Example 1. Model generation and analysis of multi-storey building in BUILDING module

In this lesson you will learn how to:
- – generate model of multi-storey building in the BUILDING module;
- – analyse the model;
- – visualize analysis results in different ways;
- – export data to other programs;
- – with the model of multi-storey building, generate its second variant - define pile foundation.

Description:
Plans of the standard storey and the basement are presented in Figures 1.a, 1.b.
Variant of pile foundation is presented in Figure 1.c (spacing of piles from 1m to 1.5m).

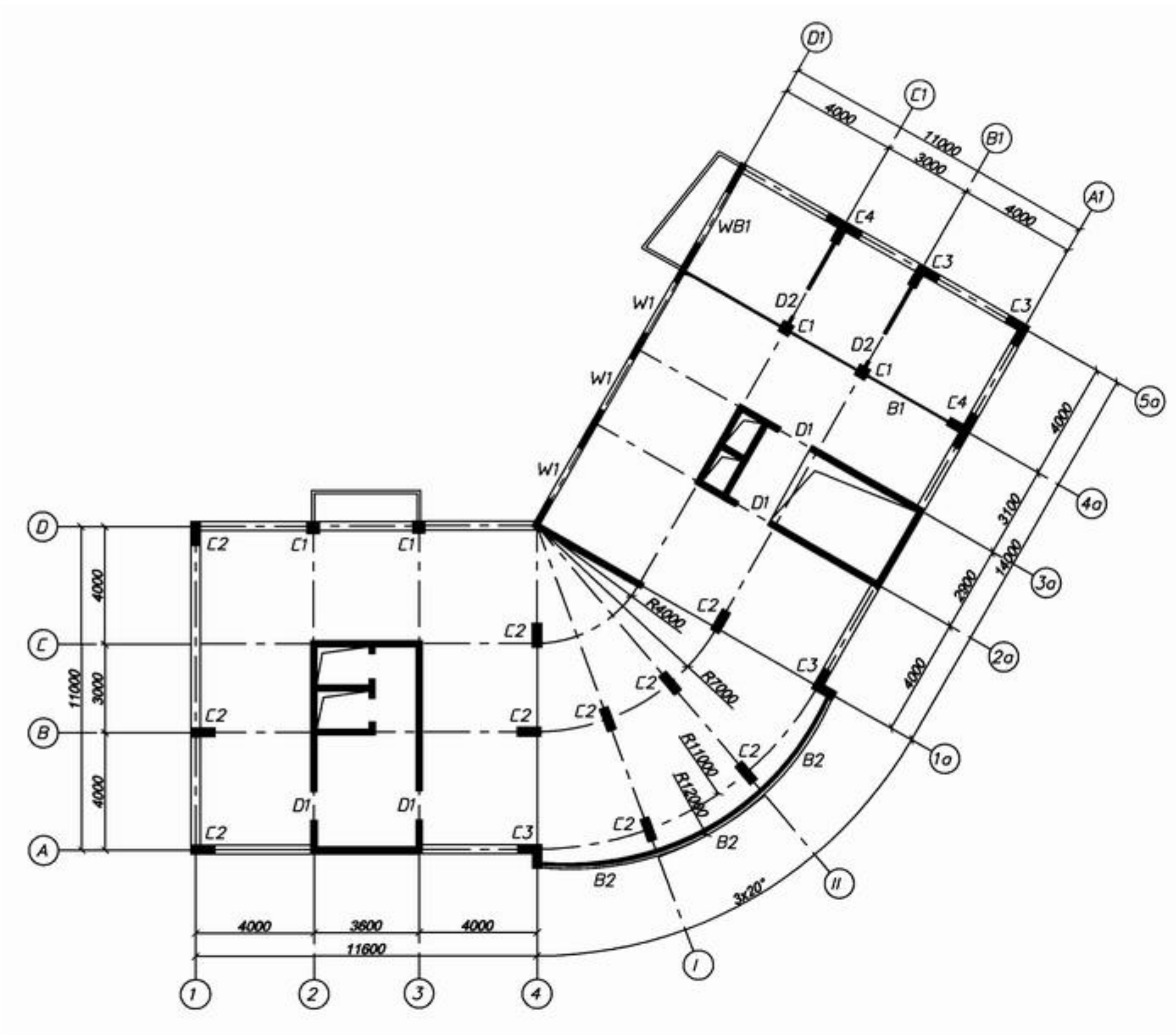


Figure 1.a Plan of the standard storey

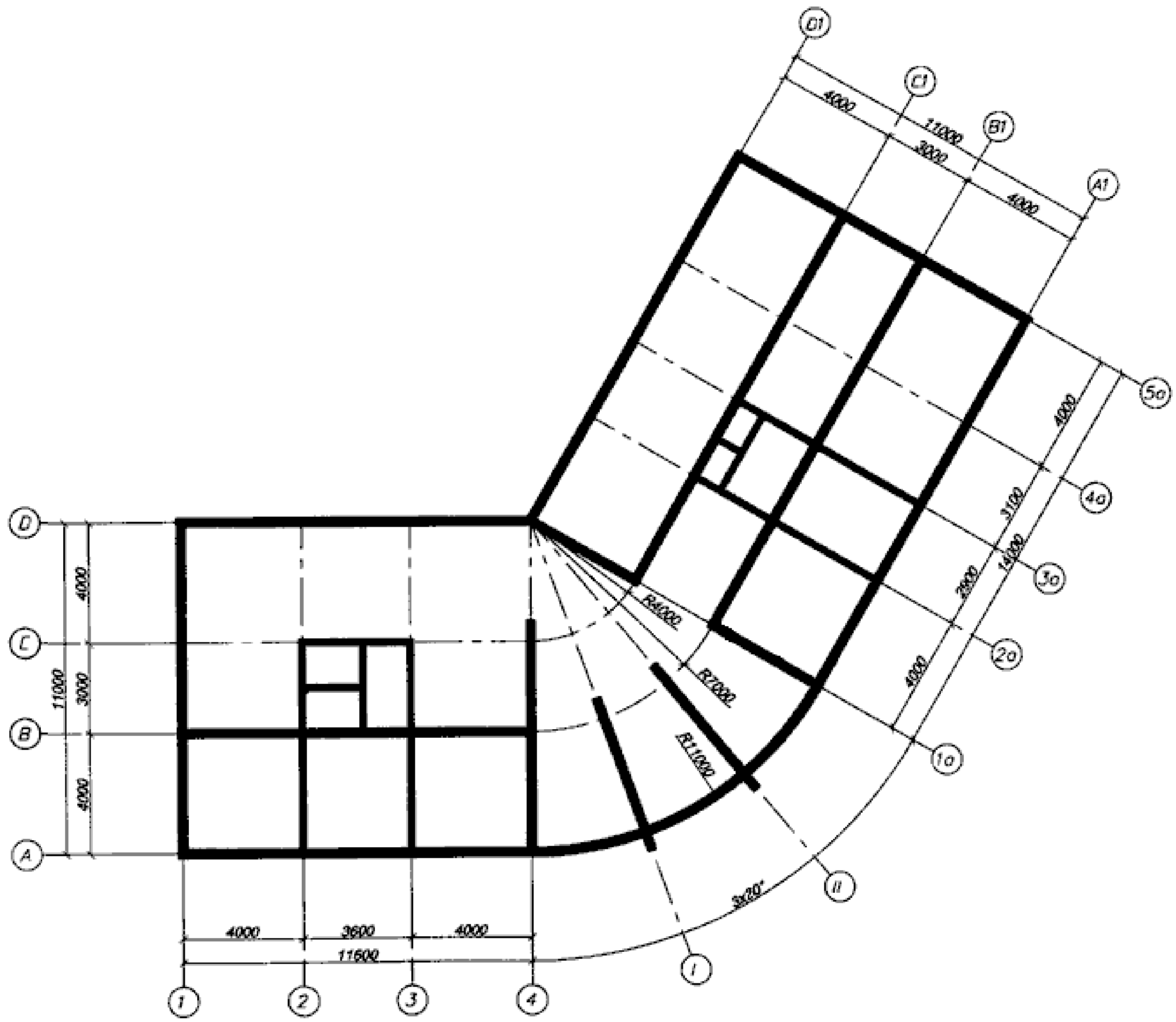


Figure 1.b Plan of the basement

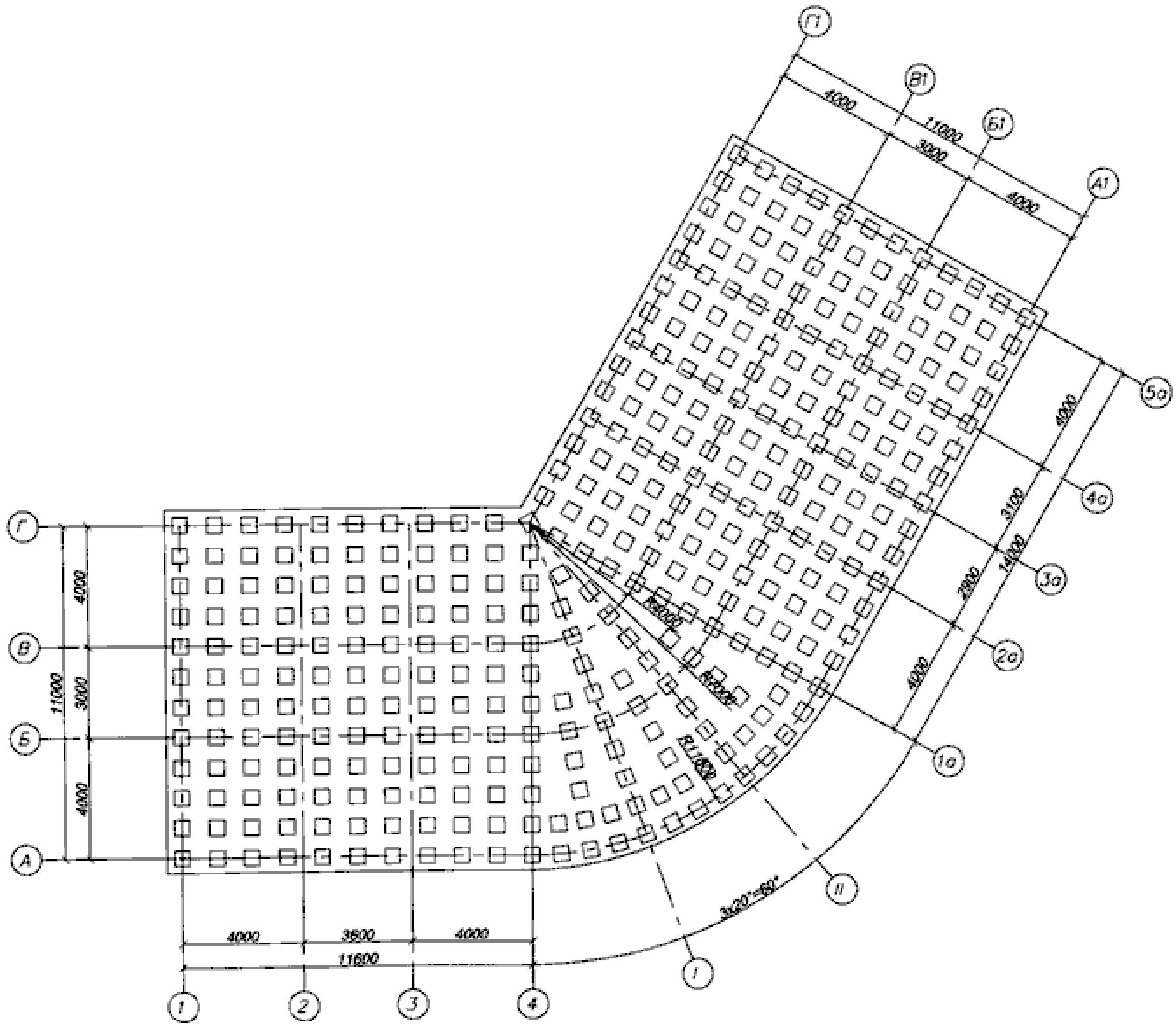


Figure 1.c Pile foundation (the second variant of the model)

Sectional elevation is presented in Figure 1.d. Height of the basement is 2.4m. Height of the standard storey is 3m. Number of storeys - 12. Height of the structure above lift shaft is 2.8m. Floor level of the 1st storey is 0.000. Soil level -0,200m.

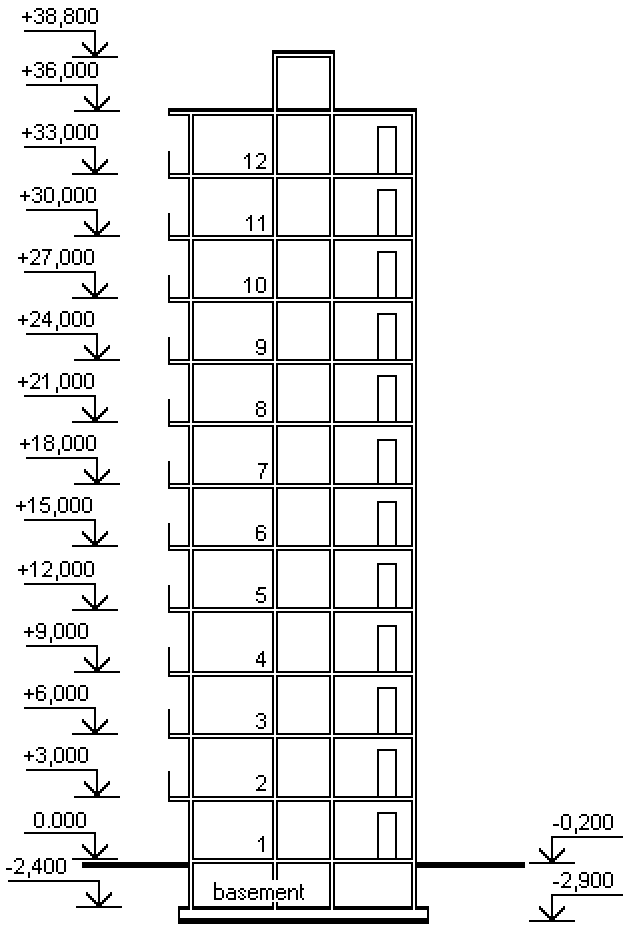


Figure 1.d Sectional elevation

Parameters of foundation soil: sandy clay, unit weight of soil 1.82 tf/m3, angle of internal friction 22 degrees, cohesion 0.8 tf/m3, modulus of elasticity 2000tf/m2, Poisson's ratio 0,3.

Building code for analysis of elements is SNIP 2.03.01-84. Material for elements: columns, beams, slabs and mat foundation – reinforced concrete B30, walls – reinforced concrete B20. Material for non-bearing walls and partitions – regular clay brick.

