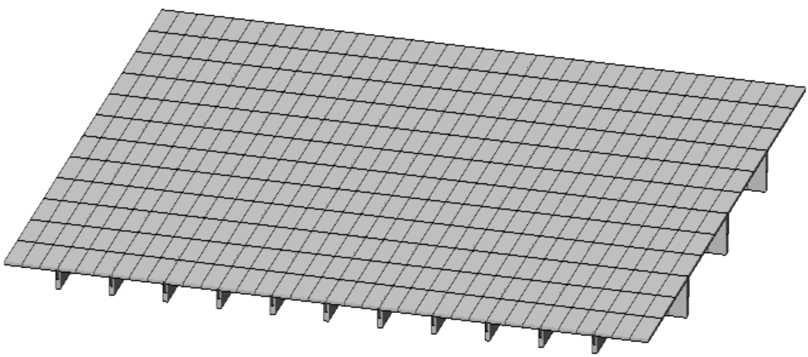
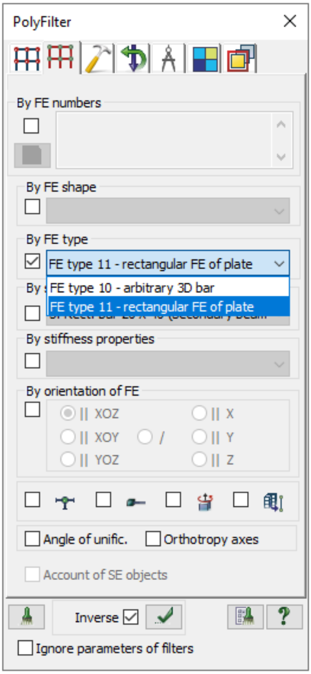


Fig. 131 – Spatial model of a beam floor

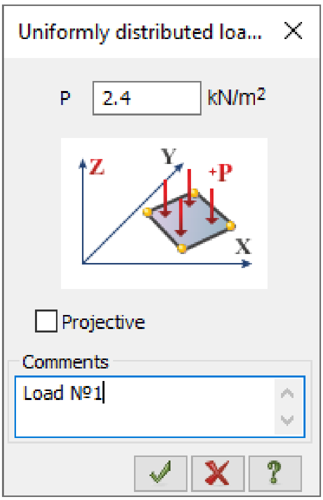
Checking the finite element (FE) type



Use the polyfilter (button  on the selection panel) to check the KE type. In the **Filter for Elements** dialog box, check the box next to KE Type (Fig. 132). Review the list. It should display two types: 10 – universal spatial bar (for beams) and 11 – rectangular FE slab (for slab elements). If necessary, the FE type can be changed using the command  – Change FE type (**Advanced editing** tab, **Diagram** panel).

Fig. 132 – *Polyfilter* dialog box

Load assignment

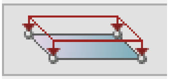


Load generation №1

On the **Load** panel (**Create and Edit** tab), select the **Load** command, and from the drop-down list, select the **Load on Plates** command.

In the Load **Assignment** dialog box, specify the coordinate system – Global, the direction of the load – along the Z axis.

Fig. 133 – Options dialog box

To set a uniformly distributed variable quasi-steady load, press the command button  , the **Parameters** window appears, in which enter P= 2.4 kN/m^2 (Fig. 133) and press the button  – Confirm.

Using the polyfilter (button  on the selection panel), select the slab elements (to do this, in the **Filter for Elements** dialog box, check the box in the By KE Type field, select Type 11 - Rectangular KE Slab from the list, and click the button  (all slab elements will be colored red). In the **Load Assignment** dialog box, press the button . A uniformly distributed load with an intensity of 2.4 kN/m^2 will be applied to the slab (Fig. 134).

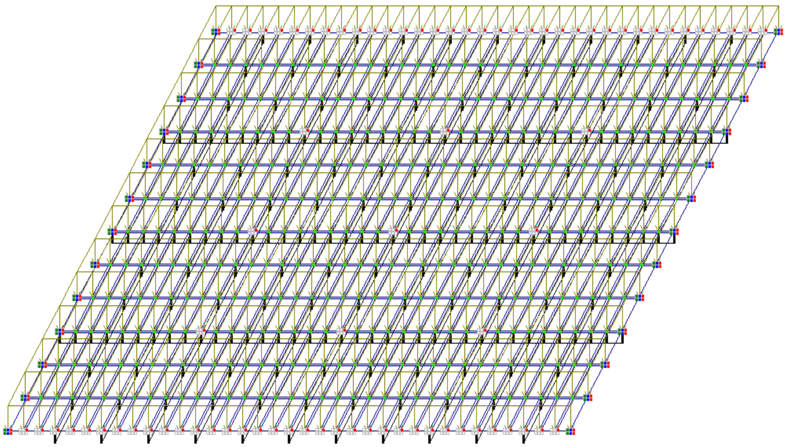


Fig. 134 – Evenly distributed load on the plate

Load generation №2

Change the number of the current download by clicking  – the *Next Download* button in the status bar (in the lower area of the working window).

On the **Loads** panel (**Create and Edit** tab), select the **Loads** command from the drop-down list, select the **Plate Loads** command, and set a variable long-term load evenly distributed over the area; for this, in the

Parameters window, and set P= 7.5 kN/m^2, click the  – *Confirm* button.

To specify *extended information about the download*, use the 

– *Download Editor* button on the Downloads panel. The **Download Editor** dialog box will appear, in which for download 1 in the Edit selected download field, select the *Permanent ribbon* from the ***Type*** list and click the  – Confirm button.

For download 2 in the Edit selected download field, select the Long ribbon from the Type list and click the  – Confirm button.

Generation of the DCF table (desing combinations of forces)

Since the calculation is performed for two loads of different durations, creating a table of calculated force combinations is necessary.

Full slab calculation

Before starting the calculation, it is recommended to save the created calculation scheme. To do this, use the Save button  on the quick access panel. To perform the calculation, click the button  – Perform full calculation on the Calculation tab.

Laboratory Work No. 9

Analysis of the results of static calculation and design of a beam floor slab

Purpose and plan of the session

Display the isopoles of displacements and stresses in the plate elements of the slab, create tables of calculated force combinations, analyze the results obtained.

Perform calculations to select reinforcement for slab elements. Review the mosaics of lower and upper reinforcement. Select working and structural reinforcement for the slab based on the tables of automated reinforcement selection results.

Review and analysis of static calculation results

The results of the static calculation are reviewed and analyzed in the **Analysis** tab.

To display the displacement isopole in the Z direction, select the command  – Displacement isopole in the global system, click the button  – Displacement isopole along Z (**Deformations** panel on the **Analysis** tab) – Fig. 135.

To display the stress isopole in the Mu direction, select the command  – Stress isopole, then click the button  – Stress isopole in Mu (**Stress in plates and solid elements** panel on the **Analysis** tab) – Fig. 136.

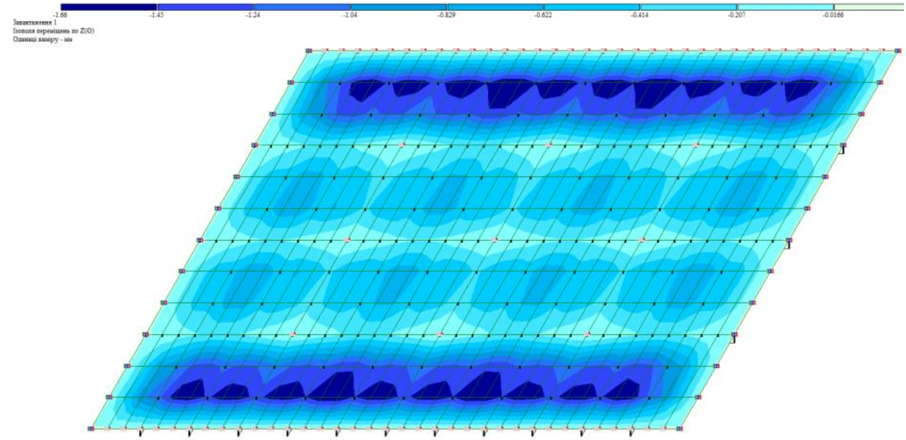


Fig. 135 – Isofield of displacements along Z

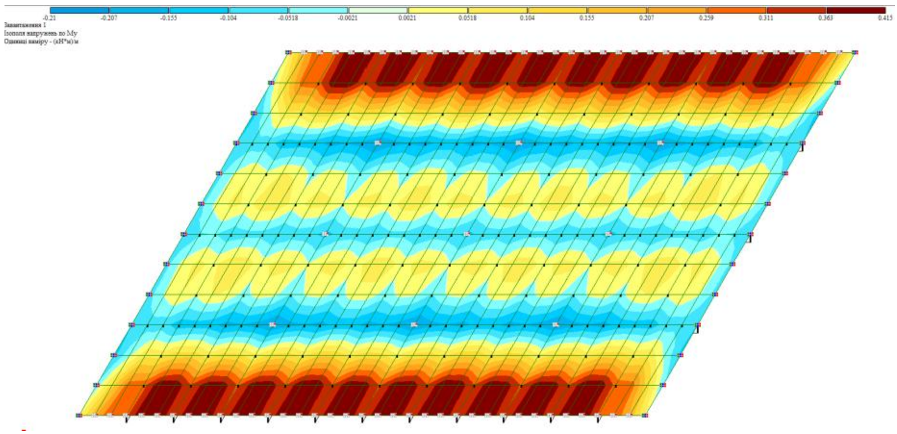


Fig. 136 – Stress isofield along Mu

To view the table with the values of the calculated force combinations in the elements of the diagram (Fig. 137), select the command – Standard Tables from the drop-down list **Documentation** (**Tables** panel on the **Analysis** tab). In the **Standard Tables** dialog box, select the RCS Calculated row and click the – Confirm button.

Одиниці виміру зусиль: кН
Одиниці виміру напружень: кПа
*Одиниці виміру моментів: кН*м*
*Одиниці виміру розподілених моментів: (кН*м)/м*
Одиниці виміру розподілених перерізуючих сил: кН/м
Одиниці виміру переміщень поверхностей в елементах: м

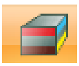
Tue Oct 26 21:38:28 2021 монолітне балкове п основна схема												
РОЗРАХУНКОВІ СПОЛУЧЕННЯ ЗУСИЛЬ												
ЕЛМ	НС	КР	СТ	КС	Г	N	MK	MY	QZ	MZ	QY	ЗАВАНТАЖЕННЯ.
433	1	1	1		A1	0	2.3949	3.3553	25.747	0	0	1 2
433	2	1	1		A1	0	2.3949	54.849	25.747	0	0	1 2
434	1	1	1		A1	0	-.30183	53.937	-12.054	0	0	1 2
434	2	1	1		A1	0	-.30183	29.828	-12.054	0	0	1 2
435	1	1	1		A1	0	-3.0506	30.099	-48.034	0	0	1 2
435	2	2	1		A1	0	-3.0506	-65.968	-48.034	0	0	1 2
436	1	2	1		A1	2.3095	.83547	-60.089	40.623	-.21026	-.05152	1 2
436	2	1	1		A1	2.3095	.83547	21.157	40.623	-.10720	-.05152	1 2
437	1	1	1		A1	2.3095	.02220	21.083	2.1354	-.10720	-.05152	1 2
437	2	1	1		A1	2.3095	.02220	25.354	2.1354	-.00415	-.05152	1 2
438	1	1	1		A1	2.3095	-1.0678	25.413	-36.199	-.00415	-.05152	1 2
438	2	2	1		A1	2.3095	-1.0678	-46.986	-36.199	.09889	-.05152	1 2
439	1	2	1		A1	2.3095	1.0678	-46.986	36.199	.09889	.05152	1 2
439	2	1	1		A1	2.3095	1.0678	25.413	36.199	-.00415	.05152	1 2
440	1	1	1		A1	2.3095	-.02220	25.354	-2.1354	-.00415	.05152	1 2
440	2	1	1		A1	2.3095	-.02220	21.083	-2.1354	-.10720	.05152	1 2
441	1	1	1		A1	2.3095	-.83547	21.157	-40.623	-.10720	.05152	1 2
441	2	2	1		A1	2.3095	-.83547	-60.089	-40.623	-.21026	.05152	1 2
442	1	2	1		A1	0	3.0506	-65.968	48.034	0	0	1 2
442	2	1	1		A1	0	3.0506	30.099	48.034	0	0	1 2
443	1	1	1		A1	0	.30183	29.828	12.054	0	0	1 2
443	2	1	1		A1	0	.30183	53.937	12.054	0	0	1 2
444	1	1	1		A1	0	-2.3949	54.849	-25.747	0	0	1 2
444	2	1	1		A1	0	-2.3949	3.3553	-25.747	0	0	1 2

Fig. 137 – DCF table for slab elements (fragment)

Review and analysis of reinforcement selection results

After calculating the task, go to the **Design** tab to review and analyze the reinforcement results. To select reinforcement in the diagram elements, click the 

- Reinforcement calculation button on the **Calculation** panel.

To view information about the selected reinforcement, use the command  – **Reinforcement area** (**Plate reinforcement** panel on the **Design** tab).

To display the selected reinforcement areas per 1 running meter along the X-axis near the bottom edge, click the button  – Bottom reinforcement in plates along the X1 axis (Fig. 138).

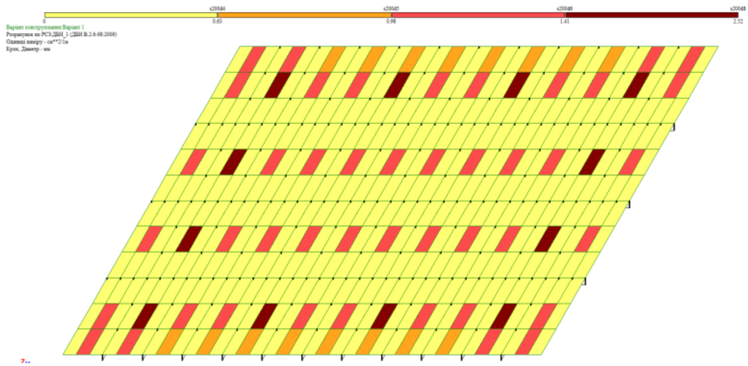


Fig. 138 – Reinforcement area per 1 linear meter along the X1 axis near the bottom edge

To view the mosaic representation of the area of upper reinforcement in the plates along the X1 axis, click the button  – Upper reinforcement in plates along the X1 axis (Fig. 139). Similarly, view the selected reinforcement in the direction of the Y1 axis.

To generate a table with the results of reinforcement selection from the drop-down list **Documentation**, select the command Results tables for reinforced concrete (**Tables** panel, **Design** tab), call the **Results tables** dialog box, and click the Plates button in the Elements field. A table with the reinforcement results will be displayed on the screen (Fig. 140).

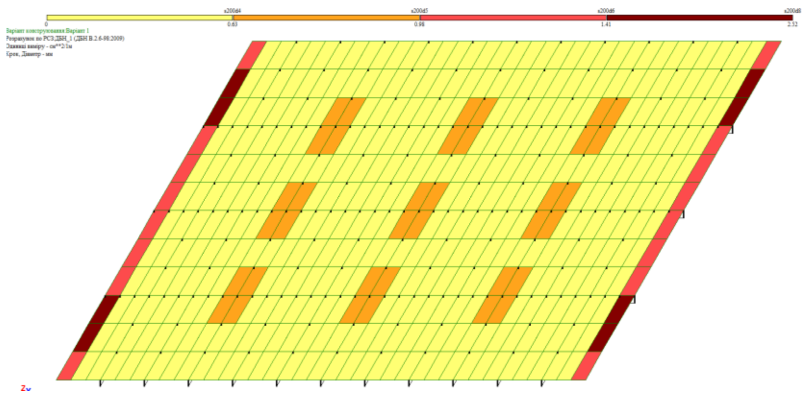


Fig. 139 – Reinforcement area per 1 linear meter along the Y1 axis near the upper face

МОНОЛІТНЕ БАЛКОВЕ ПЕРЕКРИТТЯ (LIR) 25 жовтень 2021

Результати армування у пластинах ДБН В.2.6-98:2009 (Варіант 1)

Елемент	Подовжня арматура				Поперечна арматура		Ширина розкриття тріщин	
	AS1	AS2	AS3	AS4	ASW1	ASW2	Коротк.	Трив.
Подовжня арматура: см**2 (1 м); ; Поперечна: см**2 (1 м); ; Шир.тріщин: мм;								
Плита 13; h= 7.00 см								
Бетон C16/20; Арматура: подовжня Ax: A400C, Ay: A400C; поперечна A400C								
Крок арматурних стержнів 200 мм								
13	1.38	0.35	1.46	0.35			0.25	0.25
	1.38	0.35	1.46	0.35				
Плита 14; h= 7.00 см								
Бетон C16/20; Арматура: подовжня Ax: A400C, Ay: A400C; поперечна A400C								
Крок арматурних стержнів 200 мм								
14	1.13	0.35	0.67	0.35			0.27	0.27
	1.13	0.35	0.67	0.35				
Плита 49; h= 7.00 см								
Бетон C16/20; Арматура: подовжня Ax: A400C, Ay: A400C; поперечна A400C								
Крок арматурних стержнів 200 мм								
49	1	0.35	1.26	0.35			0.3	0.3
	1	0.35	1.26	0.35				
Плита 50; h= 7.00 см								
Бетон C16/20; Арматура: подовжня Ax: A400C, Ay: A400C; поперечна A400C								
Крок арматурних стержнів 200 мм								
50	1.7	0.35	1.13	0.35			0.3	0.3
	1.6	0.35	1.13	0.35				

Fig. 140 – Table with the results of reinforcement selection in selected slab elements

Slab design

It should be remembered that in LIRA, the calculation is performed in a linear setting. In the case of slab reinforcement with separate bars with a slab thickness of up to 120 mm, separate reinforcement is used, i.e., all bars of the span reinforcement are brought to the supports, above which their reinforcement is installed. In this case, the lower span bars are designed to be continuous, passing through several supports, and in the extreme spans, where more reinforcement is required according to the calculation, additional bars are installed.

The working and structural reinforcement of the slab is selected according to the tables of results of automated reinforcement selection.In addition to the main reinforcement, it is necessary to install additional reinforcement in the non-working direction: near the supports near the walls (to absorb partial pinching moments) and above the main beams. The amount of this reinforcement is taken to be at least 1/3 of the slab's span reinforcement. The length of the reinforcement bars above the beams (both main and secondary) is determined on the condition that the horizontal sections on each side of the beam edge are equal to 1/4 of the slab span, i.e., 500 mm. The length of the pressure rods near the walls is taken on the condition that the horizontal sections from the wall are equal to 1/10 of the slab span.

Laboratory Work No. 10

Creating a calculation model of a secondary beam

Purpose and plan of the session

Create a calculation model of a secondary beam of a monolithic beam floor. Generate tables of design combinations of forces (DCF), perform a static calculation, analyze the calculation results. Select reinforcement. Take the initial data for the calculation from the previous task.

Task. Perform the calculation for a 24 m long beam for two loads:

- the first – constant uniformly distributed g= 7.77 kN/m, including its own weight;
- second – variable (long-term) uniformly distributed p= 19.24 kN/m.

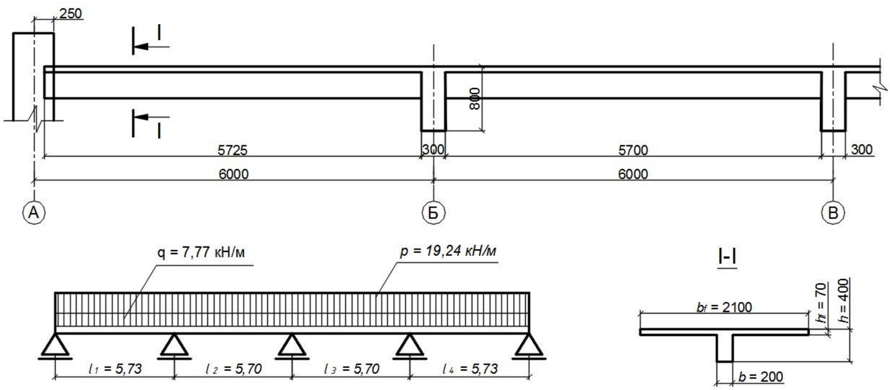


Fig. 141 – Design spans, cross-section and diagram of the secondary beam

Creating a new task

To create a new file, click the  – New button on the quick access toolbar. In the **Diagram Description** dialog box, specify the **Diagram Type – 2** – three degrees of freedom at the node (displacement X, Z, Uy) XOZ, task name: “Secondary beam.” Click the  –Confirm button.

Creating a geometric diagram of a beam

In the **Create Flat Fragments and Meshes** dialog box, on the Frame Generation tab, specify: step along the first axis L(m) – 24; N – 1, click the button  – Confirm.

Select the bar with the mouse cursor, divide it into 24 equal elements using the **Add Element – Divide into N Equal Parts** command (Fig. 142).