Sections of columns and beams are presented in Figure 1.e. Thickness of floor slabs 0.2m. Thickness of mat foundation 0.5m. Thickness of walls 0.2m. Thickness of walls in the basement is 0.24m. Thickness of partitions – 0.12m. Non-bearing walls with windows and some partitions are simulated with loads on floor slabs.

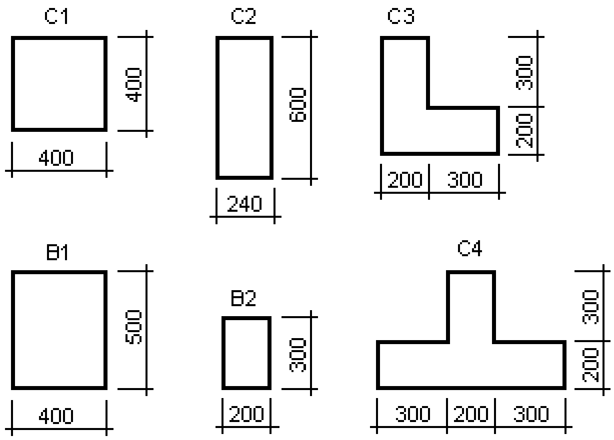


Figure 1.e Sections of columns and beams

Dimensions of window and door openings are presented in Figure 1.f.

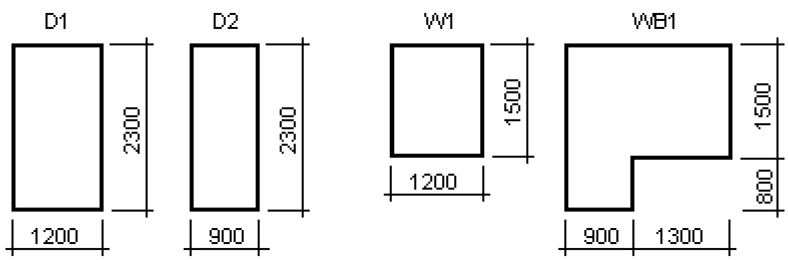


Figure 1.f Dimensions of window and door openings

Section dimensions of piles for the second variant of model are presented in Figure 1.g. Length of piles is 5.5m.

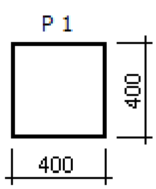


Figure 1.g Sections of piles

Loads on the floor slab (normative values):

- dead uniformly distributed load (with account of weight of partitions) g_1 = 0.3 tf/m^2;
- dead uniformly distributed load (at stair landings and lift halls) g_2 = 0.46 tf/m^2;
- live uniformly distributed load g_3 = 0.4 tf/m^2;
- dead uniformly distributed load from non-bearing walls with windows g_4 = 0.22 tf/m;
- dead uniformly distributed load along the contour of balconies g_5 = 0.14 тс/м;
- dead uniformly distributed load along the contour of openings g_6 = 0.2 тс/м.

Loads on the floor slab of the structure above the lift shaft (normative values):

- dead uniformly distributed load g_1 = 0.2 tf/m^2;
- live uniformly distributed load g_2 = 0.4 tf/m^2.