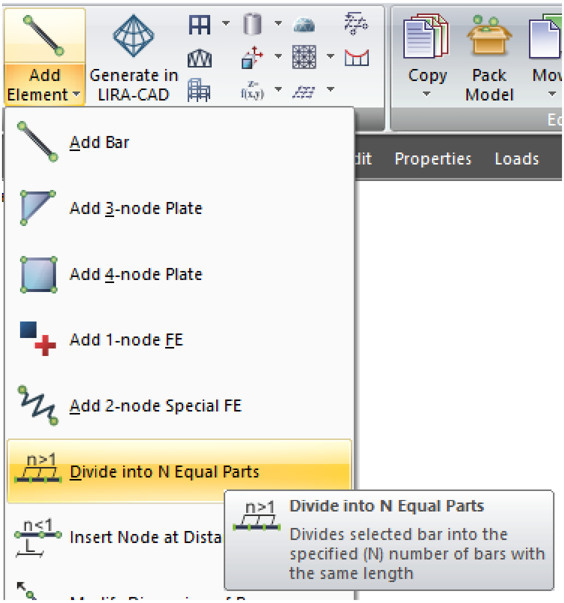


Fig. 142 – Add Element – Split command

The resulting geometric diagram of the beam is shown in Fig. 143.

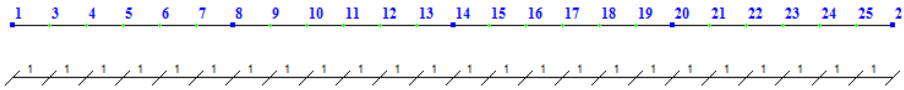
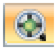


Fig. 143 – Geometric diagram of a beam

Setting boundary conditions

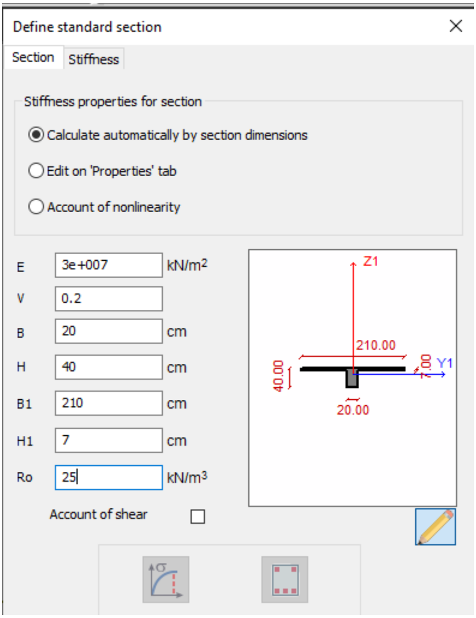
The node numbers on the calculation diagram are displayed on the screen. Activate the command button  – Node marking, mark nodes No. 1, 8, 14, 20 (they will be displayed in red). To set the constraints, use the button  . In the **Constraints** dialog box, select the directions in which node movement is prohibited (Z – hinged support), press  – Add constraints to selected nodes. Select node No. 2. In the **Connections** dialog box, select the X and Z directions (hinged-fixed support) and click  .

Selecting a calculation and design option

For structural calculations in accordance with current standards, click the button  – **Design options** and select the required standard (DBN V.2.6-98:2009) in the Design options dialog box. In the same window, specify the calculation option (based on the calculated load combinations – DCF).

Setting stiffness parameters and material parameters for elements of the diagram

Click the button  – Stiffness and Materials (**Stiffness and Constraints** panel on the **Create and Edit** tab) to open the **Stiffness and Materials** dialog box. In this window, click the **Add** button and in the **Add Stiffness** dialog box, select the first tab – **Standard Cross-Sections**, activate the Tavr_T cross-section (Fig. 144).



In the **Standard Cross-Section** Settings dialog box, specify:
- elastic modulus for C20/25 concrete – E = 3x107 kN/m2;
- geometric dimensions – B = 20 cm; H = 40 cm; B1 = 210 cm; H1 = 7 cm;
- material density – Ro = 25 kN/m^2.

To display a sketch of the cross-section being created, click the Draw button, then  – Confirm (Fig. 144).

Fig. 144 – *Specifying a standard cross-section* Dialog box

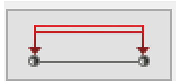
Next, in the **Stiffness and Materials** dialog box, go to the Reinforced Concrete tab to set the parameters for the beam. To do this, select the Type radio button, then click the Edit button. In the **Materials dialog box for calculating reinforced concrete structures**, specify the parameters:

- in the Bar field:
- name – Beam;
- reinforcement – asymmetrical;
- do not select corner bars;
- in the Concrete field, select class C20/25 from the list;
- in the Reinforcement field, select class A400C for longitudinal reinforcement and A240C for transverse reinforcement from the list.

Press the Apply button to confirm the data entry. Select all beam elements, and in the **Stiffness and Materials** dialog box, press the Apply button.

Load assignment

Assign loads according to the task in two loads. On the Load panel, select the **Load** command, and from the drop-down list, select Load on members. In the **Load Assignment** dialog box, use the radio buttons to specify the Global coordinate system and the direction of the load action –

along the Z axis. Click the uniformly distributed load button  to open the **Parameters** dialog box. In this window, enter P = 7.77 kN/m for the first load and click the Confirm button. Use the Node Marking button to mark all beam elements. In the **Load Assignment** dialog box, click the Confirm button again.

For the second load, set P = 19.24 kN/m.

To set the load information, use the button 

- Load Editor on the **Loads** panel, **Create and Edit** tab. In the **Load Editor** window for load 1, in the **Edit Selected Load** field, select Permanent from the list and click the button . For load 2, select Temporary Long-Term.

Generating the DCL table

To further calculate reinforcement based on design load combinations, select the **DCL** command on the **Calculation** tab (**Additional Calculations** panel) (Fig. 145).

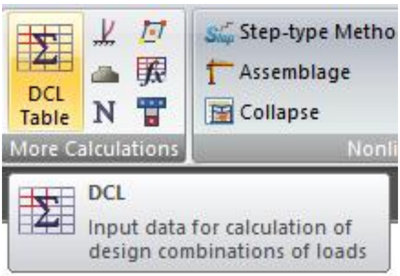


Fig. 145 – Calling the *DCL* command

The **Design combinations of loads** dialog box will appear (Fig. 146), in which you need to click the Add button. For the first main force combination, three columns will be automatically created: DCF1, DCF2, and DCF3 with load combination coefficients. It is important to remember that the loads on the design scheme were specified by design values (important for structural calculations). To analyze the deformations of the design scheme, you need to use standard loads, so the load combination coefficients must be divided by the load reliability coefficients.

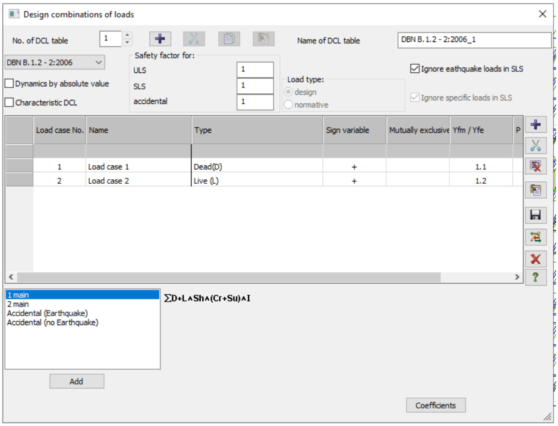


Fig. 146 – *Design combinations of loads* dialog box

Complete beam calculation

Before starting the calculation, you need to save the created calculation scheme. To do this, use the button  – Save on the quick access toolbar. To perform the calculation, click the button  – Perform calculation on the **Calculation** tab.

Laboratory Work No. 11

Analysis of static calculation results and design of a secondary beam

Purpose and plan of the session

Visualize the results of the static calculation of the beam. Create tables. Analyze the diagrams of bending moments and shear forces in the beam elements.

Visualization of calculation results

To switch to the visualization mode of static calculation results, go to the **Analysis** tab. By default, the deformed beam diagram is displayed. To display the initial diagram, click the button  (Deformations panel on the

Analysis tab).

To view the force diagrams, use the command buttons  (**Forces in the bars** panel on the **Results** tab). The force diagrams are shown in Fig. 147-148.

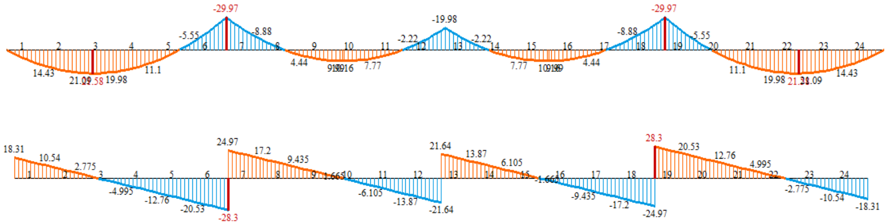


Fig. 147 – Diagrams of bending moments and shear forces in beam elements from the first load

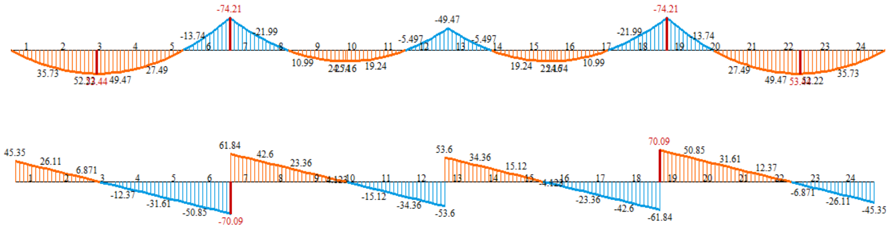


Fig. 148 – Diagrams of bending moments and shear forces in beam elements from the second load

Mark elements No. 3, 7, 10, and 13 on the diagram. Right-click to select **Standard Tables – Forces – All Loads**. The program will generate a table of forces for these elements, where the greatest moments occur (see Fig. 149). The forces from the DCF are shown in Fig. 150.

Sat Oct 30 00:01:18 2021 балка др_РСН· основна схема								
З У С И Л Л Я /НАПРУЖЕННЯ/ В ЕЛЕМЕНТАХ.								
2	3 - 1	3 - 2	7 - 1	7 - 2	10 - 1	10 - 2	13 - 1	13 - 2
	3	3	7	7	10	10	13	13
	4	4	8	8	11	11	14	14
	1 - ВЛАСНА ВАГА							
M	21.0899	19.9799	-29.9699	-8.87999	9.98999	7.76999	-19.9799	-2.21999
Q	2.77499	-4.99499	24.9749	17.2049	1.66499	-6.10499	21.6449	13.8749
	2 - ЗМІННЕ							
M	52.2227	49.4741	-74.2112	-21.9885	24.7370	19.2399	-49.4741	-5.49713
Q	6.87141	-12.3685	61.8427	42.6027	4.12284	-15.1171	53.5970	34.3570

Fig. 149 – Forces from both loads in selected elements

Sat Oct 30 00:02:47 2021 балка др_РСН· основна схема									
З У С И Л Л Я /НАПРУЖЕННЯ/ В ЕЛЕМЕНТАХ.									
2	3 - 1	3 - 2	7 - 1	7 - 2	10 - 1	10 - 2	13 - 1	13 - 2	
	3	3	7	7	10	10	13	13	
	4	4	8	8	11	11	14	14	
	1 - PCH1								
M	73.312	69.454	-104.18	-30.868	34.727	27.009	-69.454	-7.7171	
Q	9.6464	-17.363	86.817	59.807	5.7878	-21.222	75.242	48.232	
	2 - PCH2								
M	21.089	19.979	-29.969	-8.8799	9.9899	7.7699	-19.979	-2.2199	
Q	2.7749	-4.9949	24.974	17.204	1.6649	-6.1049	21.644	13.874	
	3 - PCH3								
M	73.312	69.454	-104.18	-30.868	34.727	27.009	-69.454	-7.7171	
Q	9.6464	-17.363	86.817	59.807	5.7878	-21.222	75.242	48.232	

Fig. 150 – Design combinations of loads

Reinforcement calculation and reinforcement selection

On the **Design** tab, click the button  – Reinforcement calculation, on the **Bar reinforcement** panel, select the Asymmetry command to set the display mode for asymmetric reinforcement, click the button  to display the lower distributed reinforcement (Fig. 151).

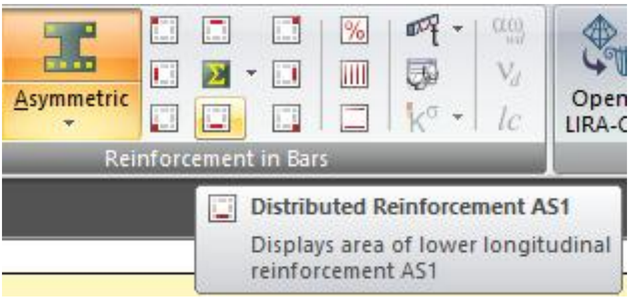


Fig. 151 – Command to display the lower distributed reinforcement AS1

The program displays the reinforcement diagram near the bottom edge of the beam (Fig. 152).

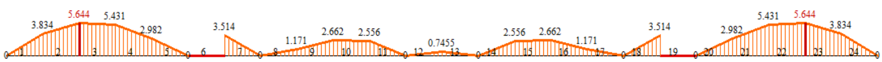


Fig. 152 – Area of lower distributed reinforcement AS1 in the beam

To display the reinforcement at the upper edge, press the button  – Fig. 153.

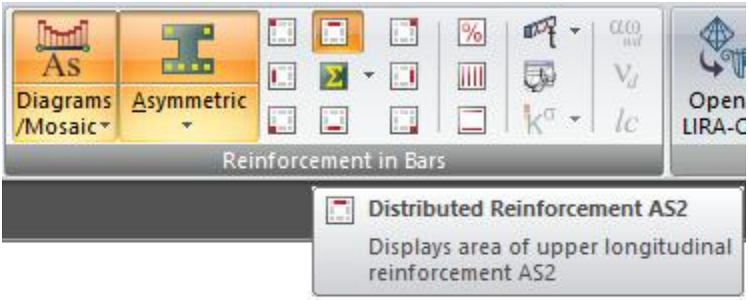


Fig. 153 – Command for displaying upper distributed reinforcement AS2

The reinforcement diagram at the upper edge of the beam is shown in Fig. 154.

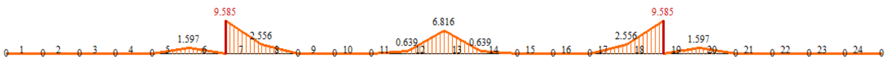


Fig. 154 – Area of upper distributed reinforcement AS2 in the beam

The table of reinforcement results for secondary beam elements is shown in Fig. 155.

Designing a secondary beam

It should be noted that in LIRA, the calculation is performed in a linear setting. In the case of beam reinforcement with separate bars, longitudinal working reinforcement for span and support sections is first selected according to the calculation (see the table of automated reinforcement selection results, columns AS1-AS2). Structural (transverse) reinforcement is marked as ASW1 in the table.

Результати армування у стержнях ДБН В.2.6-98:2009 (Варіант 1)

Переріз	С/НС	Подовжня арматура									Поперечна ар-па		Ширина розкр. тріщин	
		AU1	AU2	AU3	AU4	AS1	AS2	AS3	AS4	%	ASW1	ASW2	Коротк.	Трив.
Подовжня арматура: см**2; Поперечна: см**2; Шир.тріщин: мм;														
Балка 3; Тавр полиця зверху; B=20.00; H=40.00; B1=210.00; H1=7.00 см; L=1.00 м														
Бетон C20/25; Арматура: подовжня A400C; поперечна B500														
1	Н					5.64				0.26	0.88			
						5.64				0.26				
2	Н					5.43				0.25	0.88			
						5.43				0.25				
Балка 7; Тавр полиця зверху; B=20.00; H=40.00; B1=210.00; H1=7.00 см; L=1.00 м														
Бетон C20/25; Арматура: подовжня A400C; поперечна B500														
1	Н					3.51	9.59			0.62	8.27			
						3.51	9.59			0.62				
2	Н						2.56			0.12	5.7			
							2.56			0.12				
Балка 10; Тавр полиця зверху; B=20.00; H=40.00; B1=210.00; H1=7.00 см; L=1.00 м														
Бетон C20/25; Арматура: подовжня A400C; поперечна B500														
1	Н					2.66				0.13	0.88			
						2.66				0.13				
2	Н					2.56				0.12	0.88			
						2.56				0.12				
Балка 13; Тавр полиця зверху; B=20.00; H=40.00; B1=210.00; H1=7.00 см; L=1.00 м														
Бетон C20/25; Арматура: подовжня A400C; поперечна B500														
1	Н					0.75	6.82			0.35	7.17			
						0.75	6.82			0.35				
2	Н						0.64			0.03	4.59			
							0.64			0.03				

Рис. 155 – Results of reinforcement selection in selected beam elements

Topic 6. Design of metal trusses

Task. Calculate and analyze the stress-strain state in the elements of a metal trapezoidal truss made of seamless hot-rolled pipes. Check the specified cross-sections and select the cross-sections of the truss elements.
Perform the calculation for the following loads: 1 – dead weight of the roof structures; 2 – snow load on the entire span; 3 – snow load on half of the truss span.

Laboratory Work No. 12

Creating a calculation model, analysis of static calculation results and selection of truss element sections

Purpose and plan of the session

Create a calculation model. Select a calculation and design option. Generate tables of calculated load combinations (LC). Perform a complete calculation of the steel truss.

Review and analyze the results of the static calculation of the truss. Create tables. Analyze the diagrams of bending moments and shear forces in the truss elements. Analyze the results of checking and selecting the cross-sections of the truss elements. Design the truss.

Creating a new task

On the quick access toolbar, click the  – New button. In the **Diagram Description** dialog box, enter the name of the task: “Truss L=24m,” and in the **Diagram Type** list, select the line 1 – Two degrees of freedom at the node (X,Z displacement). Click the  – Confirm button.

Selecting a calculation and design option

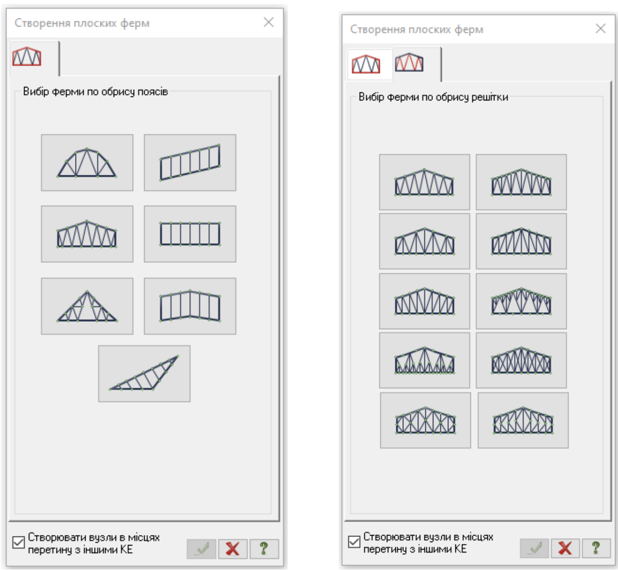
For structural calculations in accordance with current standards, click the button  – Scheme design options and select the required standard (DBN V.2.6-198:2014) in the **Design options** dialog box. In the same window, specify the calculation option (based on design combinations of forces – DCF).

Creating a geometric diagram of a truss

On the **Create and Edit** tab, click the  - Generate Trusses button and select the command of the same name (Fig. 156). The Create Flat Trusses dialog box will appear (Fig. 157).

Fig. 156 – *Creat Truss* dialog box

In this window, select a truss by the outline of the chords (Fig. 157, a) – in our case, to create a trapezoidal truss diagram, press the button . To select a truss configuration by the outline of the grid (Fig. 157, b), press the corresponding button – .



a) b)

Figure 157 – *Cenerate 2D truss* dialog box: a – select a truss by the outline of the belts, b – by the outline of the lattice

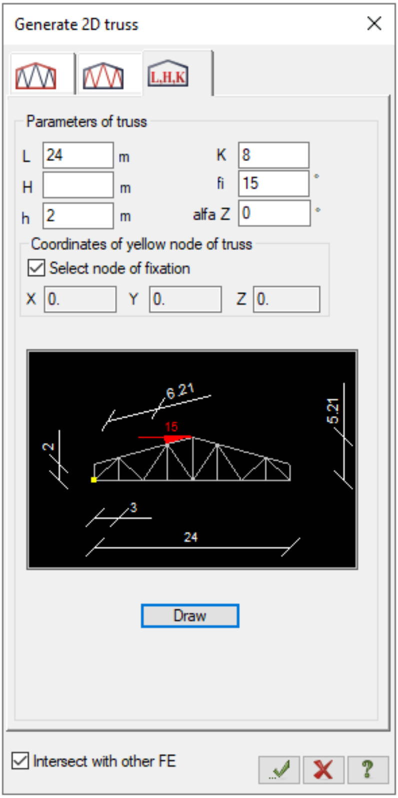


Fig. 157 – Dialog box for specifying truss parameters

In the dialog box that appears (Fig. 157), you need to specify all the numerical parameters that will determine the dimensions of the truss. To do this, fill in the appropriate fields:

- ✓ truss length L= 24 m;
- ✓ h = 2 m;
- ✓ parameter K (number of panels) – 8;
- ✓ fi – angle of inclination of the upper chord – 15^0.

After that, click on the Draw button. If the user is satisfied with the created diagram, click – Apply, if not – make corrections. The geometric diagram of the truss is shown in Fig. 158.