Loads on mat foundation (normative values):

- dead uniformly distributed load g_1 = 0.1 tf/m^2;
- live uniformly distributed load g_2 = 0.1 tf/m^2.

Wind loads (SNIP 02.01.07-85*), wind region II, type of area B:

- load direction - 90 degrees to the X-axis of the structure;
- load direction - 135 degrees to the X-axis of the structure.

Earthquake loads (SNIP II-7-81*), seismicity of the area - 7 units of magnitude, soil category III:

- load direction - 0 degrees to the X-axis of the structure;
- load direction - 90 degrees to the X-axis of the structure.

When you specify numerical values in the dialog boxes, it is necessary to place point (.) as decimal symbol.

- On the taskbar, click the **Start** button, then point to **Programs** / **MONOMAKH-SAPR 2016** folder and click **1. BUILDING**.

Step 1. Creating new problem and defining general parameters of structure

To create new problem:

- When you start the BUILDING module, the program automatically creates new document.
- Select building codes necessary for analysis. In the **Building codes for analysis of elements** dialog box (see Fig.1.1.1), remain all parameters by default and click **OK**.

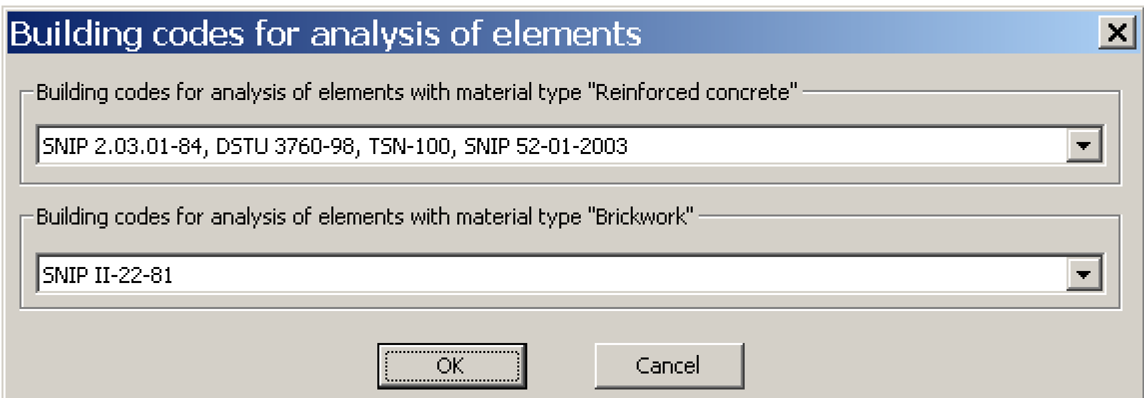


Figure 1.1.1 **Building codes for analysis of elements** dialog box

Selected building code regulates the way for defining material properties (concrete, reinforcement, brickwork), analysis methods, requirements to element design and standards that stipulate design combinations of load cases.

To create one more problem, on the FILE menu, click **New** *(button on the toolbar). To open an existing problem, on the* **FILE** *menu, click* **Open** *(button on the toolbar).*

To define general parameters of structure:

- On the MODEL menu, click **General parameters of structure**.
- In the **General parameters of structure** dialog box (see Fig.1.1.2), specify the following data:
  - soil level 0.2;
  - upper level of column base -2.4;
  - lower level of foundation -2.9;
  - unit weight of soil 1.82 tf/m^3;
  - angle of internal friction 22 degrees;
  - cohesion 0.8 tf/m^3;
  - modulus of elasticity 2000 tf/m^2;
  - Poisson's ratio 0.3;
  - other parameters remain by default.

- Click **OK**.

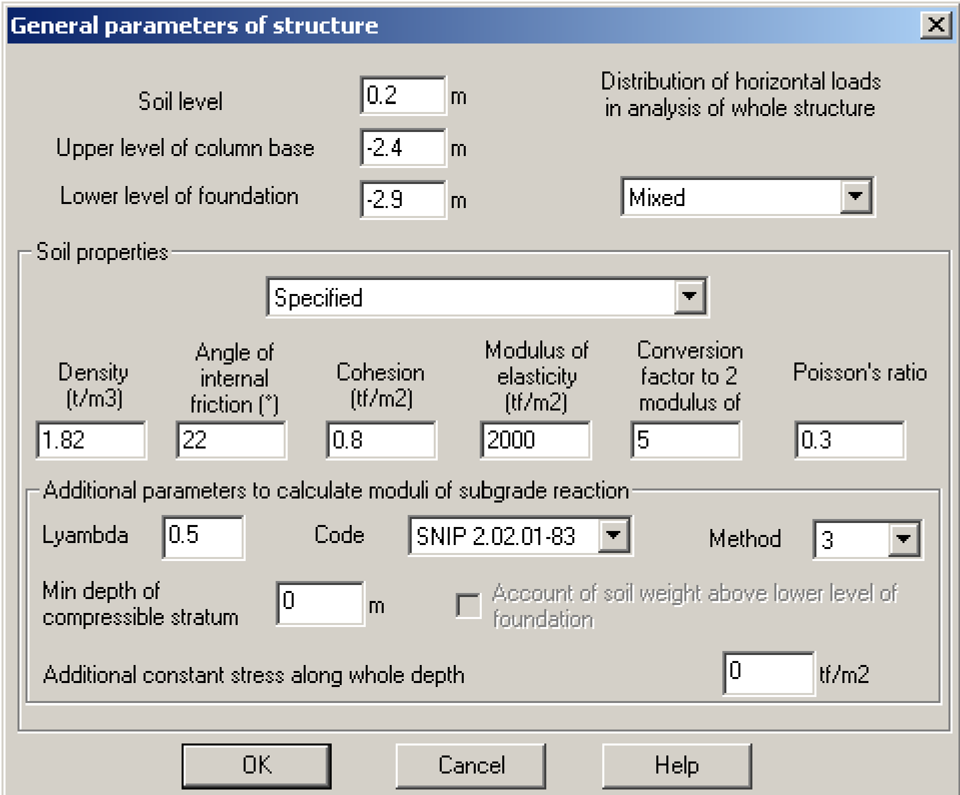


Figure 1.1.2 **General parameters of structure** dialog box

*To define name for the project, on the FILE menu, click **Project name**.*

Step 2. Defining material properties

<u>To define material properties for structures of monolithic reinforced concrete (RC):</u>

Create three different materials with different properties of concrete and reinforcement for columns, walls, beams, slabs and mat foundations.

- On the MODEL menu, click **Materials**.
- In the **Materials** dialog box (see Fig.1.2.1), select material **Reinforced concrete** suggested by default and click **Edit**.

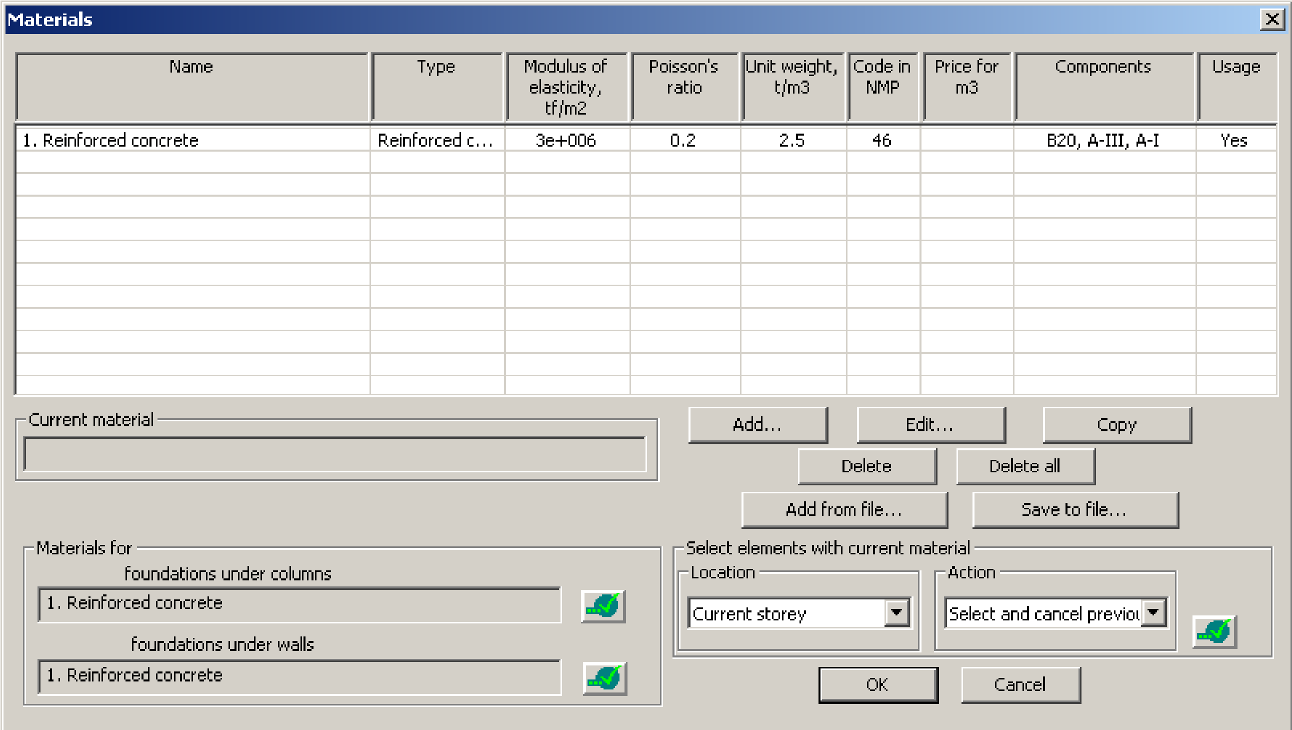


Figure 1.2.1 **Materials** dialog box

- In the **Material** dialog box (see Fig.1.2.2), define the following parameters:
  - name – reinforced concrete **RC B30 AIII AI**;
  - select class of concrete B30;
  - service conditions – standard;
  - click **Analysis** under **Serviceability limit state**;
  - other parameters remain by default.
- Click **OK**.