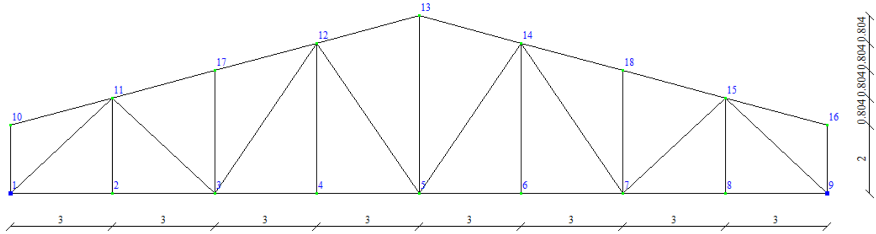


Fig. 158 – Geometric diagram of a truss

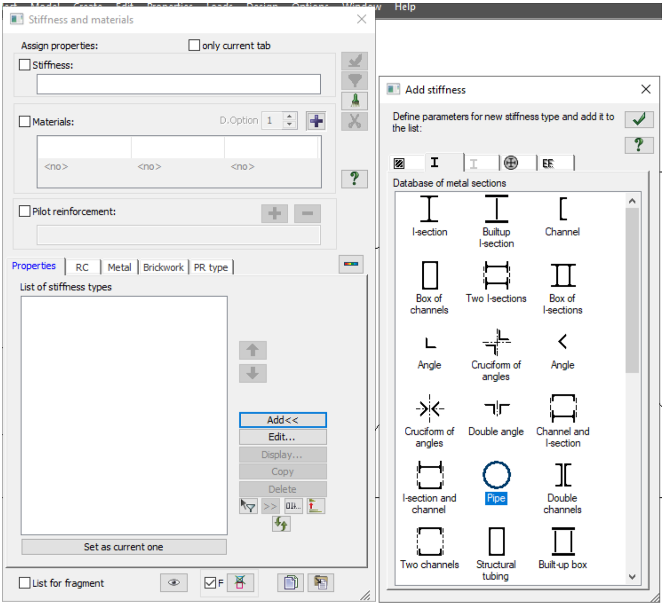
Save the task by clicking the button . In the Save As dialog box, specify the folder where this file will be saved.

Setting boundary conditions

Activate the command button  – Node Marking, mark node No. 1 (it will be displayed in red). To set the constraints, use the button . In the **Constraints** dialog box, select the directions in which node No. 1 (Z – hinged support) is prohibited from moving, click  – Add constraints to selected nodes. Next, select node No. 9. In the **Constraints** dialog box, select the X and Z directions (hinged support) and click .

Setting stiffness parameters and material parameters for diagram elements

Click the button  – Element stiffness and materials. In the **Stiffness and Materials** dialog box (Fig. 159), create a list of **Stiffness** types. On the Stiffness tab, click the Add button, select the second tab – **Metal Cross-Sections Database**, and activate the Pipe cross-section. The **Steel Section** dialog box (Fig. 160) will open, in which you select a product from the list, for example, Seamless Hot-Rolled Pipe and Pipe Profile.



a) b)

Fig. 159 – Dialog boxes: a – Stiffnesses and materials (Stiffnesses tab), b – Add stiffness

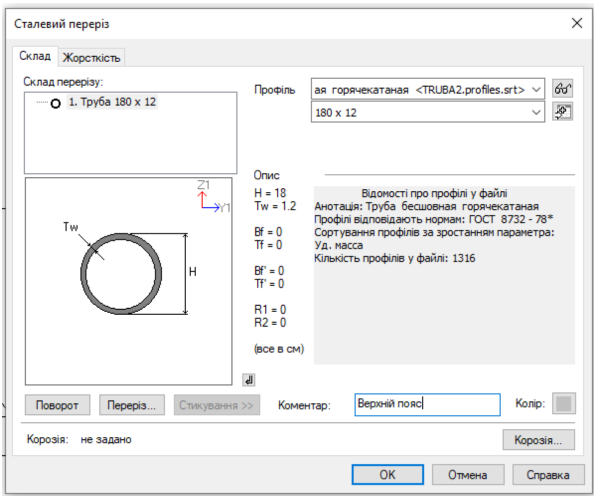


Fig. 160 – Steel Section dialog box

As for the profile, for the upper chord of the truss, which will be under compression, it is worth choosing a slightly stronger one, for example, 180x12; for the lower chord of the truss, which will be under tension, a less powerful one, 168x12; for the lattice elements, an even smaller one, 89x6. The profile is chosen arbitrarily, based on the designer's experience or intuition. Later, the program will perform a check and selection of truss elements.

For convenience, you can change the color and specify the name of the group of elements in the Comment field for each type of stiffness, for example, Upper chord. The result will be a list of rigidities (Fig. 161), each of which will need to be assigned to the corresponding truss elements. However, to design a truss, you need to specify materials and additional characteristics. This must be done separately for each type of rigidity.

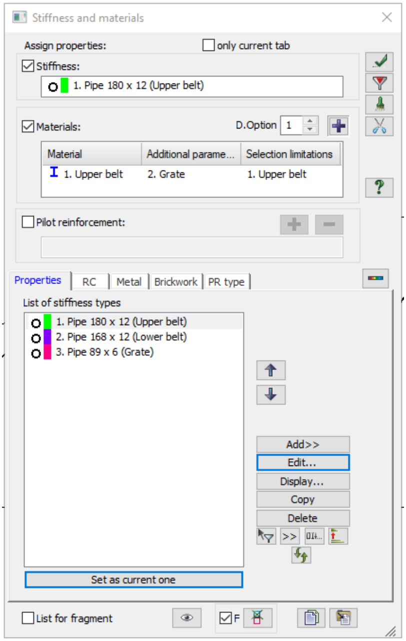


Fig. 161 – Created list of stiffnesses

To do this, in the **Stiffnesses and Materials** dialog box, on the **Stiffnesses** tab, double-click on the 1. Pipe ribbon. Upper belt, go to the

Steel tab, with the Material radio button selected (Fig. 162), click the Add button.

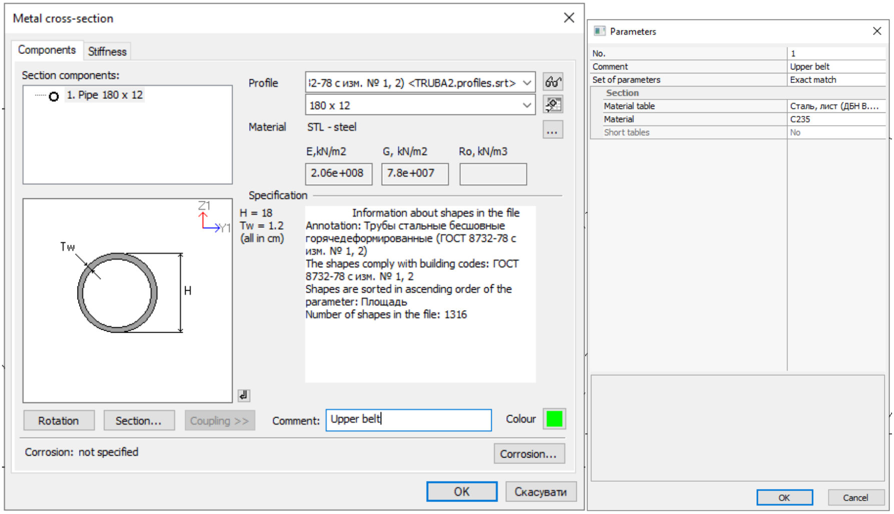


Fig. 162 – Dialog boxes: a – Stiffness and materials (Steel-Material tab), b – Parameters

In the Parameters dialog box (Fig. 162, b), select the steel grade from the list and click OK.

Next, with the Additional characteristics radio button selected (Fig. 163), specify the type of element – truss, limit flexibility, etc.

These steps must be performed sequentially for all types of stiffnesses from the list. Next, assign the appropriate stiffnesses to all truss elements, then close the **Stiffnesses and Materials** dialog box.

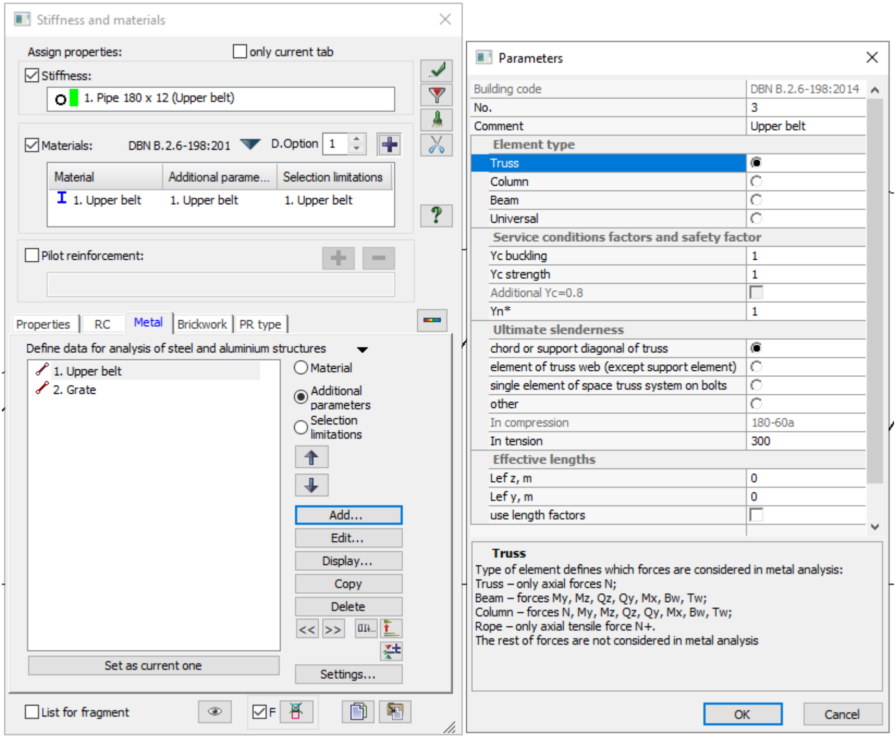


Fig. 163 – Dialog boxes: a – Stiffness and materials (*Assing properties)*, b – Parameters

Load assignment

Node loads are assigned to the truss. Activate the command button – Node Marking, mark the nodes according to the loads:

- Load 1 – permanent, from the dead weight of the roof structures, applied to the nodes of the upper chord of the truss, P = 5 kN.
- Load 2 – short-term, snow, for the entire span, applied to the nodes of the upper chord of the truss, P = 4 kN.
- Load 3 – short-term, snow, for half the span, applied to the nodes of the upper chord of the truss, P = 4 kN.

Each load is formed separately. Twice less load is applied to the extreme nodes, because the load area here is twice smaller (Fig. 164).

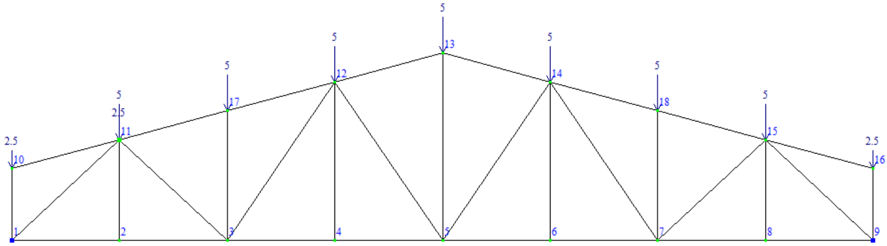


Fig. 164 – Loading 1

The second load is set in the same way (Fig. 165).

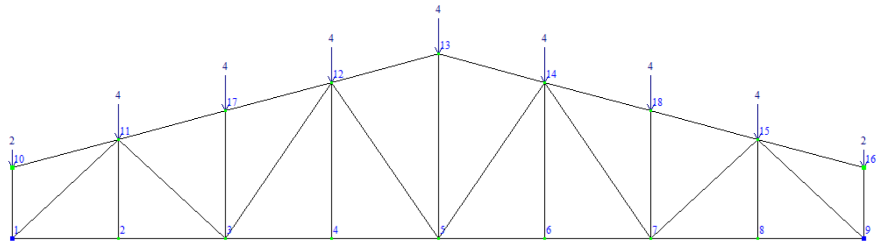


Fig. 165 – Loading 2

The third load is applied to half of the truss span (Fig. 166).

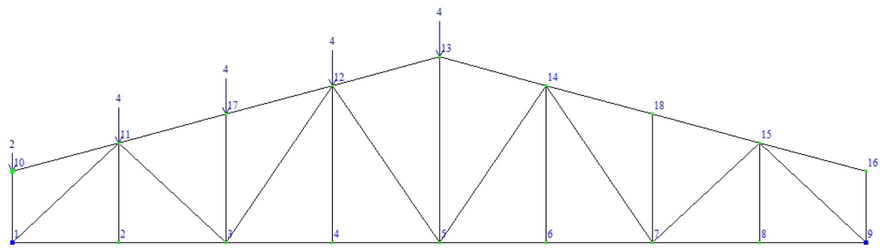


Fig. 166 – Loading 3

Generating the DCF table

Since several loads of different durations were specified, it is necessary to create an **DCF table** for further selection of truss member cross-sections based on the most dangerous combinations of forces.

The dialog box of the same name is called by clicking on the button  – **DCF** table **(DCF** panel on the **Calculation** tab) – Fig. 167.

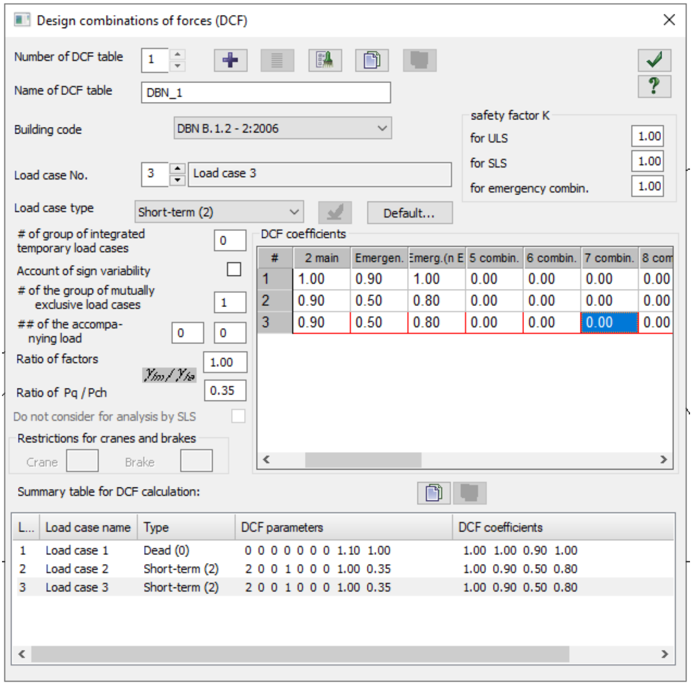


Fig. 167 – Design combinations of forces (DCF) dialog box

In this window, for the selected building codes DBN V.1.2-2:2006, you need to specify the types of loads: 1 – permanent, 2 and 3 – short-term snow.

For loads 2 and 3, enter 1 in the text field No. of mutually exclusive loads group in order to take into account only the most unfavorable snow load for a specific combination of forces. Save the table by clicking the button .

Complete calculation of the truss

Before starting the calculation, you need to save the task. To calculate, click on the button – Perform calculation.

Complete calculation of the truss

Before starting the calculation, you need to save the task. To calculate, click the button  – Perform calculation.

Viewing and analyzing the results of static calculation

On the **Analysis** tab, in the calculation results viewing mode, the calculation diagram is displayed by default, taking into account the displacements of the nodes. To display the diagram without taking into account node displacements, click the button – Initial Diagram (**Deformations** panel on the **Analysis** tab).

Displaying internal force diagrams

To display the longitudinal force diagram in the truss elements, click the button N – Longitudinal force diagrams N (**Forces in bars** panel on the **Results** tab).

Fig. 168 shows the diagram of longitudinal forces in truss members from the first load.

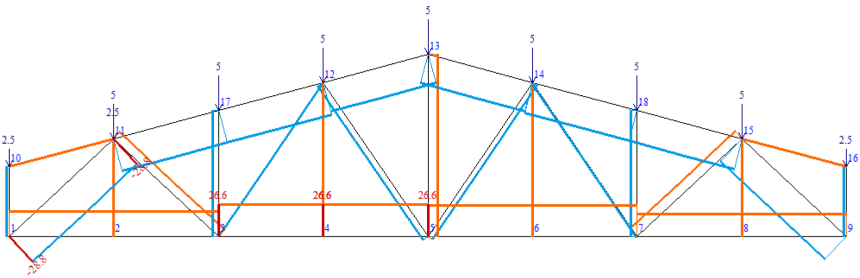


Fig. 168 – Diagram of longitudinal forces in truss elements from the first Loading

Creating and viewing calculation result tables

To display a table with the values of calculated forces in the elements of the diagram, click the button Докумен-тація – Documentation (**Tables** panel on the **Analysis** tab), select the Interactive Tables command from the **Documentation** drop-down list.

The Form Editor dialog box will appear, in which select the **DCF bars** ribbon (see Fig. 120), in the **Create DCF Table** window, select the desired option – for all elements or only for selected ones (in which case these elements must be selected in advance). Confirm the operation.

The program will generate a table of calculated force combinations for the truss elements.

Review and analysis of the results of checking and selecting steel truss member cross-sections

Go to the Design tab. On the Steel: Calculation panel, activate the Calculation command by clicking the button . The **Steel Calculation** dialog box (Fig. 169) will appear, in which you need to click the Calculate button.

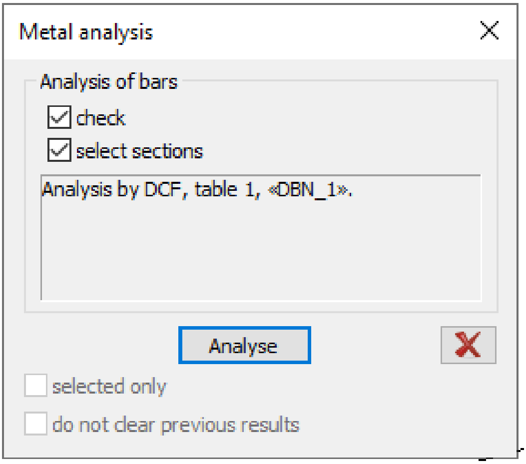


Fig. 169 – Steel Calculation dialog box

After the calculation is performed, the **Max Results by Elements** panel (Fig. 170) becomes available, which is used to analyze the specified cross-sections.

Fig. 170 – *Max Results by Elements: Inspection and Selection*

By clicking on the button 

, you will get a mosaic of the calculation results for the truss elements according to the first group of limit states (for strength) – Fig. 171. At the same time, using the color scale, you can determine the percentage of the load-bearing capacity of each truss element for the specified stiffness.

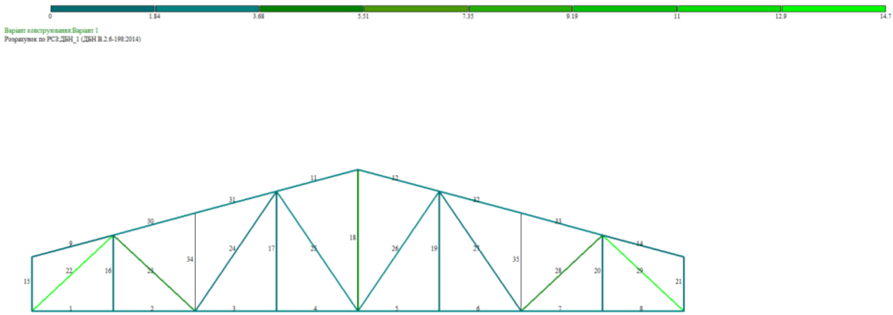


Fig. 171 – Mosaic of results of strength calculation of truss elements

By clicking on the button 

, you can obtain a mosaic of truss member calculation results for the second group of limit states (for flexibility). Local stability of members can be checked by clicking on the button 

. *The results of cross-section selection are analyzed using the corresponding buttons* 

. The selection of truss elements is carried out by pressing a button 

.

To view the results of the verification and selection of truss elements in tabular form, execute the command Documentation – Results tables for steel in the **Tables** panel (Fig. 172).

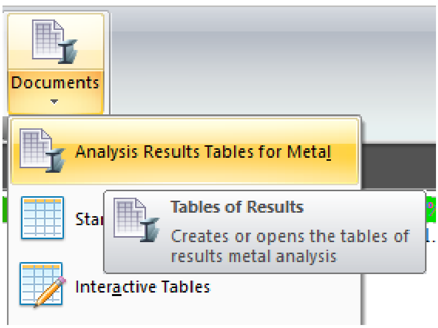


Fig. 172 – Documentation command – Results tables for steel

The **Tables** dialog box will appear, in which select the **Metal Elements** tab. **Check** (Fig. 173) and confirm the operation.

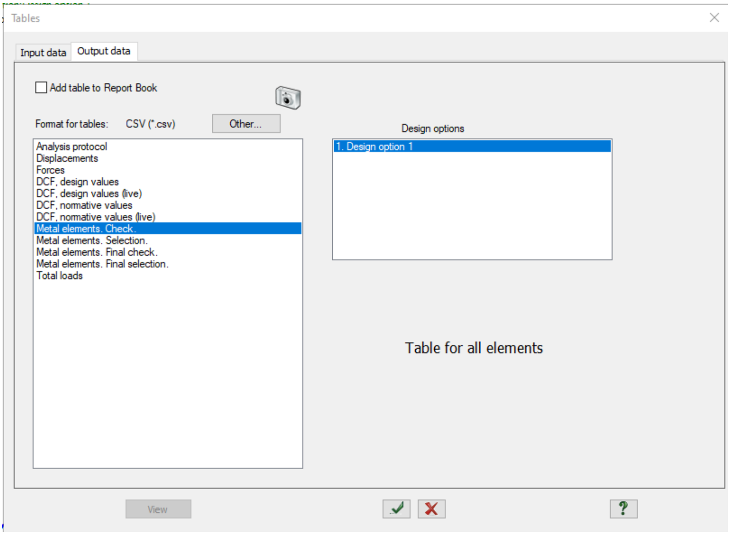


Fig. 173 –Tables dialog box

The program will generate a table (Fig. 174) with the percentages of load-bearing capacity exhaustion of steel truss element cross-sections.