- In the **Materials** dialog box (see Fig.1.2.1), click **Add**.
- In another **Material** dialog box, define the following parameters:
  - name – reinforced concrete **RC B20 AI AI**;
  - select class of concrete B20;
  - service conditions – standard;
  - longitudinal reinforcement AI;
  - click **Analysis** under **Serviceability limit state**;
  - other parameters remain by default.
- Click **OK**.

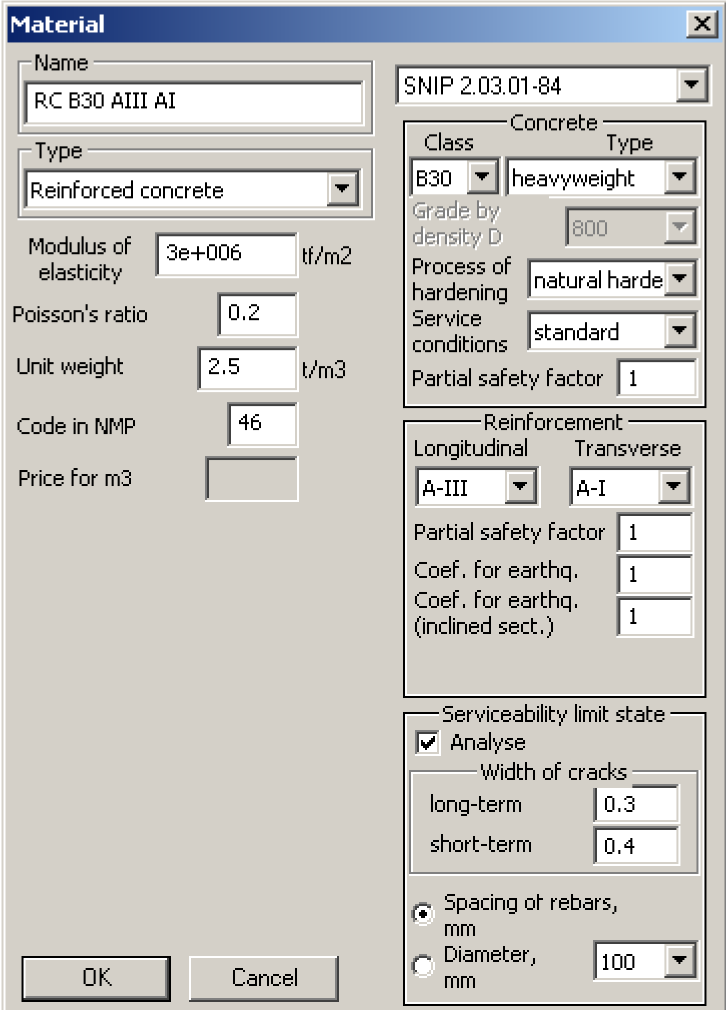


Figure 1.2.2 **Material** dialog box

➪ In the **Materials** dialog box (see Fig.1.2.1), click **Add**.

➪ In another **Material** dialog box, define the following parameters:

- name – reinforced concrete **RC B30 AIII AIII**;
- select class of concrete B30;
- service conditions – standard;
- longitudinal reinforcement AIII;
- click **Analysis** under **Serviceability limit state**;
- other parameters remain by default.

➪ Click **OK**.

Note that unit weight of reinforced concrete is defined as equal to 2.5t/m^3*, that is, with load factor Yf equal to 1. Load from dead weight of elements will be computed by the specified unit weight of material.*

To define material properties for structures of other materials:
Create material for non-bearing walls and partitions.

➪ In the **Materials** dialog box (see Fig.1.2.1), click **Add**.

➪ In another **Material** dialog box (see Fig.1.2.3), define the following parameters:

- name – **Regular brick 125-100**;
- type – brickwork;
- brick mark – 125;
- click **Calculate modulus of elasticity** to automatically determine modulus of elasticity for brickwork;
- other parameters remain by default.

➪ Click **OK**.

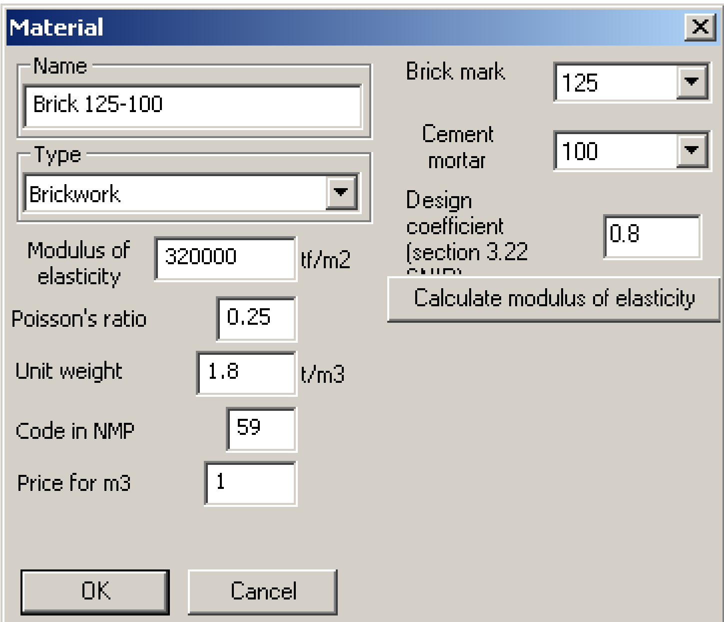


Figure 1.2.3 **Material** dialog box

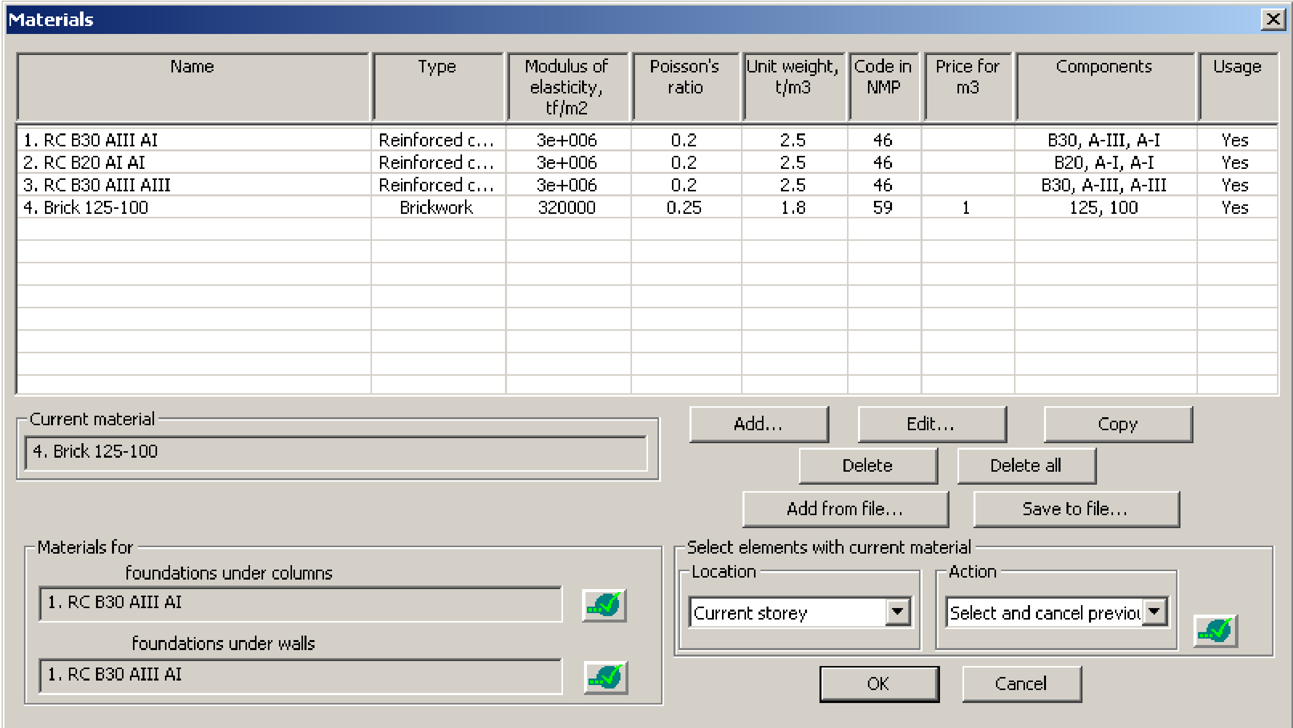


Figure 1.2.4 **Materials** dialog box

➪ In the **Materials** dialog box (see Fig.1.2.4), click **OK**.

To edit percentage of reinforcement:

There is a principle for fixation of section dimensions. Fixed parameters are not changed during analysis. When all section parameters are fixed, the percentage of reinforcement is computed and then if it is greater than maximum one, then you will get an error message. If at least one dimension of the section is not fixed, then section dimensions are computed according to optimum % of reinforcement.

➪ On the MODEL menu, click **Percentage of reinforcement**.

➪ In the **Percentage of reinforcement** dialog box (see Fig.1.2.5), click **Walls** and define the following parameters:

- maximum percentage of reinforcement 7%;
- other parameters remain by default.

➪ Click **OK**.

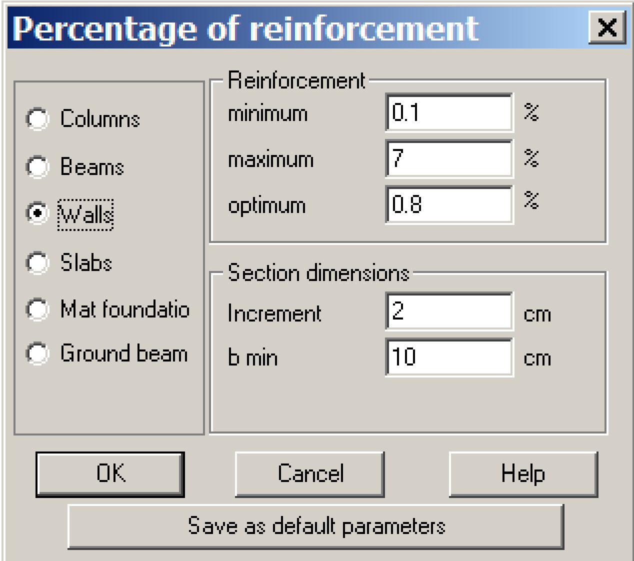
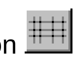


Figure 1.2.5 **Requirements for RC elements** dialog box

Step 3. Defining grid and construction lines of the building

<u>To define grid:</u>
Define the fragment of Cartesian grid between construction lines **1** and **4**, **A** and **D** according to the plan of the building.

➪ On the MODEL menu, point to **Grid** and click **Add fragment of Cartesian grid** (button  on the toolbar).

➪ In the **Cartesian grid** dialog box (see Fig.1.3.1), define the following parameters:

- Divide along X

Step (m)	Number
4	1
3.6	1
4	1

  Divide along Y

Step (m)	Number
4	1
3	1
4	1

- other parameters remain by default.

➪ Click **OK**.

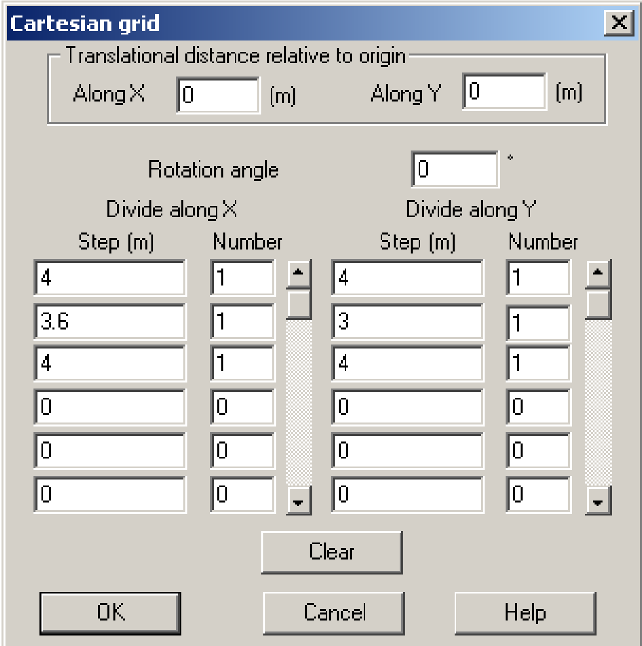


Figure 1.3.1 **Cartesian grid** dialog box

- To move the origin of coordinate system, on the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar).
- When this mode is active, click the upper-right node of the grid (see Fig.1.3.2).

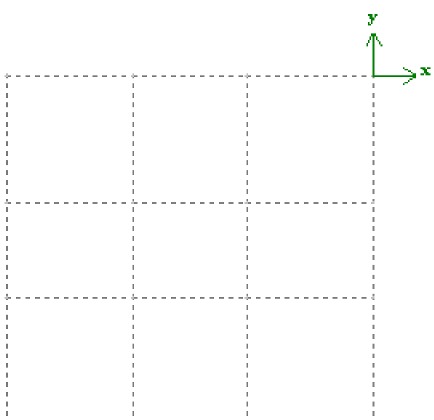


Figure 1.3.2 Move origin

<u>Define the fragment of Polar grid.</u>

- On the MODEL menu, point to **Grid** and click **Add fragment of Polar grid** (button on the toolbar).
- In the **Polar grid** dialog box (see Fig.1.3.3), define the following parameters:
  - initial angle 270 degrees;
  - Divide along circle / Divide along radius

Divide along circle Step (º)	Number	Divide along radius Step (º)	Number
20	3	4	1
		3	1
		4	1
		1	1

- other parameters remain by default.

➪ Click **OK**.

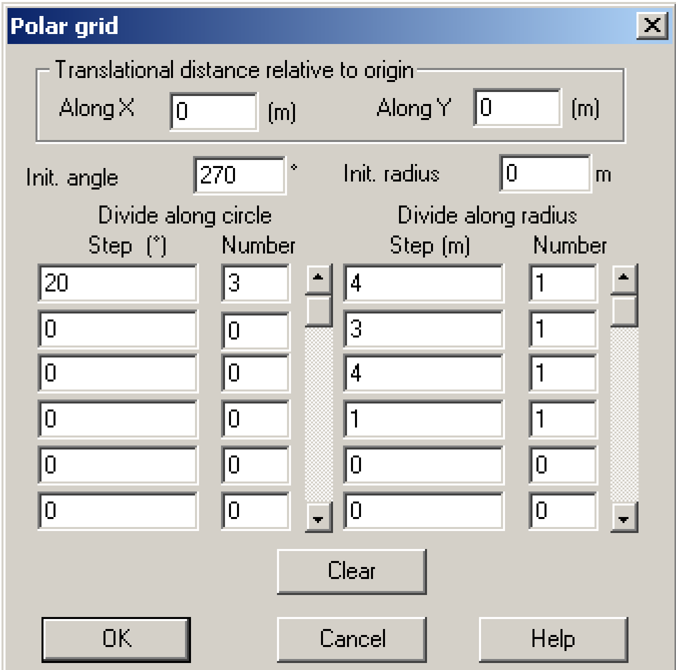
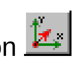


Figure 1.3.3 **Polar grid** dialog box

Move the origin of coordinate system to the point of intersection between axis **1a** and **A1** (as it presented in Figure 1.3.4) and rotate coordinate system.

➪ On the MODEL menu, point to **Coordinate system** and click **Move** (button  on the toolbar).

➪ When this mode is active, specify the node on the grid (see Fig.1.3.4).

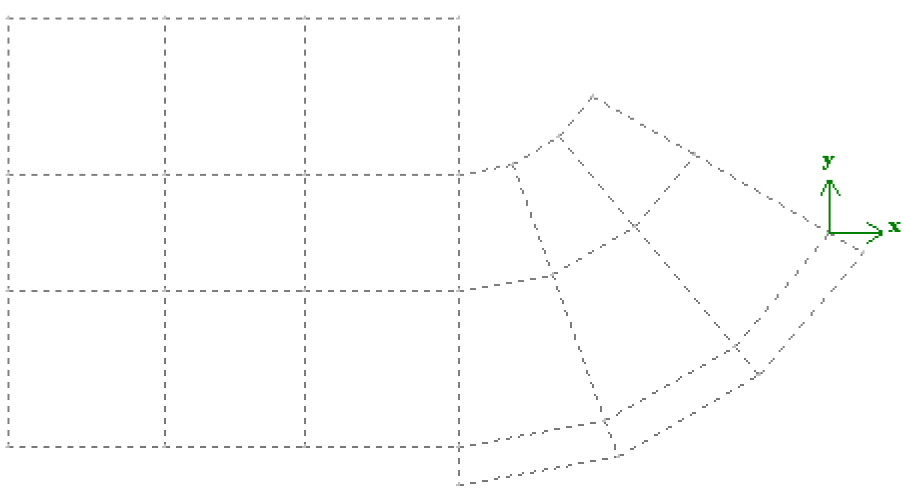


Figure 1.3.4 Move coordinate system

➪ To rotate coordinate system, on the MODEL menu, point to **Coordinate system** and click **Rotate**.