To generate a table with selected cross-sections (Fig. 175), select the **Steel Elements. Selection ribbon.**

Ферми

Елемент	НП	Група	Крок ребер (пла нок)	Прим ітка	Проценти вичерпання несучої здатності балки по перерізах, %										Довжина елемента
					нор	СУ1	СZ1	ГУ1	ГZ1	СС	СП	1ГС	2ГС	М.С	
Переріз: 1.1.1. Труба 180 x 12															
Профіль: 180 x 12; ГОСТ 8732 - 78*															
Сталь: ВСт3пс4;															
Сортамент: Труба бесшовная горячекатаная															
9	1		0.00		0	0	0	0	0	0	0	0	0	0	3.11
9	2		0.00		0	0	0	0	0	0	0	0	0	0	3.11
11	1		0.00		3	0	0	0	0	3	0	3	0	3	3.11
11	2		0.00		3	0	0	0	0	3	0	3	0	3	3.11
12	1		0.00		3	0	0	0	0	3	0	3	0	3	3.11
12	2		0.00		3	0	0	0	0	3	0	3	0	3	3.11
14	1		0.00		0	0	0	0	0	0	0	0	0	0	3.11
14	2		0.00		0	0	0	0	0	0	0	0	0	0	3.11

Fig. 174 – Table for checking specified cross-sections

Ферми

Елемент	НП	Група	Крок ребер (планок)	Примітка	Проценти вичерпання несучої здатності балки по перерізах, %										Довжина елемента
					нор	СУ1	СZ1	ГУ1	ГZ1	СС	СП	1ГС	2ГС	М.С	
Переріз: 1.1.1. Труба 180 x 12															
Профіль: 180 x 12; ГОСТ 8732 - 78*															
Сталь: ВСт3пс4;															
Сортамент: Труба бесшовная горячекатаная															
9			Підібрано: 1.1.1. Труба 25 x 4												
			Профіль: 25 x 4; ГОСТ 8732 - 78*												
			Сталь: ВСт3пс4;												
9	1		0.00		0	0	0	0	0	0	0	0	0	0	3.11
9	2		0.00		0	0	0	0	0	0	0	0	0	0	3.11
11			Підібрано: 1.1.1. Труба 25 x 4												
			Профіль: 25 x 4; ГОСТ 8732 - 78*												
			Сталь: ВСт3пс4;												
11	1		0.00		70	0	0	0	0	73	0	70	0	73	3.11
11	2		0.00		70	0	0	0	0	73	0	70	0	73	3.11
12			Підібрано: 1.1.1. Труба 25 x 4												
			Профіль: 25 x 4; ГОСТ 8732 - 78*												
			Сталь: ВСт3пс4;												
12	1		0.00		70	0	0	0	0	73	0	70	0	73	3.11
12	2		0.00		70	0	0	0	0	73	0	70	0	73	3.11
14			Підібрано: 1.1.1. Труба 25 x 4												
			Профіль: 25 x 4; ГОСТ 8732 - 78*												
			Сталь: ВСт3пс4;												
14	1		0.00		0	0	0	0	0	0	0	0	0	0	3.11
14	2		0.00		0	0	0	0	0	0	0	0	0	0	3.11

Fig. 175 – Cross-section selection table

Questions for self-assessment

1. What is the purpose of the LIRA-CAD software?
2. Describe the capabilities of the LIRA-CAD software.
3. Name the main design systems of LIRA-CAD.
4. What is a diagram feature? Why is it important to choose it correctly?
5. What operations can be performed on the Design tab?
6. What reinforcement selection algorithms are implemented in LIRA when designing bar elements?
7. How to set up units of measurement in LIRA-CAD?
8. What loads should be calculated for buildings and structures?
9. What is the sequence for creating a calculation scheme?
10. What are the features of specifying calculation combinations of forces in LIRA-CAD?
11. How to generate tables of static calculation results?
12. How to assign material parameters for structural calculation?
13. How to design a frame transom? A column?
14. How to display reinforcement diagrams/mosaics of bar elements on the calculation scheme?
15. How to display the areas of longitudinal and transverse reinforcement for plate elements on the calculation diagram?
16. How to display the percentage of reinforcement of cross-sections and the width of crack openings on the calculation diagram?
17. How to generate tables of reinforcement selection results in the elements of the calculation diagram?
18. How to specify rigid inserts when creating a calculation diagram for a monolithic beam floor?
19. How to call the local reinforcement mode?
20. How to view and analyze the results of selecting steel truss element cross-sections?

Content module 2. Calculation and design of structures of multi-story monolithic frame buildings in the MONOMACH-CAD

Topic 7. Familiarization with the software complex, its capabilities and interface

Laboratory Work No. 13
Purpose and composition of the MONOMAX-CAD software package

Purpose and plan of the session

To familiarize students with the purpose of the MONOMAH-CAD software package and the programs included in it.

The MONOMAH software package is a typical representative of intelligent design systems. The package is designed for calculating and designing monolithic multi-story frame buildings. It also calculates brick buildings up to 14 stories high with monolithic reinforced concrete elements.

The MONOMAH PC consists of nine information-linked programs, each of which can operate in standalone mode: KOMPONOVKA, BEAM, COLUMN, FOUNDATION, RETENTION WALL, SLAB, SECTION (WALL), BRICK, SOIL. Figure 1 shows the components of the MONOMAX-CAD software package.

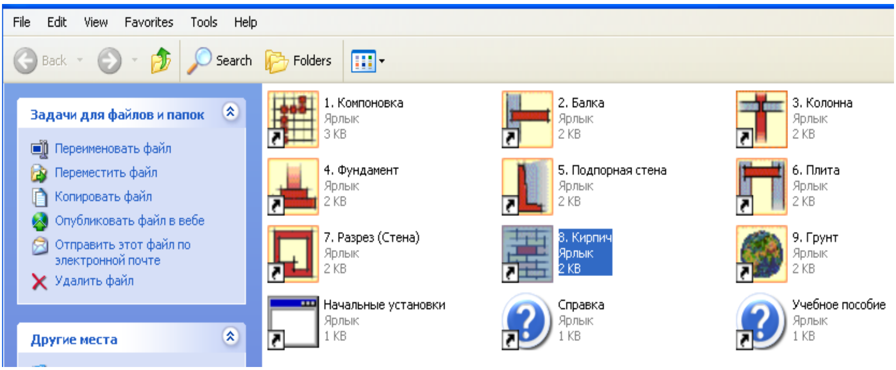


Fig. 1. Components of the MONOMAH-CAD software package

KOMPONOVKA – a root program for designing multi-story frame buildings made of monolithic reinforced concrete and buildings with brick walls. It allows you to quickly and conveniently create a building diagram in interactive graphic mode, set loads in their natural form, calculate the strength of building structural elements, select element cross-sections, determine the consumption of concrete, reinforcement, bricks, and other materials, estimate the cost of construction, and export data to other local programs.

Main functions of the program:

- creating a building model on an arbitrary grid plan with columns, beams, walls, partitions, floor slabs, foundation slabs, and piles;
- calculating brick buildings with load-bearing walls and combined frame buildings of increased height with brick infill;
- reducing the time required to create a model and perform variant design using various service capabilities - moving and rotating the coordinate system, copying, transferring, deleting one or a group of elements, modifying numerical values, copying floors;
- specifying vertical and horizontal loads on floor slabs in the form of distributed loads over the entire plane or on a section, as well as in the form of concentrated forces;
- specifying the direction of impact and information about the construction area to take into account wind and seismic loads;
- importing a soil model created by the GRUNT program;
- automatic generation of a building calculation scheme; performing static and dynamic calculations to determine displacement, force, and stress; selecting or checking element cross-sections; generating a list of material costs, frequency tables, and vibration periods. Animation of natural vibrations allows you to evaluate the accuracy of the created model;
- exporting data to the BALKA, KOLONA, FUNDAMENT, PLITA, ROZRIS (STINA), TSEGLA design programs, exporting loads on foundations to FOK-PC, and exporting the calculation scheme to PC LIRA.

BALKA is a program for designing monolithic reinforced concrete beams. The program allows you to design multi-span monolithic beams with variable cross-sections. The diagram is generated in import mode and in offline mode. Calculations are performed for groups I and II of limit states (based on crack opening). It is possible to construct an envelope diagram and find the required cross-sectional area of reinforcement. A diagram of materials is constructed. The beam is designed using welded frames or knitted reinforcement. Drawings are generated and a dxf file of the drawing is created.

COLUMN – a program for designing monolithic reinforced concrete columns of various cross-sectional shapes and pylons. The diagram is formed in offline mode and import mode. Calculations are performed for groups I and II of limit states with reinforcement selection and design.

RETAINING WALL – a program for designing and checking retaining walls. The program allows you to design a monolithic reinforced concrete corner retaining wall for the specified engineering and geological conditions of the construction site. A massive retaining wall is checked. The required reinforcement area is determined and the design is also performed. The diagram is generated in offline mode. A drawing is made and a dxf file of the drawing is created**.**

FUNDAMENT – a program for designing monolithic columnar reinforced concrete foundations on a natural basis. The program allows you to design a foundation for a column, calculate the necessary reinforcement, and perform the design. A drawing is made and a dxf file of the drawing is created.

SLAB – a program for designing monolithic reinforced concrete floor slabs and foundation slabs. The operation of the slab in the general frame is implemented by transferring displacements at the junction nodes. Based on the calculation results, stress fields and diagrams are constructed for a given segment. The stress fields under the base of the foundation slab and the force mosaic in the piles are also constructed.

SECTION (WALL) – a program for designing reinforced concrete monolithic and brick walls. The program allows you to design monolithic reinforced concrete walls of any shape in conjunction with adjacent frame structures. Variable wall thickness and the presence of openings are taken into account. The wall's role in the overall frame is taken into account by transferring displacements at the CE joints. Based on the calculation results, force fields and diagrams for bar elements are constructed. The cross-sectional area of the required reinforcement is determined, and the wall is reinforced with meshes and bars.

BRICK is a program for designing walls of brick buildings. During the calculation process, the required number of meshes and the selection of vertical reinforcement bars are determined. The diagram is generated in import mode from the KOMPONOVKA program.

Topic 8. Creating a model and calculating a multi-story monolithic frame building in the KOMPONOVKA program

Task. Create a model of a multi-story public building according to an individual task, perform a static calculation. Perform a detailed analysis of the calculation of the main structural elements.

Design guidelines

We will create the building's calculation scheme in the KOMPONOVKA program. This is the core program for designing multi-story frame buildings made of monolithic reinforced concrete and buildings with brick walls. It allows you to quickly and conveniently create a building layout in interactive graphic mode, set loads in their natural form, calculate the strength of structural elements, select element cross-sections, determine the consumption of concrete, reinforcement, bricks, and other materials, estimate the cost of construction, and export data to other local programs.

At the first stage of work, the main characteristics of the building are specified (planning mark, soil characteristics, materials of structural elements). After that, a typical floor plan is constructed, columns (pylons), walls, partitions, existing openings, loads on floor slabs and roofs are specified.

The shape of the building in the plan can be arbitrary. To form a spatial planning scheme in height, use the floor copying function with subsequent adjustment of the scheme on any floor. The structural scheme of a building with load-bearing brick walls can include concrete and reinforced concrete walls, columns, beams, i.e., create a combined frame with brick filling.

To correctly collect loads from the dead weight of structures, the user must assign the appropriate material from the list of materials to them.

A detailed description of the process of constructing a calculation diagram is given below. You can also use the MONOMAX-CAD electronic training manual.

Input data for creating a calculation model in the KOMPONOVKA program

The input data for designing a building in the KOMPONOVKA program are:

- floor plans (basement, first floor, typical floor),
- sections (with elevation marks),
- information about the structural design of the building, main load-bearing elements (location on the plan, cross-sections, and materials),
- dimensions of window and door openings,
- permanent and variable loads on floors and roofs,
- construction area (to determine wind and snow loads),
- geological conditions of the construction site.

Load collection

The loads acting on floors and roofs are constant and variable. Constant characteristic loads include the weight of the floor or roof. The value of the characteristic temporary load is calculated by multiplying the thickness of a given layer of material by its volume mass (density). The dead weight of a monolithic reinforced concrete floor slab (roof) is automatically taken into account by the program when modeling the calculation scheme, so there is no need to specify it again. The design values of loads are calculated by multiplying the standard values by the reliability factors for load action and by the responsibility class of the structure . The calculation is summarized in tables.

The characteristic and quasi-constant values of uniformly distributed variable (temporary) loads on floor slabs (depending on the purpose of the premises) are given in Table 6 of Appendix A (or in Table 6.2 [14]).

The characteristic values of loads and effects (wind and snow loads) for cities in Ukraine are given in Table 11 of Appendix A (or in Appendix E [14]).

Table 1. Total load per 1 m^2 of floor slab

№	Load name	Characteristic load, kPa	coefficients		Design load, kPa
			γ_{fm}	γ_n	
	Permanent load				
1	Ceramic tiles on cement-sand mortar, δ = 0,015 м, ρ_m = 1900 кг/м^3	0,285	1,1	0,95	0,298
2	Cement-sand screed, δ = 0,02 м, ρ_m = 1800 кг/м^3	0,360	1,2	0,95	0,410
3	Insulation δ = 0,05м, ρ_m = 90 кг/м^3	0,045	1,3	0,95	0,056
4	Cement-sand screed, δ = 0,02 м, ρ_m = 1800 кг/м^3	0,360	1,2	0,95	0,410
5	Reinforced concrete floor slab, δ = 0,16 м, ρ_m = 2500 кг/м^3	Load is automatically taken into account by the program			
Total:		1,05			1,174
	Variable load, including:	1,50			1,710
	quasi-permanent	0,35	1,2	0,95	0,399
	short-term	1,15	1,2	0,95	1,311
	Full load	2,55			2,884

Table 2. Total load per 1 m^2 of coverage

№	Load name	Characteristic load, kPa	coefficients γ_f	coefficients γ_n	Design load, kPa
	Permanent load				
1	Rolled carpet, δ = 0,015 м, ρ_m = 600 кг/м^3	0,120	1,1	0,95	0,125
2	Cement-sand mortar screed M100, reinforced with mesh 150×150 діам. 4 Bp-I, δ = 0,04 м, ρ_m = 1800 кг/м^3	0,720	1,2	0,95	0,821
3	Insulation - slabs mineral wool, δ = 0,25 м, ρ_m = 100 кг/м^3	0,250	1,3	0,95	0,185
4	Vapor barrier film	0,014	1,2	0,95	0,016
5	Cement-sand screed, δ = 0,02 м, ρ_m = 1800 кг/м^3	0,360	1,2	0,95	0,410
6	Reinforced concrete floor slab coverage, δ = 0,16 м, ρ_m = 2500 кг/м^3	Load is automatically taken into account by the program			
Total:		1,464			1,557
	Variable load				
	snow	1,240	1,14	0,95	1,414
	Full load	2,704			2,971

Table 3. Collection of loads from external walls

№	Load name	Characteristic load, kPa	coefficients γ_f	coefficients γ_n	Design load, kPa
	Permanent load				
1	Lime-sand mortar, δ = 0,02 м, ρ_m = 1600 кг/м^3	0,320	1,2	0,95	0,365
2	Lightweight concrete blocks, δ = 0,2 м, ρ_m = 400 кг/м^3	0,800	1,2	0,95	0,912
3	Insulation – slabs mineral wool, δ = 0,05 м, ρ_m = 90 кг/м^3	0,045	1,3	0,95	0,056
4	Brickwork made of ceramic hollow bricks on cement-sand mortar δ = 0,12 м, ρ_m = 1200 кг/м^3	1,440	1,2	0,95	1,642
Total:		2,605			2,975

The load on 1 m of the floor slab from the wall is determined depending on the height of the floor (for example, 3 m):

- *constant characteristic* $2{,}605 \cdot 3 = 7{,}815$ ***kN/m;***

- *constant design* $2{,}975 \cdot 3 = 8{,}925$ ***kN/m.***

Laboratory Work No. 14

Creating a calculation model of a building

Purpose and plan of the session

Create a new task and assign general characteristics to the building. Assign materials to the elements of the calculation scheme. Set the coordinate axis grid for the projected building. Arrange vertical structural elements—columns, pylons, and walls—on the plan. Model openings in the walls.

Creating a new task and assigning general building characteristics

To start working with the MONOMAH software package's LAYOUT program, run the **Start – Programs – Monomakh-SAPR – 1. KOMPONOVKA** command. The program will automatically create a new document, so no additional actions are required to create a new project.

To set general data about the building, use the **Scheme - Building Characteristics** menu to open the **General Building Characteristics** dialog box, where you can set:

- the layout elevation (ground elevation);
- the top of the column elevation (basement floor elevation);
- the base elevation (foundation depth);
- soil characteristics – specified (characteristics of the bearing layer – soil bulk weight, internal friction angle, adhesion, deformation modulus, Poisson's ratio). Attention! Be careful with the units of measurement for these values;
- the rest of the parameters can be left at their default values (Fig. 2).

To save the data, click the OK button.

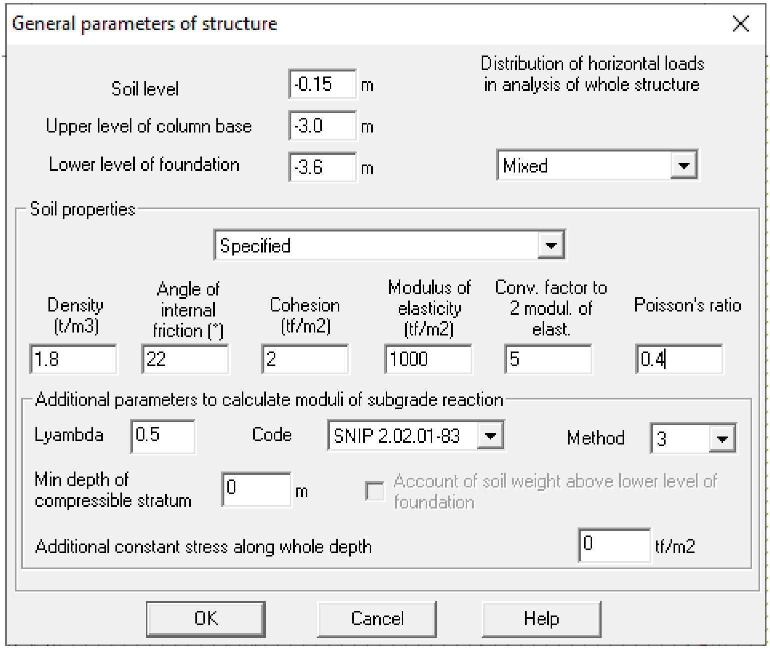


Fig. 2 – *General parameters of strukture* dialog box

The name of the object is specified using the **File - Object Name** menu. The name may include the student's surname and the name of the object.

Assigning material characteristics to building structural elements

Using the **Scheme - Materials** menu, open the **Materials** dialog box, in which you can specify the characteristics of materials for all structural elements (Figs. 4, 5). If the assignment specifies the main load-bearing elements of the building made of monolithic reinforced concrete, you can specify them as follows:

- ✓ foundations - concrete class C12/15, working reinforcement - class A400C, structural - class A240C;
- ✓ columns (pylons) - concrete class C16/20 - C20/25 (depending on the number of storeys of the building), working reinforcement - class A400C, structural - class A240C;
- ✓ floor slabs (coverings) - concrete class C20/25, working

reinforcement - class A400C, A500C structural - class A240C.

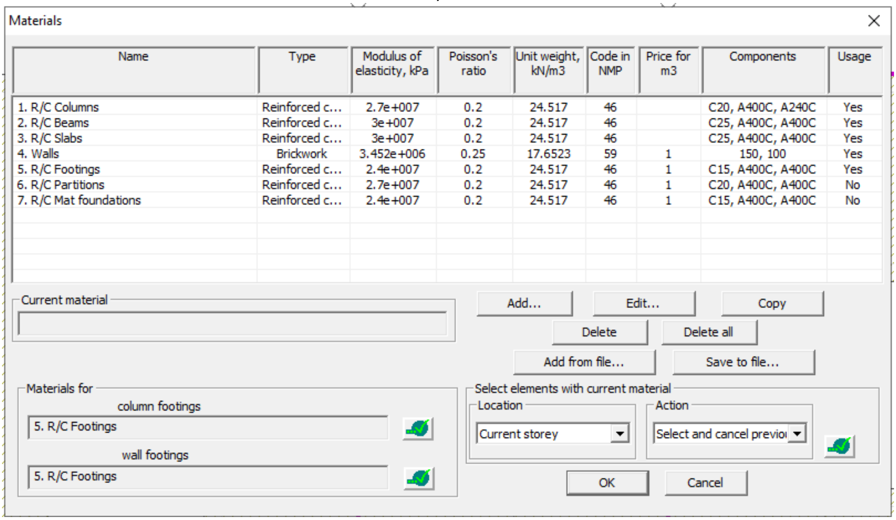


Fig. 4 – Materials dialog box

If the building is designed as a frame, then external non-load-bearing or self-supporting walls can be left unmodeled, and the load from them can be specified as a linear load on the floor at the point where the walls rest (along the contour of the floor slab).

If the building is designed with an incomplete frame or frameless at all, with brick load-bearing walls, then the wall material is specified as follows:

- in the **Materials** dialog box (Fig. 4), click the **Add** button;
- in the **Type** field (Fig. 5), select “Masonry”, specify the brick and mortar grades, price per m3 in the appropriate fields, and click the “Calculate modulus of elasticity” button.