➪ When this mode is active, click any node on the grid so that the coordinate system will be located as presented in Figure 1.3.5.

*The **Rotate** command is also available in the shortcut menu. To open this menu, right-click the mouse button.*

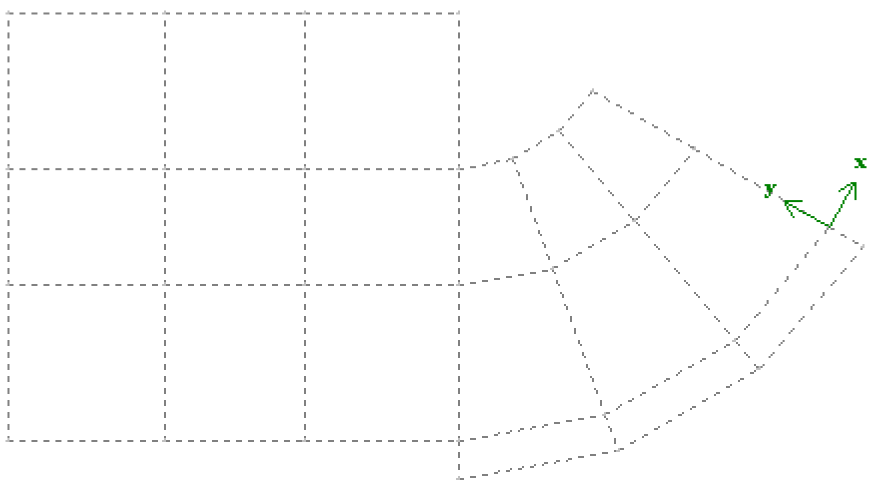


Figure 1.3.5 Rotate coordinate system

Define the second fragment of Cartesian grid between lines **1a** and **5a**, **A1** and **D1**.

- On the MODEL menu, point to **Grid** and click **Add fragment of Cartesian grid** (button on the toolbar).
- In the **Cartesian grid** dialog box, define the following parameters:
  - Divide along X / Divide along Y

Divide along X: Step (m)	Number	Divide along Y: Step (m)	Number
4	1	4	1
2.9	1	3	1
3.1	1	4	1
4	1		

  - other parameters remain by default.
- Click **OK**.
- To restore initial location of coordinate system, on the MODEL menu, point to **Coordinate system** and click **Initial location**.

The specified grid should look like the one presented in Figure 1.3.6.

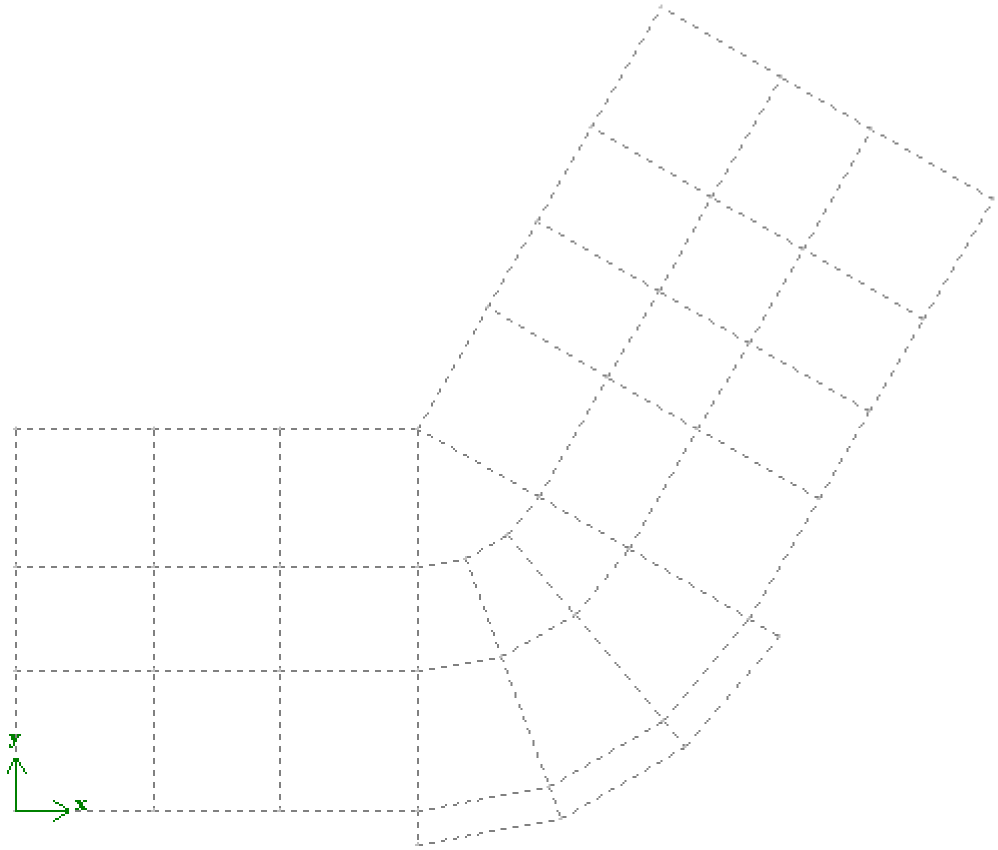


Figure 1.3.6 Specified grid

Construction lines of the building as well as the other auxiliary lines that will be used for model generation in future may be defined as grid lines.

To save data about design model:

- On the FILE menu, click **Save** (button on the toolbar).
- In the **Save as** dialog box specify the following data:
  - file name **Model 1**;
  - location where you want to save this file - select MONOMAKH-SAPR 2016 (directory where MONOMAKH-SAPR program is installed).
- Click **Save**.

File *Model1.chg* will be created on the disk.

To define construction lines of the building:

- On the MODEL menu, point to **Add** and click **Construction line** (button on the toolbar).
- In the **Add construction line** dialog box (see Fig.1.3.7), define the following parameters:
  - name of construction line **1** (name **a1** is displayed by default).

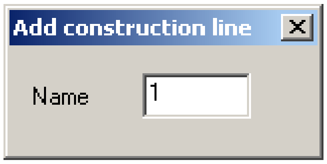


Figure 1.3.7 **Add construction line** dialog box

- Click two nodes of the grid so that the specified points will define location of construction line.
- In the **Add construction line** dialog box, define names of construction lines and specify location of construction lines 2, 3, 4 according to the plan of the building.

The specified construction lines should look like the ones presented in Figure 1.3.8.

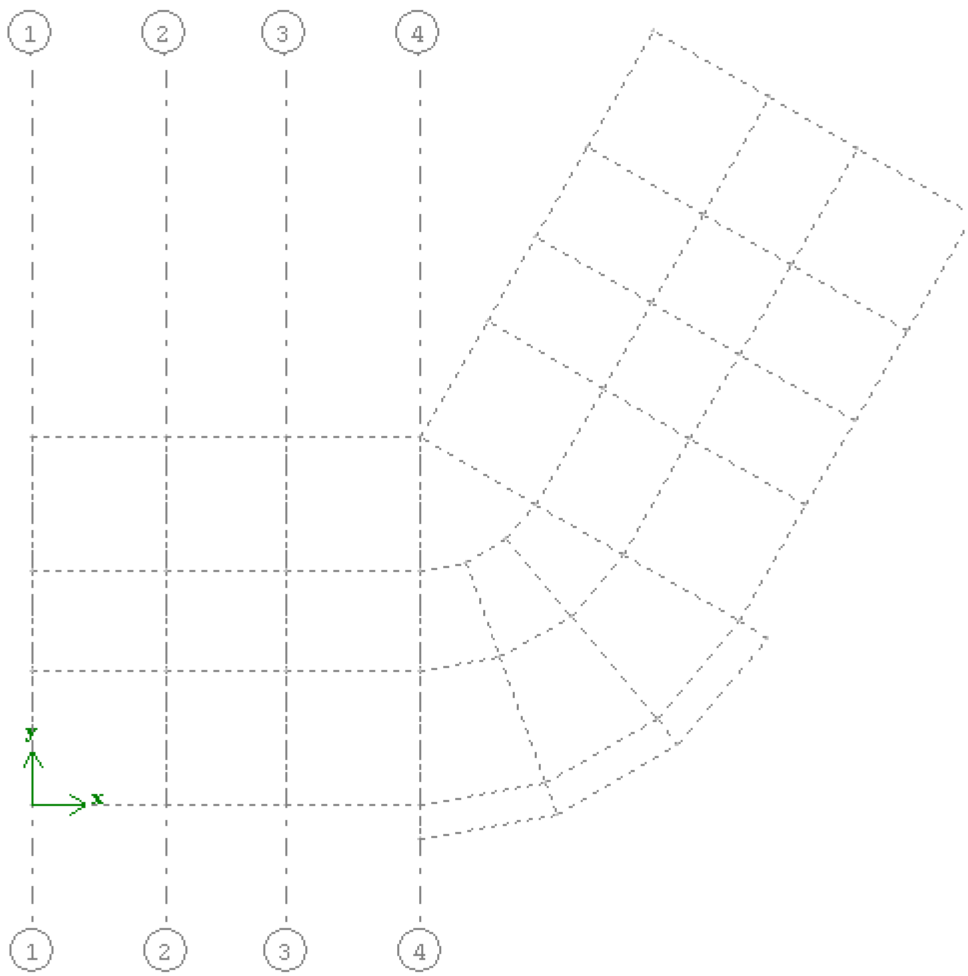


Figure 1.3.8 Construction lines

*For further work with the model, it is possible to display or hide construction lines as well as the other elements of the model. On the VIEW menu, click **Display options**. In the **Display options** dialog box, select the **Additional** tab, clear **Construction lines** check box and then click **Apply**. To do the same from the toolbar, click **Construction lines** button on the VISUALIZATION toolbar (see Fig.1.3.9).*