To save the data, click the OK button.

Setting the construction grid and coordination axes of the building

Open the **Cartesian Grid** dialog box (Fig. 6) using the **Scheme - Grid - Add Cartesian Grid Fragment** menu, set the axis spacing in the X and Y directions in accordance with the floor plan of the designed building. Click OK.

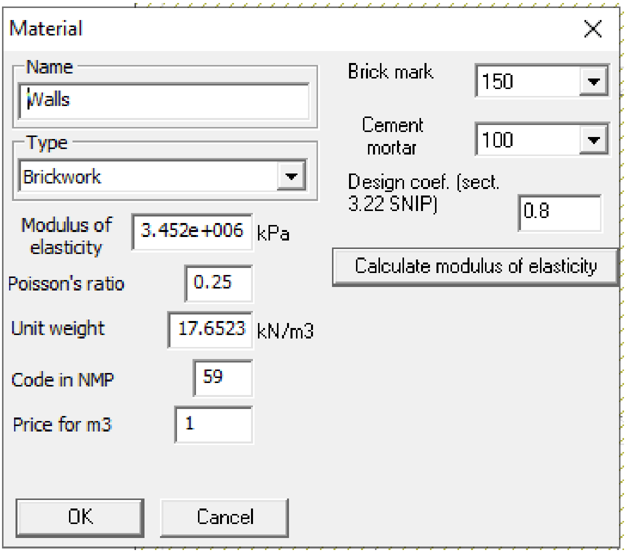


Fig. 5 – Dialog box for specifying brickwork for walls

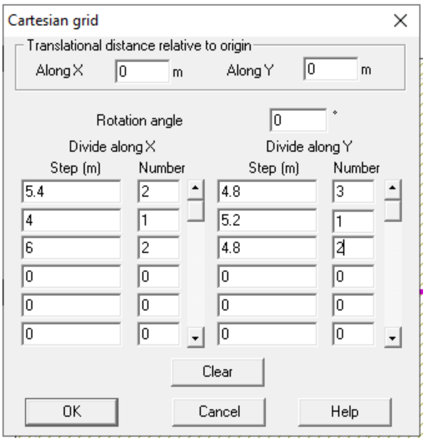


Fig. 6 – Cartesian Grid dialog box

Set the position of the coordination axes of the building using the **Scheme - Add elements - Add axis** menu (button  on the toolbar). In the **Add axis** dialog box, set the name of axis 1 (by default - a1), click the mouse button on two grid nodes so that the specified points determine the

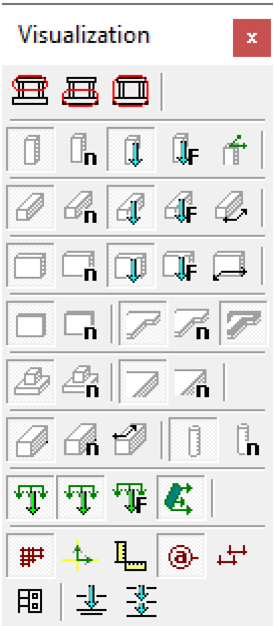
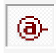


Fig. 7 – Visualization toolbar

position of the coordination axis. Consistently, setting the required axis names in the **Add axis** dialog box, specify the positions of axes 2, 3, 4, etc. according to the building plan.

To display the created axes on the screen, you can use the button  to call the **Visualization** toolbar (Fig. 7) and click the button .

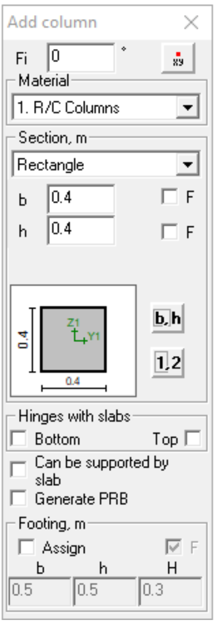


Fig. 8 – Add Column dialog box

Creating and placing columns on the plan

Set the parameters and position of the columns using the **Scheme - Add elements - Add column** menu (button  on the toolbar). In the Add column dialog box (Fig. 8), set the following parameters:

- material - select the one intended for the column from the list of materials;
- cross-section - rectangle (or other);
- fill in the fields related to the cross-section dimensions (to make sure that the dimensions are set correctly, click the button  to display them).

To place a column on the plan, select the single selection cursor using the **Scheme – Element Selection – Single Mark Cursor** menu (button  on the toolbar). Indicate on the scheme the node at the intersection of the axes according to the building plan.

You only need to click the mouse once - otherwise you will place several columns in one place, which will lead to errors when running the scheme for calculation.

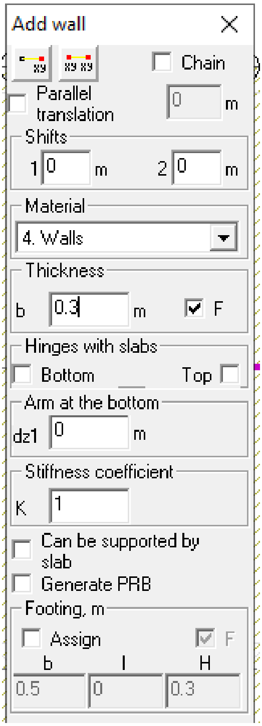
Saving model information

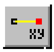
To save model information, select the File – Save menu item (button on the toolbar).

Creating and placing walls (pylons) on the plan

Pylons (short walls) can be modeled as columns with a rectangular section or as short walls (up to 3 meters long). Set the parameters and position of the pylon using the **Scheme - Add elements - Add wall** menu (button  on the toolbar). In the Add wall dialog box (Fig. 9), set the following parameters:

- ➢ material – select from the list of materials the one intended for the wall (pylon);
- ➢ thickness b;
- ➢ support conditions, etc.



Indicate in pairs on the diagram the nodes at the intersection of the axes where the walls (pylons) are located, according to the building plan. If you need to specify an additional node for modeling a short wall that is not located at the intersection of the axes, you can use the button  — **Specify node coordinates.**

In the dialog box (Fig. 10), specify the node coordinates (the **Absolute** option is selected by default); click the Apply button .

Fig. 9 – Add Wall dialog box

Fig. 10 – Dialog box for specifying the coordinates of one node

If it is necessary to specify two additional nodes (for example, if the wall is offset relative to the axis), you can use the button  — **Specify coordinates of two nodes** (Fig. 11).

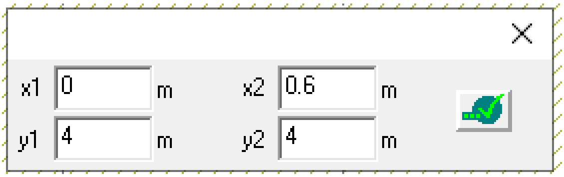


Fig. 11 – Dialog box for specifying the coordinates of two nodes

Modeling openings in walls

To model openings in walls (partitions), you should first add as many standard sizes of windows and doors to the **Openings Database** as the project requires. To do this, use the Tools menu **- Openings Database**. In the **Openings Database** dialog box (Fig. 12), perform the following actions:

- click the Rectangular button — the Opening dialog box will open (Fig. 13), in which you can specify the window (door) parameters;
- click the OK button — the specified opening will be added to the end
- in the opening list group;
- click on the name of the specified opening in the list ribbon (by default Contur1) — its contour will be displayed in the viewing window;
- click on the name of the specified opening again and change the name of the opening to D1 (or B1) – depending on whether it is a door or a window. It is worth entering the dimensions of the given opening here, so that later it will be easier to navigate in the opening database.

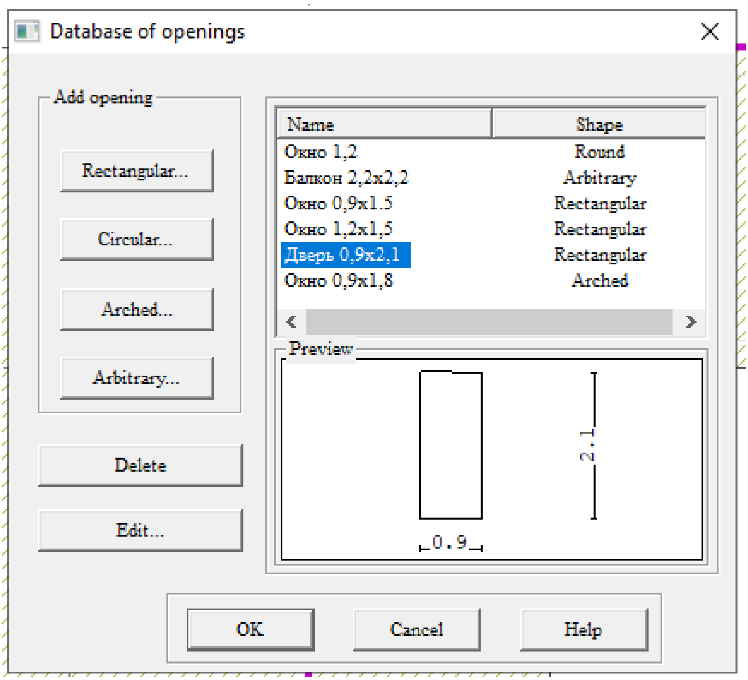


Fig. 12 – *Hole Database* dialog box

To specify a hole in the wall, use the menu **Scheme — Adjustment — Hole in wall (partition)** (button  on the toolbar). After activating this mode, specify the wall in the scheme.

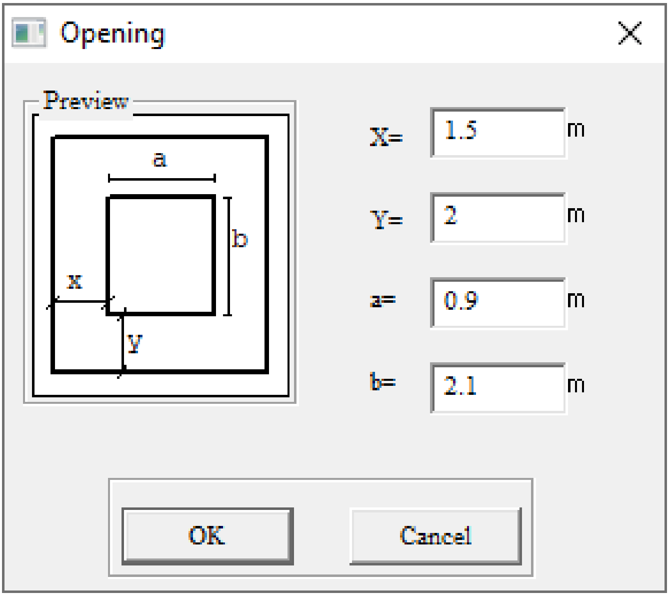


Fig. 13 – *Opening* dialog box

In the **Wall Holes** dialog box (Fig. 14), perform the following actions:

- in the **Add** group, click the From database button — the Add hole **from database** dialog box will open;
- select the desired hole from the list and specify the coordinates of the hole insertion point in the wall;
- click the OK button — the specified hole will be added to the wall drawing in the Wall Holes dialog box;
- click the OK button.

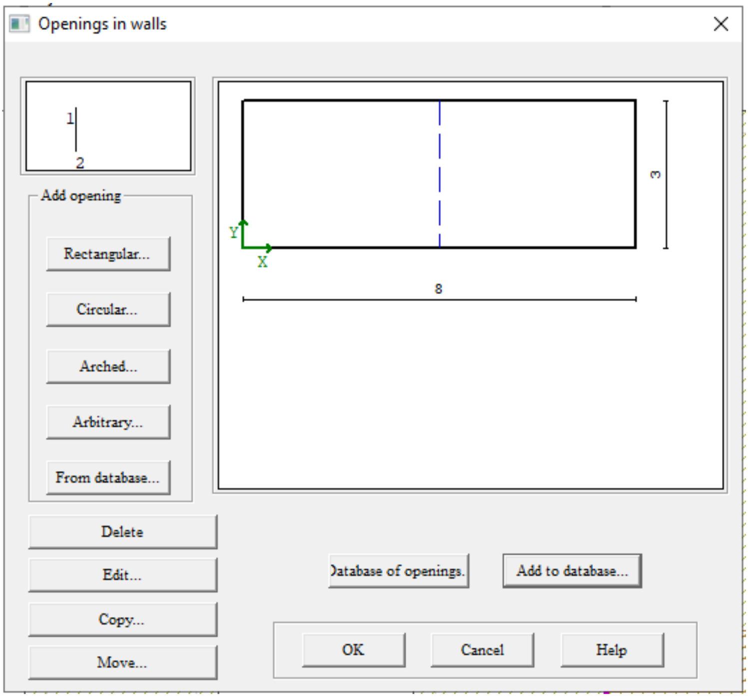
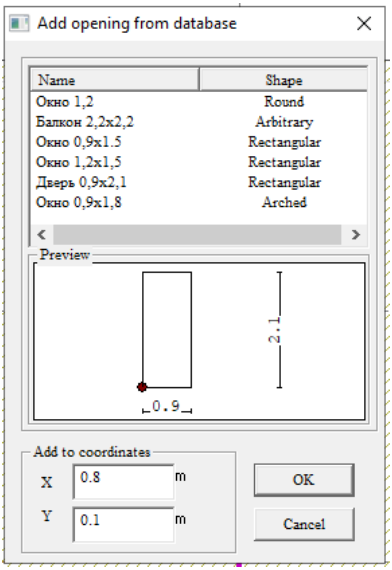


Fig. 14 – *Wall Holes* dialog box



In the same way, specify all the openings on the building plan.

Fig. 15 – Add Hole from Base dialog box

Laboratory Work No. 15

Modeling of floor slabs

Purpose and plan of the session

Define the contour of the slab. Adjust the contour by specifying new nodes. Model the holes in the slab. Specify the loads. Specify the reliability and duration coefficients of the loads.

Specifying the floor slab contour

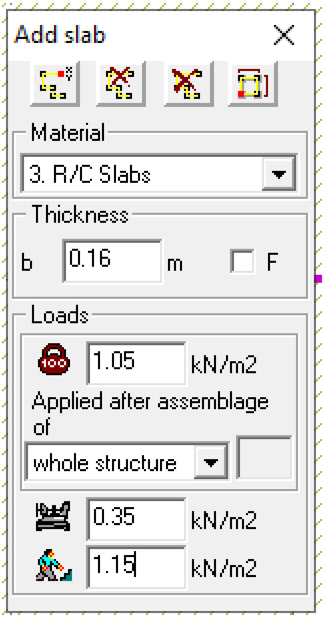


Fig. 16 – Add Slab Dialog Box

Specify the parameters and contour of the floor slab using the **Scheme - Add Elements - Add Slab** menu.

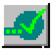
In the Add Slab dialog box (Fig. 16), set the following parameters:

- material – select the one intended for the floor slab from the list of materials;
- slab thickness b;
- load on the slab (permanent, short-term, quasi-permanent).

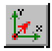
Please note that the load evenly distributed over the entire slab is set as a slab parameter. The slab's own weight is generated automatically by the program, so it does not need to be taken into account in the permanent load.

Specify the contour of the floor slab – sequentially indicate the intersection points of the axes on the diagram according to the building plan.

If you need to specify a node that is outside the grid nodes, then in the **Add Slab** dialog box, click the button  — **Specify node coordinates**. In the dialog box (Fig. 10), select the desired option (absolute or relative) and specify the node coordinates.

Click the button  — Apply.

Adjusting the floor slab contour – adding new nodes and moving existing ones

You can change the floor slab contour (for example, add a balcony ledge) using the **Scheme menu — Adjustment — Move or add contour node** (button  on the toolbar). To make it easier to specify the coordinates of additional nodes, move the origin of the coordinate system to the node closest to the desired one at the intersection of the axes using the **Scheme menu — Coordinate system — Move** (button  on the toolbar).

Modeling openings in the slab

Openings in the slab are defined using the menu **Scheme — Add elements — Add opening in the slab** (button  on the toolbar). Specify the opening nodes on the diagram according to the building plan.

If there are not enough nodes to model the hole, use the button — Specify node coordinates (far left in the **Add Hole** dialog box (Fig. 17).

Fig. 17 – Add Hole Dialog Box

A rectangular hole can be defined by coordinates using the button  — **Define Rectangular Hole** (far right in the **Add Hole** dialog box). In the window that opens, define the coordinate of the first node and the X, Y increment (Fig. 18).

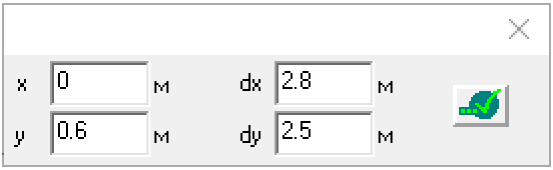


Fig. 18 – Dialog box for specifying the coordinates of the nodes of a rectangular opening

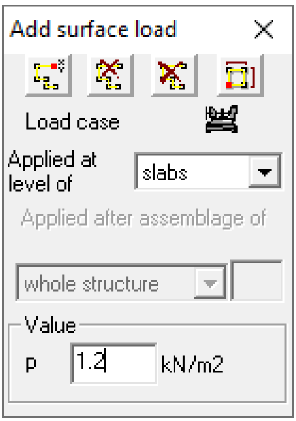
Specifying loads on slabs

Considering the self-weight of elements

The self-weight of elements as an external load does not need to be specified – it is automatically taken into account based on the specified volumetric mass of materials. The load from the self-weight of elements is included in the constant load.

Modeling of load stamps

When modeling the floor slab, a uniformly distributed load on it was already specified. At the same time, the constant load included the weight of the floor, the variable short-term load from people, and the variable long-term (quasi-permanent) load from furniture, equipment, etc. The variable load was taken into account according to the purpose of the building (see Table 6 of Appendix A). If there are premises for different purposes on one floor of the designed building (for example, offices, assembly halls, archives, etc.), then it is necessary to take into account the difference in loads, specifying it for specific premises in the form of a load stamp.



Make sure you have selected the required loading – short-term or long-term (quasi-permanent) – by clicking the or button ,  on the toolbar, respectively.

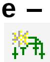
The load stamp is set using the menu **Scheme – Add elements – Add load stamp** (button  on the toolbar).

Fig. 19 – Add Load Stamp Dialog Box

In the **Add Load Stamp** dialog box (Fig. 19), you need to enter the difference between the collected load on the given area and the load already applied to the floor slab.

The load on balconies and loggias can also be specified as a stamp or linear evenly distributed over a 0.8 m wide area along the fence.

Linear Load Modeling

The load from external non-load-bearing or self-supporting walls can be specified as a linear load on the floor slab at the point of support of the walls - along the contour of the slab (Fig. 20).

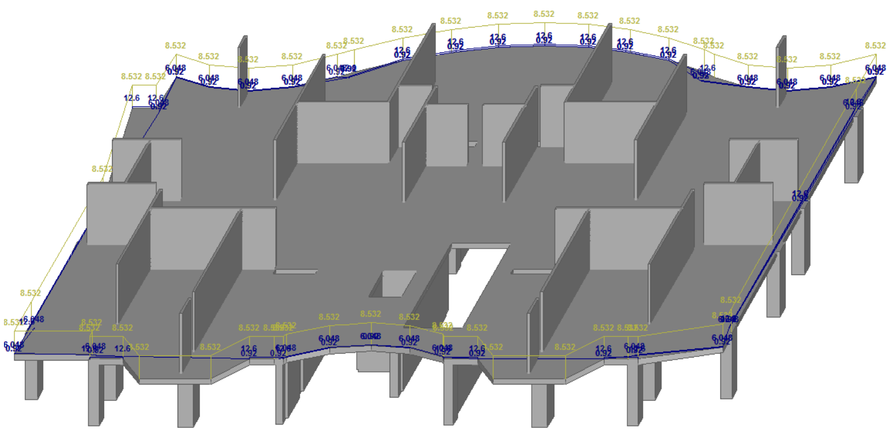
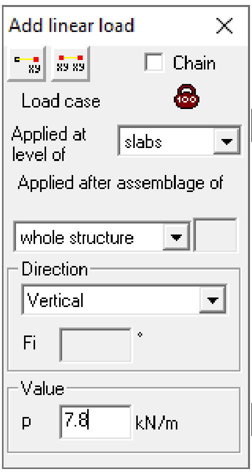


Fig. 20 – Linear load on the floor from external walls



A linearly distributed load is specified using the **Scheme – Add elements – Add linear load** menu (button on the toolbar).

In the **Add linear load** dialog box (Fig. 21), you need to specify its value (running load!).

Attention! First, make sure that the selected one is required for the duration of the load!

Fig. 21 – Add Line Load dialog box

Specifying reliability and duration coefficients

Loads according to DBN V.1.2-2:2006 [4] in the MONOMAKH PC are recommended to be specified as follows:

• to apply operational values of loads to the calculation scheme;

• in the dialog box **Wind according to DBN**, specify the reliability coefficient according to the operational value $\gamma_{fe} = 0{,}21$ (so that operational values of wind loads are applied to the scheme);

• in the **Coefficients** dialog box, called by the **Loads - Load coefficients** command, specify reliability coefficients as the ratio of coefficients according to the limit value to coefficients according to the operational value ($\gamma_{fm} / \gamma_{fe}$) of loads.

Thus, operational loads according to DBN correspond to the normative ones according to SNiP, and the limit loads according to DBN correspond to the calculated ones according to SNiP.

With this approach, in the design programs of the PC MONOMAKH, limit loads are used to calculate the strength of reinforcement, and operational loads are used to calculate the crack resistance of structures.