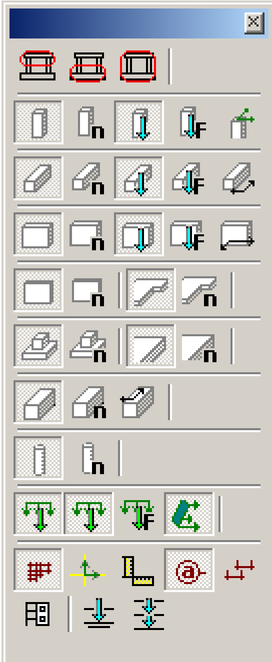


Figure 1.3.9 **Visualization** toolbar

Step 4. Defining columns

<u>To define group of columns:</u>

Define parameters and location of four columns C2 on lines I and II according to the plan of the building.

- On the MODEL menu, point to **Add** and click **Column** (button  on the toolbar).
- In the **Add column** dialog box (see Fig.1.4.1), define the following parameters:
  - material – RC B30 AIII AI
  - width of section b=0.24m;
  - height of section h=0.6m;
  - other parameters remain by default.

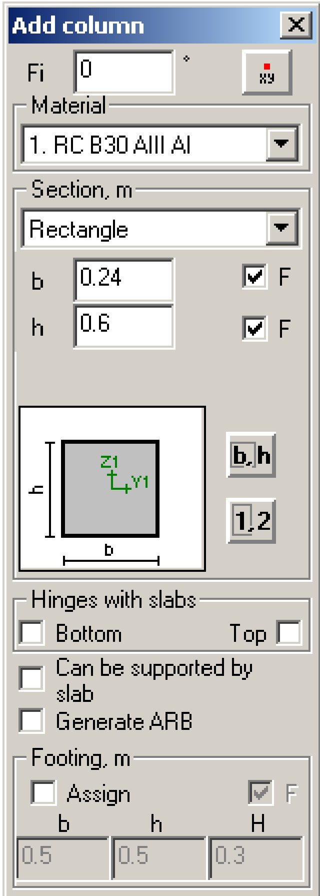


Figure 1.4.1 **Add column** dialog box

- On the MODEL menu, point to **Select** and click **Group pointer** (button on the toolbar).
- Select the rectangle that will include nodes of intersection of construction lines I, II with lines A1, B1 according to plan of the building.

To select rectangular fragment, click the first node of rectangle and then move the pointer diagonally and click once again.

The specified columns will look as presented in Figure 1.4.2.

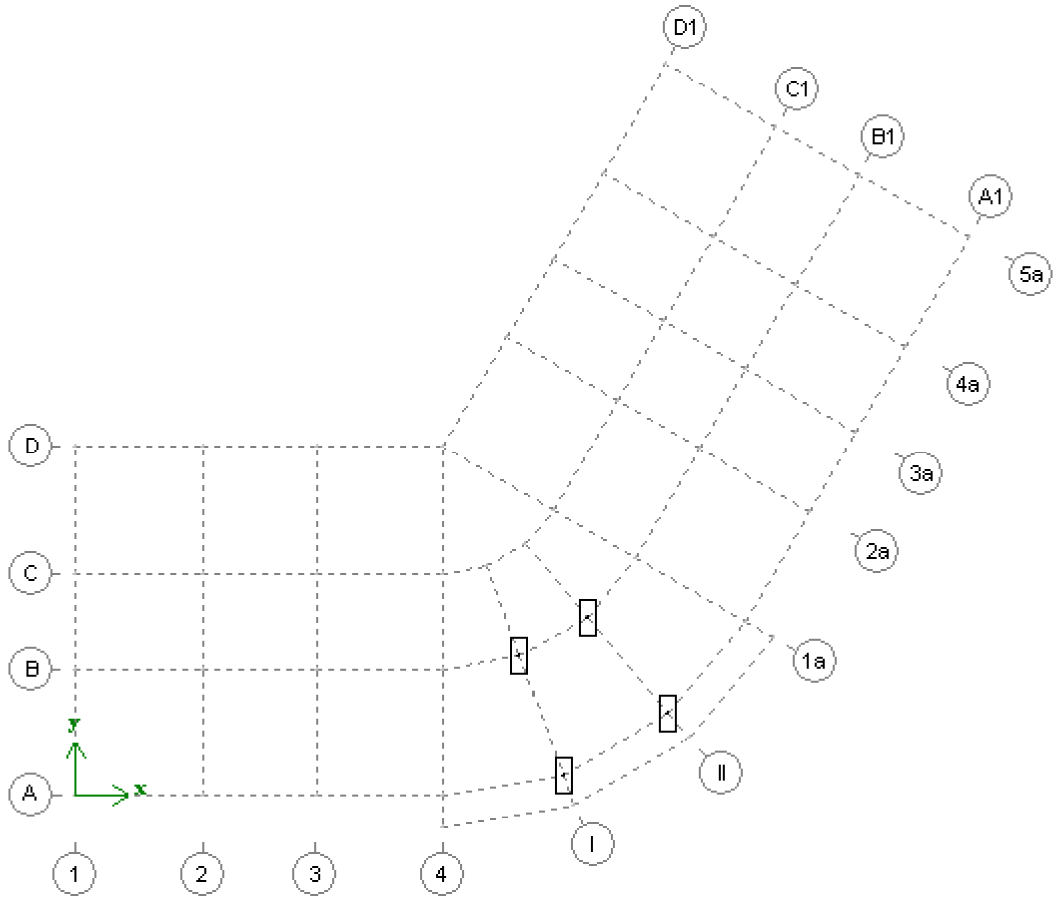


Figure 1.4.2 Group of columns

In this and further similar figures, notation for construction lines differs from the one displayed on the screen: visualization of construction lines is cancelled and they are drawn in Paint.

<u>To rotate local axes of group of columns:</u>
Rotate local axes of four columns C2 on lines I and II.

- On the VIEW menu, click **Display options**.
- In the **Display options** dialog box, on the **Local axes** tab, select the **Columns: Local axes Y1, Z1** check box and click **Apply**. To do the same from the toolbar, simply click the **Columns: Local axes Y1, Z1** button on the **Visualization** toolbar.
- To move the origin of coordinates to the centre of Polar grid at intersection of construction lines **4** and **D** (see Fig.1.4.3), on the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar).