Recommended values of load coefficients are given in the **Coefficients** dialog box (Fig. 22).

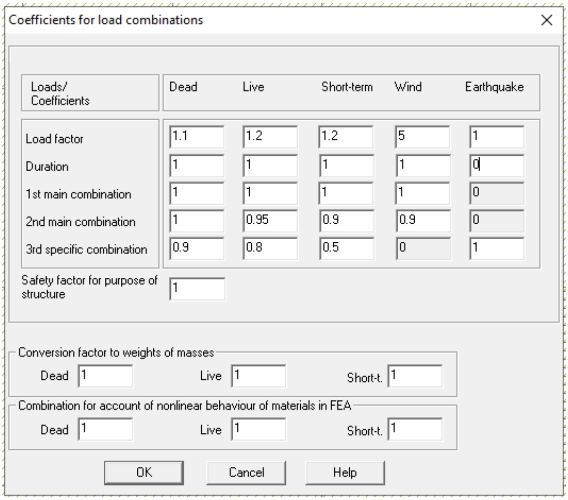


Fig. 22 – Coefficients dialog box

Laboratory Work No. 16

Calculation and adjustment of floors

Purpose and plan of the session

Perform a floor calculation. Perform floor copying. Change the height of floors. Perform floor adjustments.

Floor calculation

The KOMPONOVKA program provides for the execution of two types of calculations - preliminary (simplified) and MSE calculation. The purpose of the preliminary calculation is to identify the structural scheme of the building, collect loads, check or select cross-sections of structural elements. The preliminary calculation consists of a series of calculations and is performed using the **Calculation - Calculation of the current floor** and **Calculation - Calculation of the entire building** menus.

During the calculation, the program performs diagnostics of the created model, displays the detected errors in a dialog box. If you select a ribbon in the list of errors, the element that caused the error will be highlighted in red on the diagram. If several floors have the same configuration and load, then create one floor, perform its calculation using the **Calculation - Calculation of the current floor** menu, and then copy it to other floors. In this case, both the floor scheme and the calculation results are automatically copied. This significantly reduces the time for calculating the building.

So, to check the created scheme, perform the floor calculation using the **Calculation – Current floor calculation** menu (button  on the toolbar).

The calculation results are shown in Fig. 23.

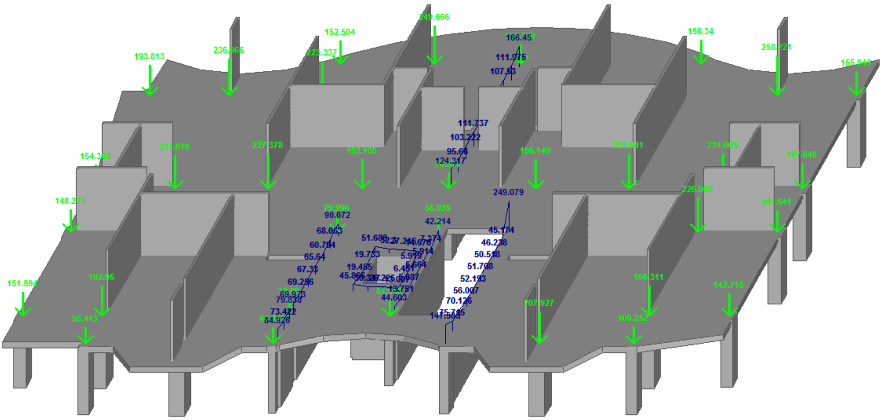


Fig. 23 – Results of preliminary calculation of a typical floor in 3D

Copying a floor

To form a spatial-planning scheme along the height of the building, use the function of copying floors (**Floor - Copy floor**) with subsequent adjustment of the scheme on the first (basement) and last (technical) floors. If the planning scheme of other floors differs significantly from the typical one, it is advisable to make adjustments on these floors as well.

Copy a typical floor using the menu **Floors - Copy floor** (button  on the toolbar). In the dialog box **Copy current floor** (Fig. 24), specify the data from which floor to copy. After that, click on the OK button. Now the specified model will consist of 13 identical floors

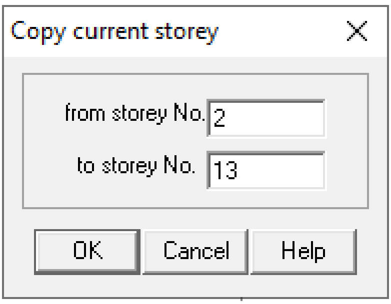


Fig. 24 – Copy Over dialog box

Changing the floor height

To call the desired floor, click on the corresponding selection button  on the toolbar.

The floor height can be changed using the **Floors — Floor Properties** menu. In the **Current Floor Properties** dialog box (Fig. 25), you can change the floor name and set a different height.

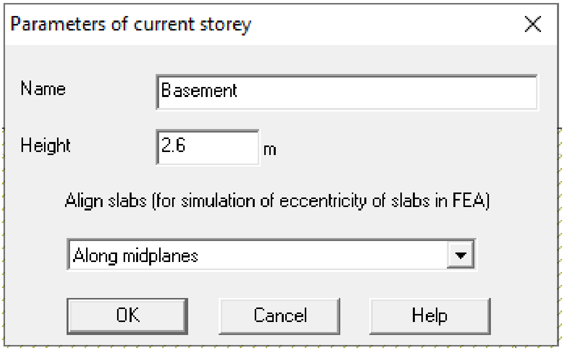


Fig. 25 – Floor Characteristics dialog box

Floor Adjustment

Perform floor adjustments according to your task. For example, on the first (basement) floor, the load-bearing walls may have different parameters (thickness, material), may be located slightly differently, etc. On the last (technical) floor, it is worth removing all partitions and linear loads from the walls, openings in the slab for the staircase, etc.

Attention! The load on the floor slab will be slightly different than on the floor slabs (see the load collection tables).

To make adjustments, click on the button with the number of the desired floor, for example, . In our case, this is the last floor of the building. To remove all partitions and line loads from the external walls, use the menu **Scheme — Select elements — Select elements by criteria** (button  on the toolbar).

In the Select elements by criteria dialog box (Fig. 26), perform the following actions:

- click on the  — Partitions tab;

- click the — Apply button;
- click on the — Linear loads tab;
- select the — Selection action from the list so that the sequential selection of partitions is not canceled;
- click the — Apply button.

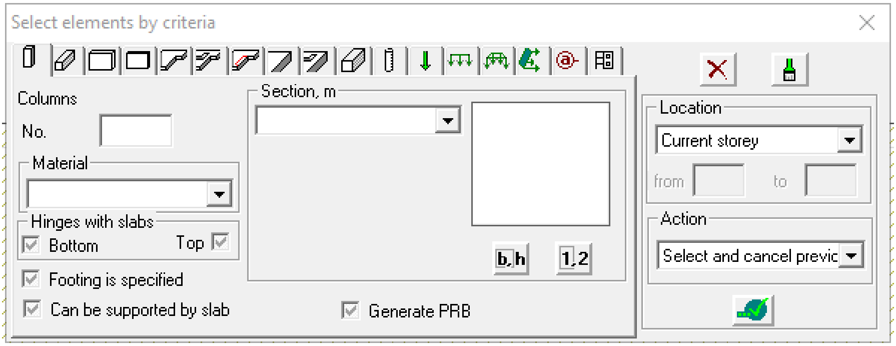


Fig. 26 – Select elements by criteria dialog box

Close the **Select Elements by Criteria** dialog box. Delete the selected elements using the **Diagram — Delete Elements** menu (button on the toolbar).

To delete holes, load stamps, or other elements on the floor, use the **Diagram — Element Selection — Select Elements** menu (button on the toolbar). After activating this mode, select these elements on the diagram and delete them using the **Diagram — Delete Elements** menu (button on the toolbar).

If you need to change the load on the slab, use the menu **Scheme — Adjustment — Element Properties** (button on the toolbar).

To copy and move elements, use the menu **Scheme — Adjustment — Copy and Move** (button on the toolbar) – Fig. 27.

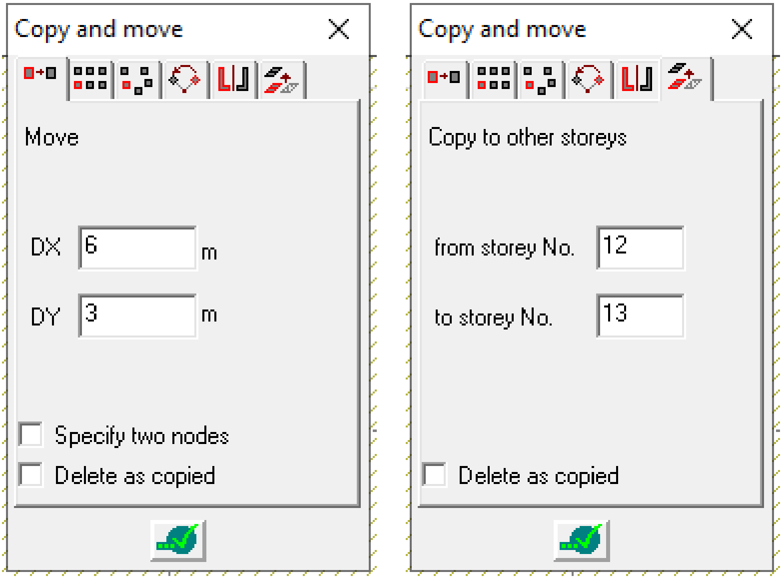


Fig. 27 – *Copy and Move* dialog box

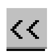
Laboratory Work No. 17

Modeling of Foundation Slabs

Purpose and plan of the session

To model foundation slabs, view the 3D model of the building, and adjust the design scheme.

Modeling Foundation Slabs

Make the first floor active by clicking on the floor selection buttons  and  on the toolbar.

Set the parameters and contour of the foundation slab using the menu **Scheme - Add elements - Add foundation slab** (button  on the toolbar)

In the **Add foundation slab** dialog box (Fig. 28), set the following parameters:

- overhang – 0.5 m;
- material – select the one intended for the foundation slab from the list of materials;
- slab thickness b (minimum recommended thickness – 0.6 m);
- load on the slab (constant, short-term, quasi-constant).

Define the contour of the foundation slab - consistently indicate the intersection points of the axes on the diagram according to the building plan.

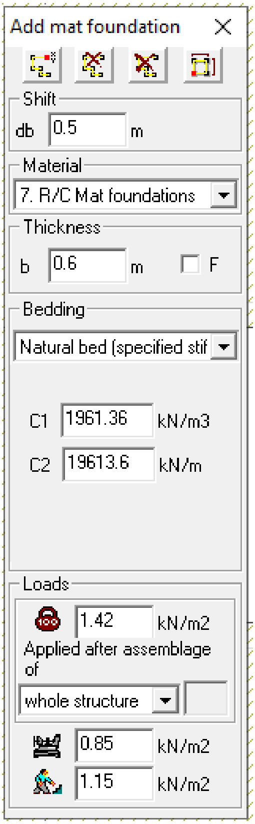


Fig. 28 – Add Base Slab dialog box

Viewing the 3D view of the model

View the 3D view of the created model (Fig. 29) using the **View menu - 3D View - Entire Building** (button  on the toolbar).

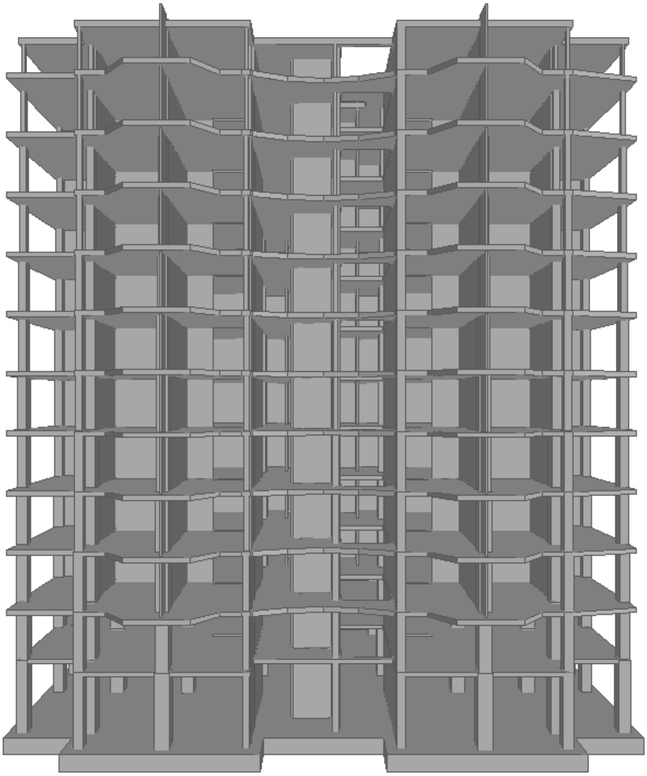


Fig. 29 – Three-dimensional image of the designed building (perspective)

- Switch the axonometric image to perspective using the **View — Projection — Perspective** menu (button on the toolbar).
- Move the model image down by pressing the PageUp key several times.
- Zoom out by pressing the v key several times.
- Rotate the model image around its vertical axis by pressing the or buttons on the toolbar.
- Return to the main view using the **View — Main View** menu (button on the toolbar).

Laboratory Work No. 18

Performing calculations of the entire building. Creating a calculation note and exporting to design programs

Purpose and plan of the session

Perform a preliminary calculation of the entire building. Display the specified loads and the results of the floor calculation. Perform the final calculation using the finite element method (FEM calculation).

Generate and prepare for printing a calculation explanatory note based on the FEM calculation results. Export data to the design programs of the MONOMAKH-SAPR PC.

Calculation of the entire building

Perform a preliminary calculation of the entire building using the **Calculation – Calculation of the entire building** menu (button on the toolbar).

You can view the calculation results in a three-dimensional image (Fig. 33) using the **View – 3D View – Entire building** menu (button on the toolbar). Enable the display of the specified loads and calculation results using the **View – Display** menu. In the **Display** dialog box, perform the following actions:

- on the active tab — **Elements**, check the box for the Load option;
- click on the tab — **Load** results;
- check the box for the **Floor calculation results** option;
- click the button — **Apply**.

This can be done by clicking the — **Loads** and — **Floor calculation results** buttons (Visualization toolbar), and then the — Redraw button.

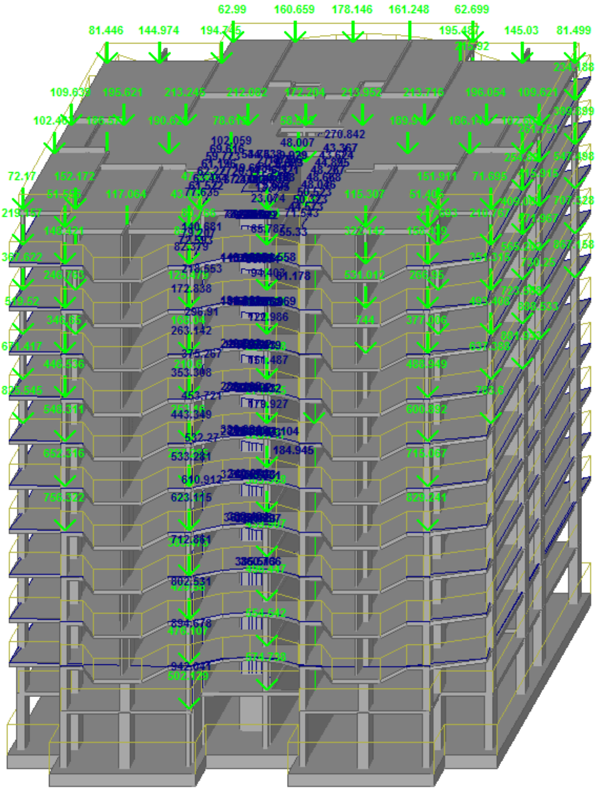
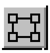


Fig. 33 – Three-dimensional image of the designed building with the results of the preliminary calculation

You can return to the main view using the **View — Main view** menu (button  on the toolbar).

You can view the calculation results of each floor by clicking on the floor selection buttons .

The calculation results are saved using the **File — Save** menu.

The calculation and explanatory note is generated based on the results of the preliminary calculation of the entire building using the **Results — Calculation Note — Calculation Note** (rtf file) menu. In the Calculation

Note dialog box, select the elements of the diagram whose calculation data you want to view or print.

FEM calculation

FEM calculation (by finite element method) is mandatory, and the final results are accepted according to it. The calculation is performed using the **Calculation — FEM calculation** menu (button  on the toolbar). To do this, a finite element scheme is automatically generated taking into account the specified parameters in the **FEM calculation** dialog box (Fig. 34). For the upper floors of a multi-storey building, the triangulation step of slabs and walls is taken as enlarged (3 m each), and for the lower three floors, where greater calculation accuracy is required, it is 1.5 m each. To do this, in the FEM calculation dialog box, using the **Set unique floors** button, the dialog box of the same name is called (Fig. 35).

The calculation task is started by clicking the button 

in the **MSE calculation** dialog box (Fig. 34).

The calculation processor window will display the calculation scheme, the main characteristics of the scheme, and the calculation protocol during the calculation.

After calculating the MSE, the expert system checks the sections of reinforced concrete elements for the forces obtained. The detected violations are displayed in a dialog box (this concerns the reinforcement of sections).

The results of the MSE calculation are viewed using the **View menu — MSE calculation results** (Fig. 36). The deformed scheme is displayed on the screen using the **Results** menu — **Deformed scheme**. Displacements are analyzed using the **Results menu — Displacement isofields**, stresses – using the **Results menu — Stress and force isofields**. To view information about a specific element or node of the scheme, use the **Results menu — Information about element or node** (button  on the toolbar), point to the element or node and get the necessary information.

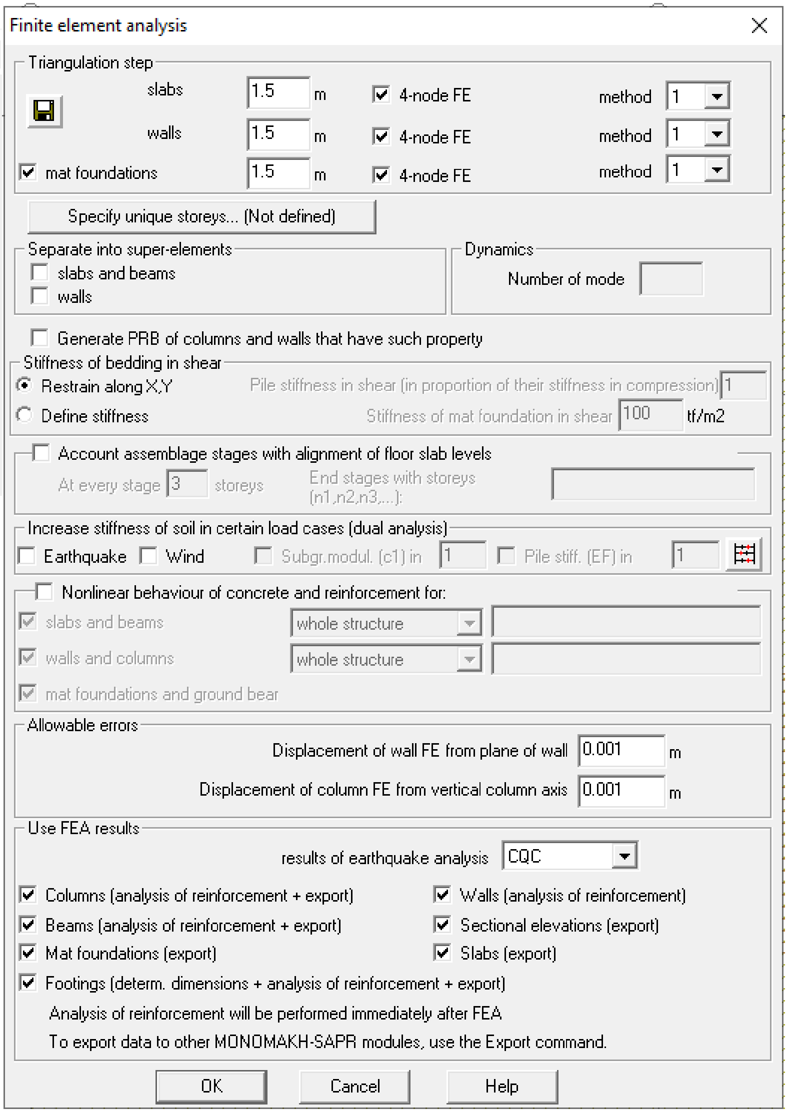


Fig. 34 – MFE Calculation dialog box

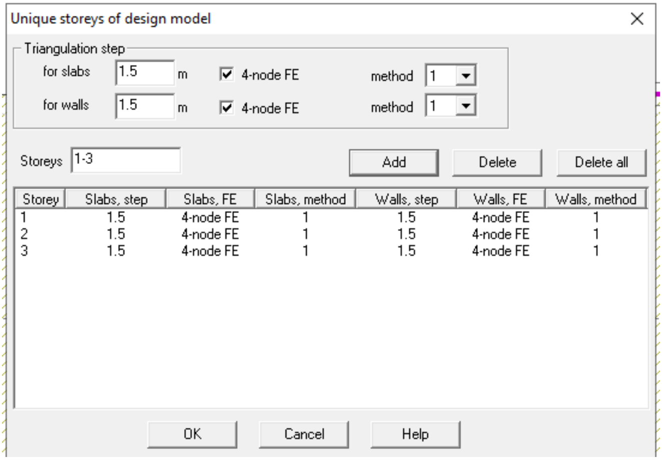


Fig. 35 – *Unique floors of the calculation scheme* dialog box

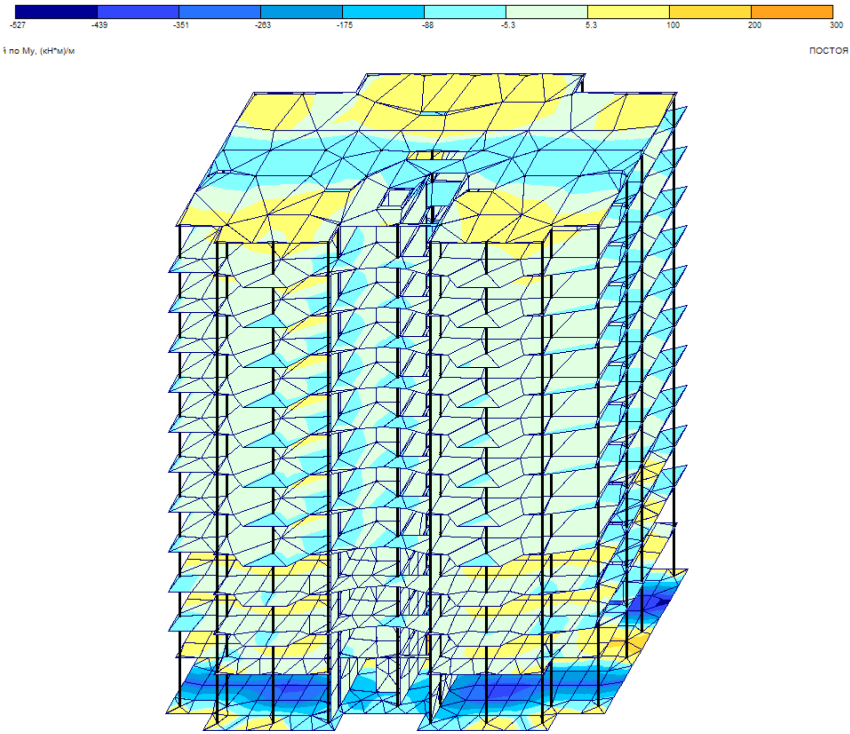


Fig. 36 – Results of MFE calculation

Formation of an explanatory note

A calculation note based on the results of the MSE calculation is formed using the **Results menu — Calculation note — Calculation note** (rtf file).

In the **Calculation note of the results of the MSE calculation** dialog box, in the Output data field, select the options: building characteristics, materials, load coefficients. In the Material consumption field, select the options: by floors and total.

In the **Calculation results** field, specify which elements' calculation results should be printed (for a multi-story building, these may be the most loaded elements of floors 1-3), otherwise the explanatory note will be very voluminous.

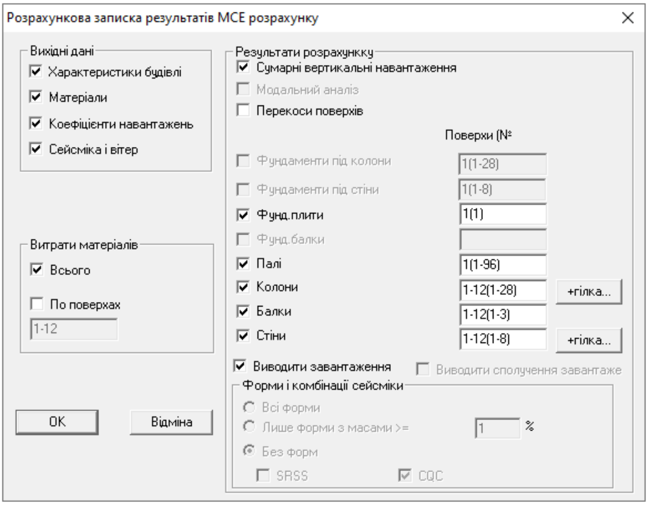


Fig. 37 – Dialog box Calculation note of MFE calculation results

The explanatory note is created. To save it, execute the Save As command. By default, the file will be saved in the Notes directory of the MONOMAKH PC.

Export to design programs of the MONOMAKH PC

Export the results of the MFE calculation to the MONOMAKH PC design programs using the Results menu — Export to the MONOMAKH-SAPR PC design program (button on the toolbar). A directory with the name of your task will be created on the disk in the Port directory of the MONOMAKH PC. Files with data on the structural elements of the building will be placed in this directory (Fig. 38).

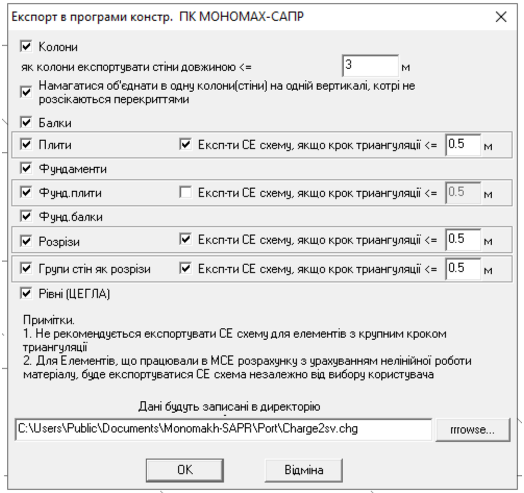


Рис. 38 – *Export to design programs MONOMAKH-SAPR* dialog box

Questions for self-control

1. What is the MONOMAKH PC intended for?
2. What programs are included in the MONOMAKH PC?
3. What initial data is required to create a calculation model in the LAYOUT program?
4. Describe the main stages of building a calculation scheme.
5. The results of which building calculation are final?
6. What commands can be used to display the results of a static calculation on the screen - displacements, isofields of stresses and forces?
7. How to view information about a specific element or node of a calculation scheme?
8. How is an explanatory note formed based on the results of a static calculation?
9. How is the calculation results exported to element design programs?
10. Give an algorithm for creating a building model by importing data from a dxf file from the AutoCAD program.

TOPIC 9. Calculation and Design of a monolithic floor slab in the SLAB program in import mode from the KOMPONOVKA program

Task. Using import, obtain a floor slab model in the SLAB program, using the building model created in the KOMPONOVKA program; perform calculation and design of the slab.

Laboratory Work No. 19

Creating a new problem in import mode

Purpose and plan of the session

Import the file with the slab from the KOMPONOVKA program. Calculate the model in the SLAB program.

Import of the slab calculation scheme

Work with the SLAB program is started by the command **Start – LIRA-SAPR – Monomakh-SAPR – 6. Slab.**

To import files created in the KOMPONOVKA program, select the **File – Import** menu item (button  on the toolbar), open the desired file from the saved folder in the **Port** PC MONOMAKH directory (slabs are numbered with two digits, the first of which means the floor number, the second – the slab number).

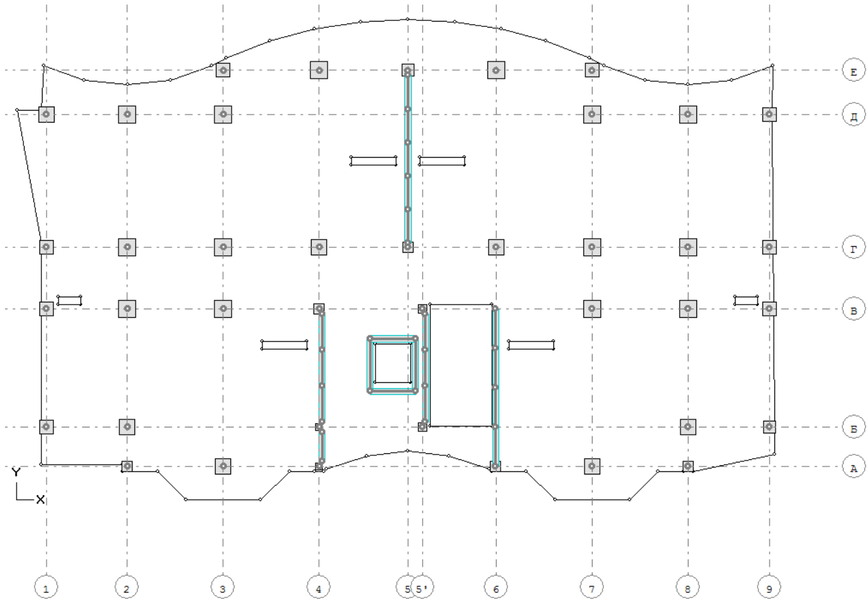


Fig. 40 – Slab No. 2_1 (formwork drawing)

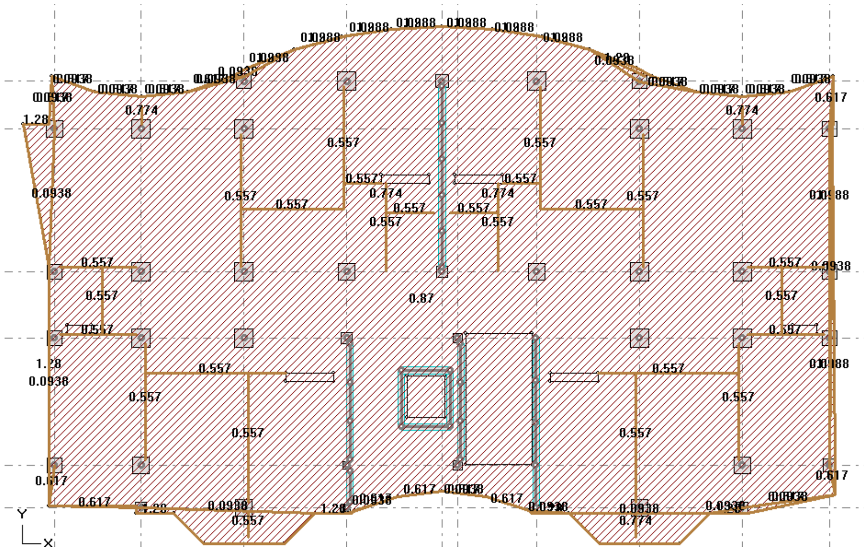


Fig. 41 – Slab No. 2_1 with loads

To check the import of loads, select the menu item **Load – Permanent** (button on the toolbar).

It is important to remember that the element “partition” in the SLAB program is interpreted as a linear load. For floor slabs, vertical loads of permanent, long-term and short-term loads are imported. To take into account the self-weight of the slab when calculating it, the option **Load – Taking into account the self-weight of the slab** must be set (button on the toolbar), and when taking into account the self-weight of the beams, the option **Load – Taking into account the self-weight of the beams** (button on the toolbar).

Information about the model must be saved (**File – Save**).

The slab model is ready for calculation, but it is worth specifying some of the parameters accepted by default. In the **Materials** dialog box, on both tabs, you need to check all the data, specify the type of support of the slab on columns and walls. Hinged support of the slab on columns and walls is accepted by default. You can adjust the support both selectively and for all elements at the same time.

Slab calculation

Slab calculation is performed using the **Calculation — Calculation** menu (button on the toolbar). During the calculation process, the SLAB program automatically generates a finite element calculation scheme taking into account the specified triangulation step. Static calculation is performed by the calculation processor using the finite element method (FEM).

Saving calculation results and exporting to the LIRA PC

When saving the model using the **File — Save** menu, the calculation results are also saved in the *.plt file.

If further work with the scheme in the LIRA PC is planned, export the calculation scheme using the **File — Export to the LIRA PC** menu.

Laboratory lesson 20

Analysis of the results of calculation and design of the slab

Purpose and lesson plan

In the calculation results display mode, view and analyze the results of the calculation and design of the slab. Set the reinforcement color scale. Create a calculation note.

Perform the design of the slab according to the reinforcement mosaics: main reinforcement, control of residual reinforcement. Prepare slab drawings and specifications.

Viewing calculation results

The calculation results display mode is selected using the **Results – Results display modes** menu. To display displacements and forces, it is advisable to select the Isofield+Isolines option, and to display reinforcement – **Mosaic**.

You can view the displacement isofields using the **Results – Displacements** menu (button  on the **Visualization** toolbar). Displacement isofields are constructed as envelopes. When viewing displacements, their calculated values are given.

To display the moment isofields Mx (Mu) on the screen (Fig. 42), execute the command: **Results – Forces – Moments Mx (Mu)** (button  or  on the toolbar). Force isofields are constructed based on loads. By default, when viewing forces, their calculated limit values are given. Operational (normative) force values can be viewed by clicking the button  on the **Visualization** toolbar.

In the SLAB program, the reinforcement of the slab is considered separately - near the upper and lower faces of the slab. Transverse reinforcement and reinforcement calculated for compression are considered separately. Isofields and mosaics of the calculated reinforcement of the slab are constructed separately for the X and Y directions.

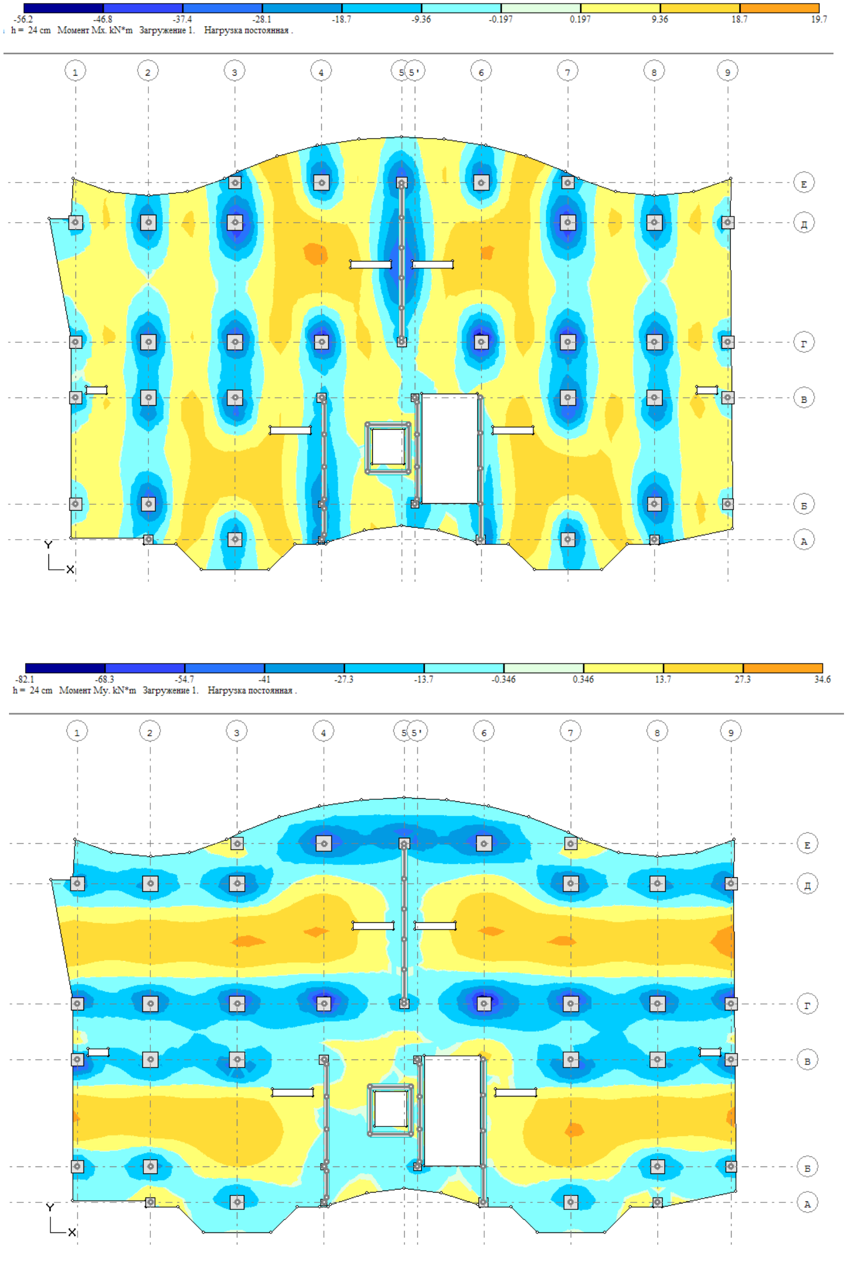
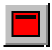


Fig. 42 – Isofields of moments Mx, Mu

To display the reinforcement near the top edge of the slab, execute the command: **Results – Top reinforcement** (button  on the toolbar).

The mosaic of the calculated reinforcement Ax (Fig. 43) is viewed using the menu **Results – Results display modes – Isofields along the X axis** (button  on the toolbar).

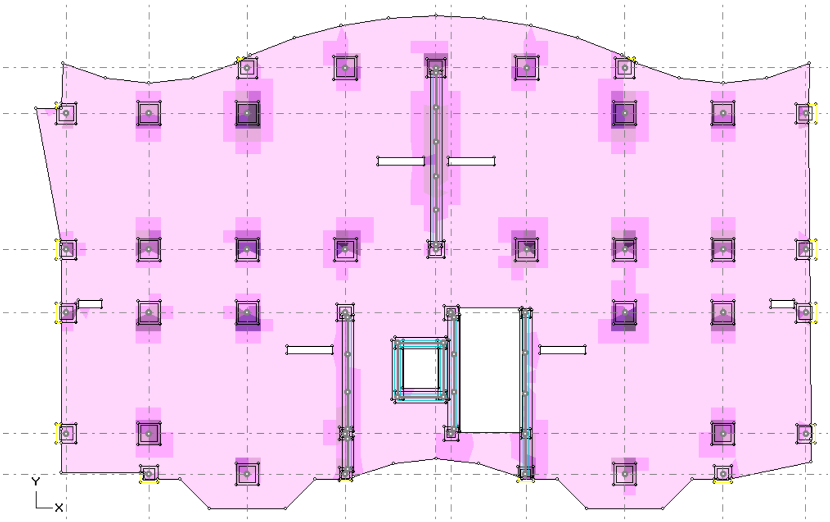


Fig. 43 – Mosaic of reinforcement Ax near the upper face of the slab

The mosaics help to identify in the SE diagram the slabs with the maximum calculated value of the design reinforcement.

The transverse reinforcement is displayed after selecting the type of reinforcement using the **Results - Transverse reinforcement** menu. To display the transverse compression reinforcement, a special compression calculation must be performed.

Setting the color scale of reinforcement

When setting the color scale of the design reinforcement, you can specify ranges not only in numerical form, but also in analytical form. In this case, the value is specified as a formula. *For example, the formula s200d10 means the diameter of the reinforcing bar d = 10 mm with a step of s = 200 mm and corresponds to a reinforcement area of 3.925 cm*^{2}*. The formula*

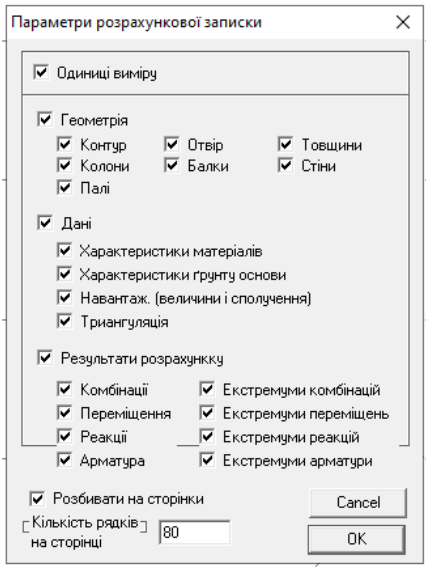
5d10 (or k5d10) means 5 bars with a diameter of d = 10 mm per 1 linear meter and corresponds to a reinforcement area of 3.925 cm2. You can specify an expression of the form s200d8 + s200d22 or a mixed expression of the form s200d8 + 19.01, where 19.01 is the area value, cm^2.

If it is necessary to change the scale parameters, open the **Scale Parameters** dialog box using the **Results – Scale Parameters** menu (button  on the toolbar) and make the necessary changes. As a result, the isofields will be displayed according to the new scale parameters.

This presentation of reinforcement isofields is most often used in real design, because it corresponds to the common principle of slab reinforcement: along the entire slab near the lower and upper edges of the slab, bars of the main reinforcement are inserted, and in places where reinforcement is required, bars of additional reinforcement are inserted.

It is necessary to carefully analyze the location and dimensions of the range on the diagram that does not have a formula description, and if necessary, make corrections to the range table. For example, you can increase the diameter of the additional reinforcement layer or reduce the reinforcement pitch.

Generating and reviewing a settlement note



Create a settlement note using the menu Results – Settlement note – Save html file (button on the toolbar). You can also save the explanatory note as an rtf file (button). In the Settlement note parameters dialog box, check the appropriate boxes (Fig. 44).

Fig. 44 – *Payment Note Settings* dialog box

Slab design

Slab design is performed according to reinforcement mosaics. For the specified layout zones, the pitch and diameter of the longitudinal reinforcement bars near the upper and lower edges of the slab are selected. The residual reinforcement is monitored. The residual reinforcement area is determined for each SE of the slab as the difference between the calculated value of the reinforcement area and the value of the reinforcement area provided by the grids or bars specified in the layout areas. For a slab with a cross-sectional height of 160 mm, transverse reinforcement is not used, therefore, design with transverse reinforcement is not performed.

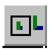
Layout of main reinforcement bars in a given area

Main reinforcement with bars is performed on the entire contour of the slab using the **Design menu – Mesh and bar layout – Rectangular contour** (button  on the toolbar). In the **Reinforcement zone assignment** dialog box, select the diameter of the reinforcement bars that are laid out orthogonally to and along the baseline from the list, and click the **On the entire contour** button.

The orientation of the bars in the layout area is determined by the baseline — the longitudinal bars will be laid out perpendicular to this line, and the transverse bars will be laid out along this line. When laying out the bars, it can be specified arbitrarily relative to the position of the area. The baseline is displayed in blue on the diagram.

Similarly, the reinforcement area for the second layer is assigned.

Residual reinforcement control

Residual reinforcement is displayed when viewing the reinforcement area Ax, Ay in graphical or numerical form for the selected reinforcement type - near the upper or lower face of the slab. To do this, execute the command **Results - Reinforcement display modes - Residual** **reinforcement** (button  on the toolbar).

Layout of additional reinforcement bars in specified areas

Assign a section for the bar layout (additional reinforcement) using the **Design menu - Mesh and bar layout - Rectangular contour** (button  on the toolbar).

In the **Reinforcement Zone Assignment** dialog box, perform the following actions:

• select from the list the diameter of the bars that will be laid out orthogonally to the baseline - 0 mm;

• select from the list the diameter of the bars that will be laid out along the baseline - 0 mm;

• leave the rest of the parameters by default.

If only the bar pitch is specified, the selection of the required diameter will be performed based on the required residual reinforcement area.

Viewing the parameters of the assigned layout areas is performed using the **Design menu — Adjust layout area** (button on the toolbar). Deleting assigned layout areas is performed using the **Design menu — Delete layout area** (button on the toolbar).

Displaying assigned layout areas is performed using the **Design menu — Show layout areas of main meshes and bars** (button on the toolbar) and **Show layout areas of additional meshes and bars** (button on the toolbar).

Saving design results

When saving a model using the **File - Save** menu (button on the toolbar), the design data is also saved in the *.plt file.

Slab drawing

Slab drawing when reinforced with individual bars is performed using the **Results — Drawing — Bar drawing** menu (button on the toolbar).

The SLAB DRAWING program will be launched (in the bar reinforcement version).

A drawing consists of separate fragments: grid and bar layout diagrams, specifications, main inscription, etc. A certain area is allocated for each fragment on the drawing sheet. The scale of the fragment image, with the exception of fragments with tables or texts, is determined by the dimensions of this area. You can change the sizes of fragment areas, move, delete and add new fragments from the existing list of fragments. You can also change the sheet format, color and dimensions of individual drawing elements (for example, the height of symbols), change the position of footnotes, etc.

Main reinforcement bar layout scheme

By default, a sheet with the main reinforcement bar layout scheme near the bottom edge of the slab opens (the and buttons on the toolbar are pressed).

Drawing correction

The drawing correction mode is entered using the **Tools menu — Drawing correction** (button on the toolbar). The positions of the bars are corrected using the **Tools menu — Reposition reinforcement image** (button on the toolbar); the positions of the callout lines are corrected using the **Tools menu — Reposition callout segment** (button on the toolbar).

Specification parameters

Since the main reinforcement bars have a length exceeding the maximum bar length of 12 m, the specification in the **Name** column indicates the linear length of the bars for each item. The linear length of the bars is given taking into account the bar allowance (the total length of the bars is multiplied by the allowance coefficient). The **Weight per** unit column indicates the weight of 1 m of a linear bar.

In order to hide information in the specification and in the steel consumption information about bars that are not currently displayed on the drawing (additional reinforcement), you need to cancel the **Full specification table** command using the **View — Full specification table** menu (button on the toolbar).

You can familiarize yourself with the specification parameters using the **Service — Specification parameters** menu (button on the toolbar).

Filling in the main text

You can fill in the main text using the **Sheet - Main Text** menu (button on the toolbar).

Printing a drawing

To print, you need to select a different style for setting colors and sizes of drawing elements than the on-screen style using the **Tools menu — Select colors and sizes — For printing** (button on the toolbar).

You can change the styles for setting colors and sizes of drawing elements using the **Tools menu — Colors and sizes** (button on the toolbar).

You can print drawings using the **File menu — Print**.

Saving the drawing

To save the corrected drawing, select the **File — Save** menu item (button on the toolbar).

Importing a dxf drawing file

If you need to further refine the drawing, you need to create a dxf drawing import file.

To do this, select a different style for setting the colors and sizes of the drawing elements using the **Tools — Select color and size** settings — For dxf file menu (button on the toolbar).

Check whether the zoom mode is enabled in the **Fragment — Scale fragments for dxf file** menu (button on the toolbar).

Check the scales of the fragments using the **Fragment — Scale fragments for dxf file** menu.

Save the dxf drawing file using the **File — Save drawing as dxf file** menu (button on the toolbar).

Additional reinforcement bar layout scheme

You can restore the original style of setting colors and sizes of drawing elements using the **Tools menu - Select color and size settings - For screen** (button on the toolbar).

*Attention: specifications are given for all reinforcement. To display only additional reinforcement, use the **View — Full specification table** command (button on the toolbar).*

On a new sheet, perform the commands necessary to obtain a high-quality drawing – adjust the position of the footnotes, set the specification parameters, fill in the main inscription.

Questions for self-control

1. How to start working in the SLAB program?
2. How to import the necessary file with the results of the slab calculation from the KOMPONOVKA program?
3. How is the slab calculation performed?
4. How to view the results of the static calculation of the slab?
5. How to view the results of the selected slab reinforcement?
6. How to set the color scale of the calculated reinforcement?
7. How to create an explanatory note?
8. How to design the slab?
9. How to check the residual reinforcement?
10. How to lay out the main and additional reinforcement?
11. How can I correct the drawing?

TOPIC 10. Design of columns and pylons in the COLUMN program

Laboratory lesson 21

Model creation and column calculation

Purpose and plan of the session

Create a column model in the COLUMN program, perform its calculation. Change design parameters. Export data on releases. Generate a calculation note. Prepare a column drawing, add details.

Creating a new task

To start working with the COLUMN program, execute the Windows commands: **Start – Programs – LIRA-SAPR – Monomakh-SAPR – 3. Column.**

When launched, the COLUMN program automatically creates a new document, so no additional actions are required to create a new task.

To save information about the model, execute the **File – Save** menu item (button  on the toolbar).

Data correction

The new document already contains some data accepted by default.

Using the **Materials – Material characteristics** menu, open the **Column – Material characteristics** dialog box, in which you specify the concrete class and operating conditions, leaving the remaining parameters unchanged. Click the OK button.

Adjust the cross-sectional dimensions and column length using the **Data – Geometry** menu (toolbar button ).

To make the program take into account the presence of outlets in the upper column, select the **Data – Column top – Slab and upper section menu item** (toolbar button ).

Set the column load using the **Data – Loads** menu (toolbar button ).

*Important: in the COLUMN program, it is customary to set the standard values of loads with a load safety factor, because load safety factors are set in a separate table using the **Column – Loads** menu (**Coefficients** tab).*

Column calculation

Column calculation is performed using the **Calculation – Calculation** menu (button  on the toolbar).

In the calculation process, first a calculation scheme is formed and a static calculation is performed to determine displacements and forces. After that, calculation force combinations are formed, calculation reinforcement is determined and the column is designed.

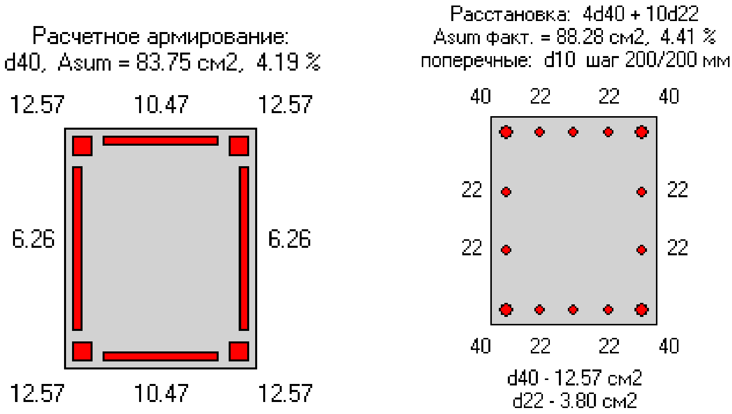


Fig. 45 – Calculation and design results

Changing design parameters

If necessary, change the design diameter and assortment using the **Data – Materials** menu (button  on the toolbar).

In the **Column – Material Characteristics** dialog box, on the **Diameters** tab, select the design diameter, set the thickness of the protective layer;

on the **Requirements** tab, specify the design requirements. Perform the calculation again (button  on the toolbar) and view the reinforcement results.

Exporting Issue Data

Create an export file for the FOUNDATION program using the **Results – Export Issue Data** menu.

In the **Save As** dialog box, save the file Column.

Creating a settlement note

Create a settlement note using the **Results-Settlement note-Save as rtf file** menu (button  on the toolbar).

To view the settlement note, execute the Windows command: **Start – Programs – Microsoft Word.**

- Open the note using the File –Open menu.
- In the File type list, select Text in RTF format (*.rtf).
- Open the desired file.

Column drawing

A column drawing is performed using the **Results – Drawing** menu (button  on the toolbar). The COLUMN DRAWING program starts.

Adding a list of details

Add a list of details (by default it is not drawn) using the **Fragment – List of details** menu.

On the diagram, click the mouse in the upper left point of the fragment location area and, without releasing the button, set the dimensions of the fragment area by moving the cursor to the lower right corner.

Laboratory Work No. 22

Import from the KOMPONOVKA program and pylon calculation

Purpose and plan of the session

Using import, obtain a pylon model in the COLUMN program, using the building model created in the KOMPONOVKA program; perform the calculation and construction of the pylon.

Creating a new task

To start working with the COLUMN program, execute the Windows commands: **Start – Programs – LIRA-SAPR – Monomakh-SAPR – 3. Column.**

To import a file, execute the **File - Open** menu item (button  on the toolbar). Select the required file from the **Port** folder of the MONOMAKH PC.

The pylon diagram is shown in Fig. 46. The loads on the pylon (normative values) are also imported from the KOMPONOVKA program.

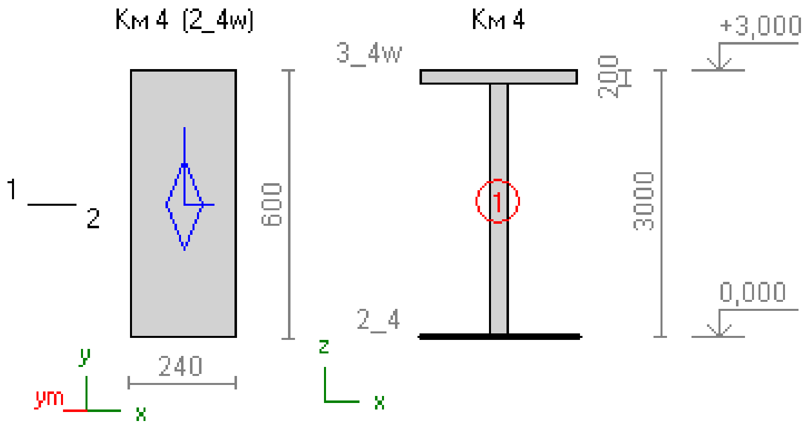


Fig. 46 – Pylon diagram

To save information about the model, select the **File – Save** menu item (button on the toolbar).

Data correction

Data adjustments (structural requirements, concrete cover, design factors, etc.) in the pylon model are performed in the same way as for columns.

The program allows you to select reinforcement and design pylons both according to the classical scheme and according to the arrangement of reinforcement along the longer sides of the pylons.

The **Reinforce as pylon** characteristic is set using the **Data – Materials – Requirements** menu.

In the **Column – Material Characteristics** dialog box, on the **Diameters** tab, select the design diameters, specify the protective layer; on the **Requirements** tab, select the type of transverse reinforcement.

To take into account the presence of outlets in the upper column or pylon, select the menu item **Data – Column top – Slab and upper section** (button on the toolbar).

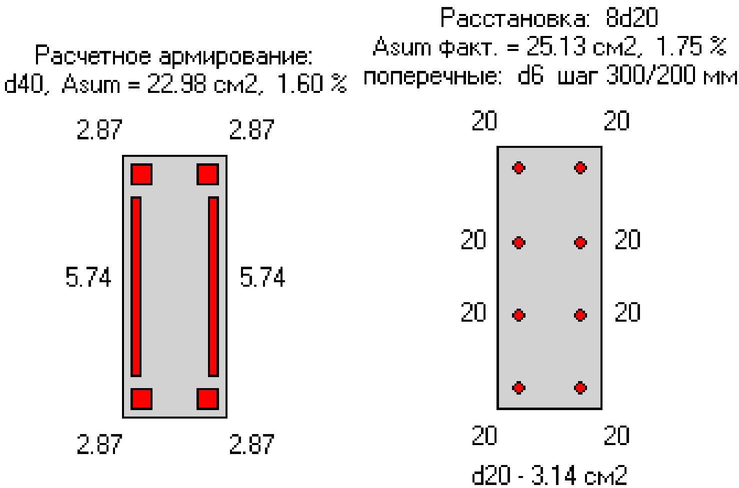
If there is a need to design a pylon or column with two or three floors, select the menu **item Data – Floors – Add floor from import file.**

Pylon calculation

The pylon is calculated using the **Calculation – Calculation** menu (button  on the toolbar). During the calculation, the program determines the calculated reinforcement and performs the pylon design (Figure 7.3).

If you check the **Reinforce as pylon** and **Select angle bars** options in the **Data – Materials – Requirements** tab menu, you can get another design option – with developed angle bars (Figure 47).

Analysis of the design results, formation of an explanatory note, and drawing of a pylon created in import mode are performed in the same way as for a column.



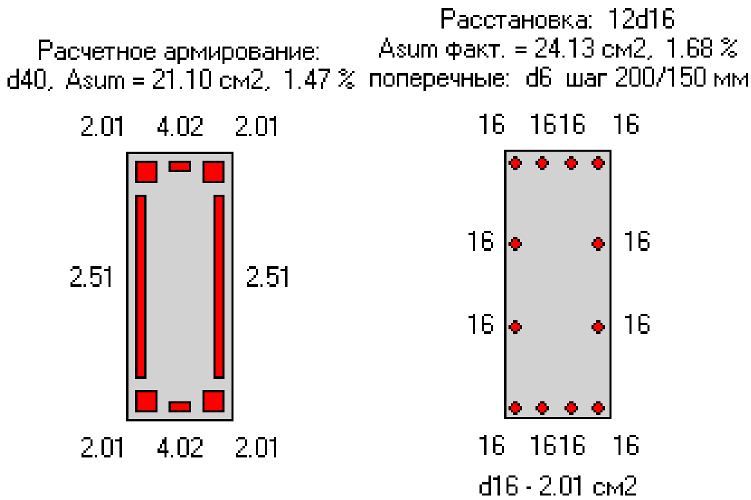


Рис. 47 – Результати розрахунку та конструювання пілона за класичною схемою

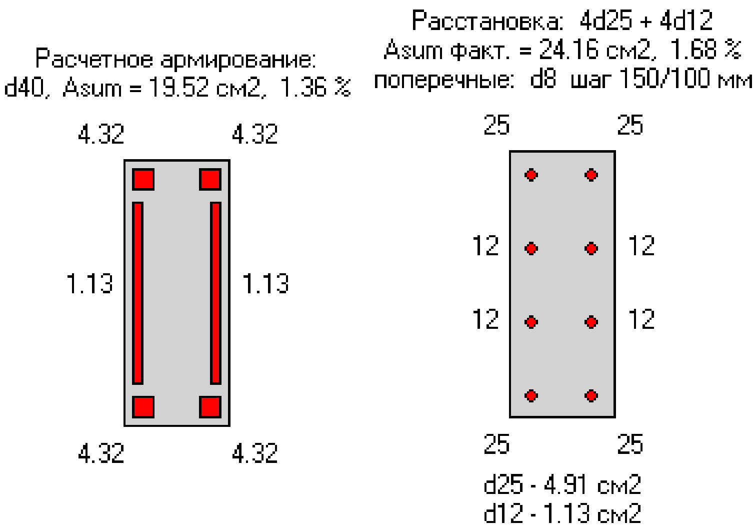


Fig. 48 – Results of calculation and design of a pylon with developed corner rods

Questions for self-control

1. What is the COLUMN program intended for?
2. List the main stages of work in the calculation scheme creation mode.
3. How to specify material characteristics?
4. How to specify geometric parameters of an element?
5. How to specify the load on a column?
6. How to change design parameters?
7. How to export data on releases?
8. How to prepare an explanatory note?
9. How to create a column drawing?
10. How to add a bill of materials to a column drawing?
11. List the main stages of creating a pylon calculation scheme in the import mode from the KOMPONOVKA program.
12. How to perform data correction?
13. How to specify the design diameters of rods, the thickness of the protective concrete layer?
14. How to specify the type of transverse reinforcement?
15. How to change design parameters?
16. What command should be executed to take into account the presence of outlets in the upper column or pylon?
17. What command should be executed to design a pylon with two or three floors?

REFERENCES

1. М.С. Барабаш, П.М. Кір'язев, О.І. Лапенко, М.А. Ромашкіна. Основи комп'ютерного моделювання. К.: НАУ, 2019. 500 с.
2. ПК ЛІРА-САПР. Книга І. Основи. Е.Б. Стрілець-Стрілецький, А.В. Журавльов, Р.Ю. Водоп'янов Р.Ю. Під редакцією академіка Городецького О.С. Електронне видання, 2019. 154 с.
3. ПК ЛІРА-САПР. Приклади розрахунку і проектування. Водоп'янов Р.Ю., Тіток В.П., Артамонова А.Е, Ромашкіна М.А. Під редакцією академіка Городецького А.С. Електронне видання, LIRALAND Grup. 635 с.
4. Мономах-САПР. Навчальний посібник. Приклади розрахунку та проектування. Електронне видання. Городецький Д.А., Юсипенко С.В., Батрак Л.Г., Лазарев А.А., Расказов А.А. ООО «ЛІРА-САПР», 368 с. https: //www.liralend.ua/books/11/1410.
5. Ротко С.В., Ужегова О.А., Задорожнікова І.В., Кислюк Д.Я., Ужегов С.О. Залізобетонні конструкції: Навчальний посібник / Луцьк: ЛНТУ, 2021. 404 с.
6. Кислюк Д.Я., Ротко С.В., Ужегова О.А., Задорожнікова І.В., Сунак О.П. Інженерні споруди: Навчальний посібник. Луцьк: РВВ Луцького НТУ, 2020, 368 с.
7. ДБН В.1.2-14:2018. Загальні принципи забезпечення надійності та конструктивної безпеки будівель і споруд. – [Чинні від 2019-01-01]. – К.: Мінрегіон України, 2018.
8. Гераськін О. О., Ротко С. В., Ужегова О. А. Розрахунок монолітної плити з урахуванням реологічних властивостей залізобетону. Сучасні технології та методи розрахунків у будівництві: зб. наук. праць, 2020. Вип. 14. С. 63-72.
9. Ротко С.В., Ужегова О.А., Талах Л.О., Булда К.О., Артемук Т.С., магістри. Урахування послідовності зведення на НДС конструкцій при автоматизованому проєктуванні будівель і споруд // Сучасні технології та методи розрахунків у будівництві: будівництві: зб. наук. праць, 2023. Вип. 20. С.108-116.
10. Ротко С.В., Зінькевич К.Я., ст. До розрахунку будівель на прогресуюче руйнування за допомогою програмних комплексів // Сучасні проблеми містобудування. Перспективи та пріорітети розвитку: збірник тез доповідей Всеукраїнської науково-практичної інтернет-конференції молодих учених та студентів (19 листопада 2021 р., м. Луцьк) [Електронний ресурс]. Режим доступу: https://konf-mbg.wixsite.com/lntu-bci-mbg-2021/tezi-dopovidej
11. Ротко С.В., к.т.н., доц., Талах Л.О., к.т.н., доц., Кузьменко К.А. розрахунок будівель і споруд на вибухові впливи у середовищі

ЛІРА-САПР // Інновації у будівництві: збірник тез доповідей IX міжнародна науково-практична інтернет-конференція здобувачів вищої освіти та молодих учених (14 травня 2024 р., м. Луцьк) [Електронний ресурс]. Режим доступу: https://sites.google.com/view/iic-2024

Information resources

1. Офіційний сайт ЛІРА-САПР LIRALEND GROUP [Електронний ресурс]: [Веб-сайт]. Електронні дані. К: ВАТ «ЛІРА САПР», 2002-2024. Режим доступу: https://www.liraland.ua/
2. Форум користувачів ЛІРА-САПР [Електронний ресурс]: [Веб-сайт]. Електронні дані. Режим доступу: https://www.liraland.ua/forum/
3. ЛІРА-САПР. Офіційний канал [Електронний ресурс]: [Веб-портал]. Електронні дані. YouTube LLC, 2024. Режим доступу: https://www.youtube.com/user/LiraLand
4. МОНОМАХ-САПР. [Електронний ресурс]: [Веб-сайт]. Електронні дані. К: ВАТ «ЛІРА САПР», 2011-2024. Режим доступу: https://www.liraland.ua/mono/

EDUCATIONAL AND METHODOLOGICAL EDITION

C 13 **CAD in Construction** [text]: methodological instructions for laboratory classes for applicants of the first (bachelor's) level of higher education of the educational and professional program "Construction and Civil Engineering" in the field of knowledge 19 Architecture and Construction, specialty 192 Construction and Civil Engineering full-time and part-time forms of study / compiled by S. Rotko. Lutsk: LNTU, 2026. 188 p.

Computer typing and layout S. Rotko
Editor S. Rotko

Signed for publication in 2026.
Format 60× 84/16. Paper off. Headset Times.
Volume 11.75 printing pages. Circulation 50 copies.

Lutsk National Technical University
43018, Lutsk, street Lvivska, 75