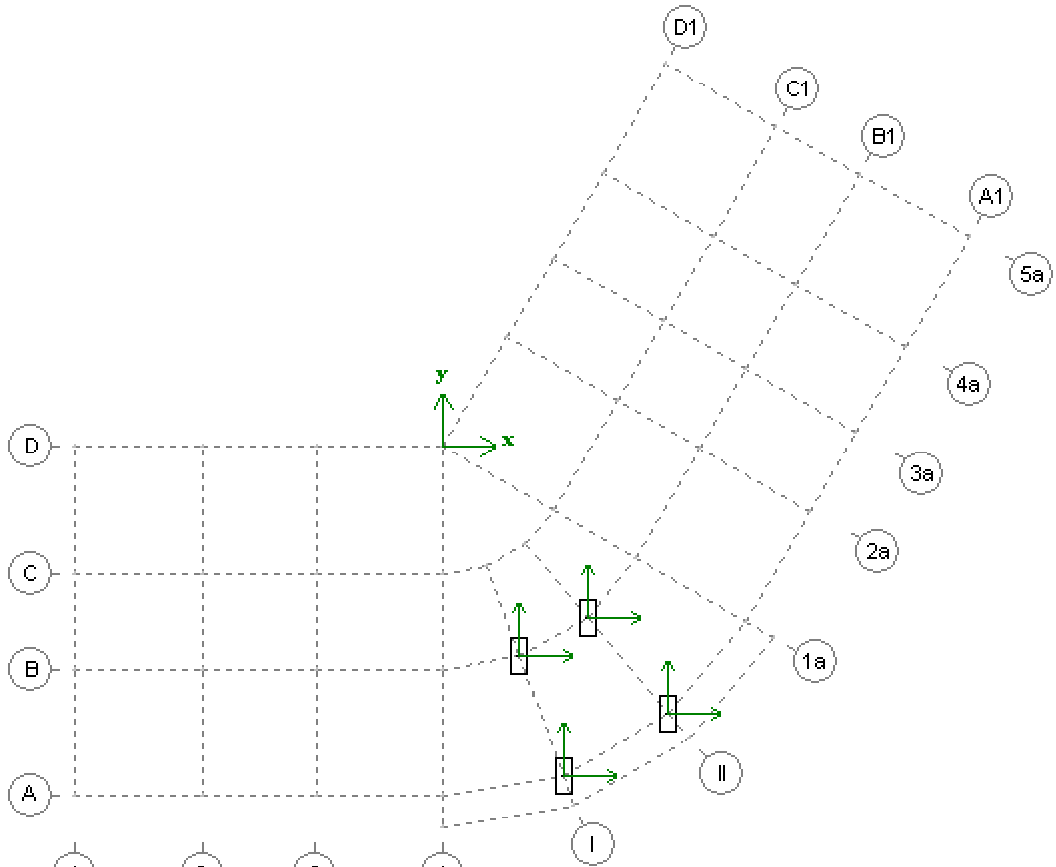


Figure 1.4.3 Local axes of columns

- To select columns, on the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).
- When this mode is active, click the **Group pointer** button and with the group pointer select the rectangular fragment that will include group of columns. Selected columns will be coloured red.

*Element selected on the model is coloured red. Note that commands to edit and delete are related to all selected elements. That's why, it is necessary to keep in mind the number of selected elements and which elements are selected. To unselect all elements, on the MODEL menu, point to **Select** and click **Unselect all** (button on the toolbar).*

- To rotate local axes of columns, on the MODEL menu, point to **Edit** and click **Direct columns** (button on the toolbar).
- In the **Direct columns** dialog box (see Fig.1.4.4), on the **Direct to point** tab , define the following parameters:
  - select **Z1-axis** option (default option is **Y1-axis**);
  - other parameters remain by default;
  - click **Apply** .

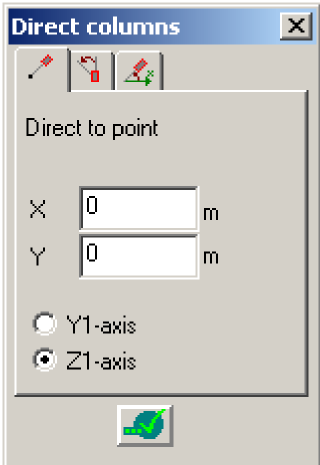


Figure 1.4.4 **Direct columns** dialog box

Rotated columns will look as presented in Figure 1.4.5.

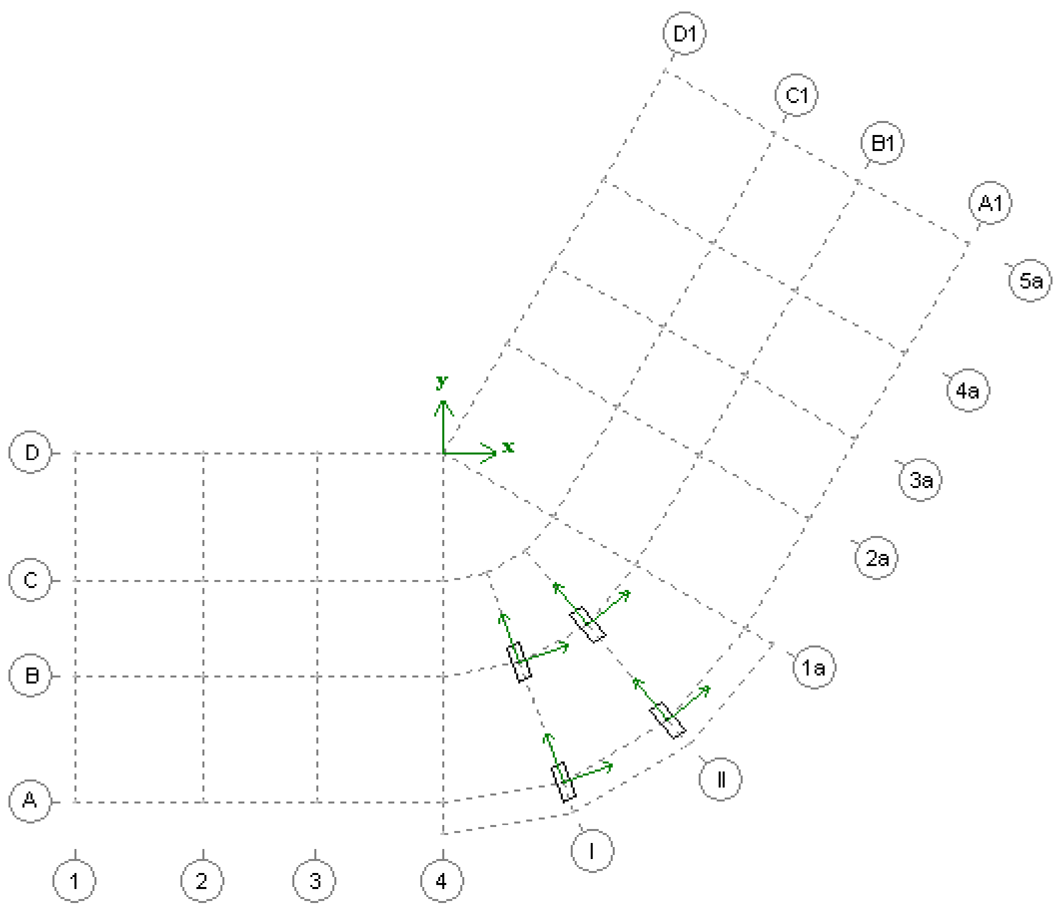


Figure 1.4.5 Group of columns after rotation of local axes

<u>To define a single column:</u>
Define parameters and location of column C1 at the intersection of lines **2** and **D** according to plan of the building.

- On the MODEL menu, point to **Add** and click **Column** (button on the toolbar).
- In the **Add column** dialog box, define the following parameters for column C1:

- width of section b=0.4m;
- height of section h=0.4m;
- other parameters remain by default.

➪ On the MODEL menu, point to **Select** and click **Single pointer** (button on the toolbar).

➪ Specify with the pointer node of intersection between line 2 and line D according to plan of the building.

The shape of the mouse pointer indicates which mode is selected at the moment:

– single pointer is selected (button on the toolbar is active), or

– group pointer is selected (button on the toolbar is active).

To copy a column:

Copy the column C1 located at intersection of lines **2** and **D** according to plan of the building.

➪ On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).

➪ Select the column at intersection between line 2 and line D. Selected column will be coloured red.

➪ To copy the column, on the MODEL menu, point to **Edit** and click **Copy and move** (button on the toolbar).

➪ In the **Copy and move** dialog box, on the **Move** tab (see Fig.1.4.6), define the following parameters:

- increment DX=3.6m;
- other parameters remain by default;
- click **Apply** .

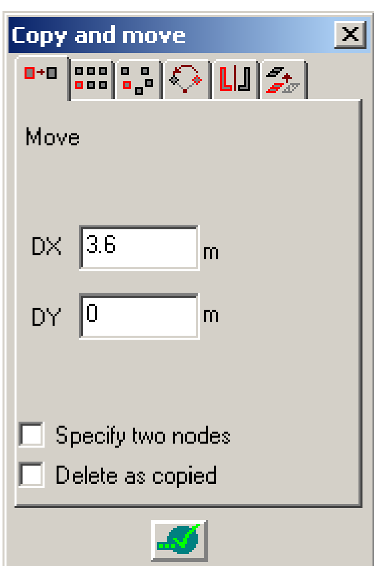


Figure 1.4.6 **Copy and move** dialog box

The new column will appear at intersection between line 3 and line D.

*It is possible to copy the column to the specified nodes on the model. To do this, on the MODEL menu, point to **Edit** and click **Copy and move** (button on the toolbar). In the dialog box, on the **Multiply by nodes** tab , select the **Specify nodes** check box. On the model, first of all, select the node of column that should be multiplied and then in sequence specify nodes on the model where similar columns should be placed. Then click **Apply** .*

<u>To define a column of complex shape:</u>

Define column C3 located at intersection of lines **4** and **A** according to plan of the building.

- On the MODEL menu, point to **Add** and click **Column** (button on the toolbar).
- In the **Add column** dialog box (see Fig.1.4.7), define the following parameters for column C3:
  - select the section shape **Cross**;
  - b1=0.3m;
  - b2=0.2m;
  - b3=0 m;
  - h1=0.3m;
  - h2=0.2m;
  - h3=0 m;
  - other parameters remain by default.

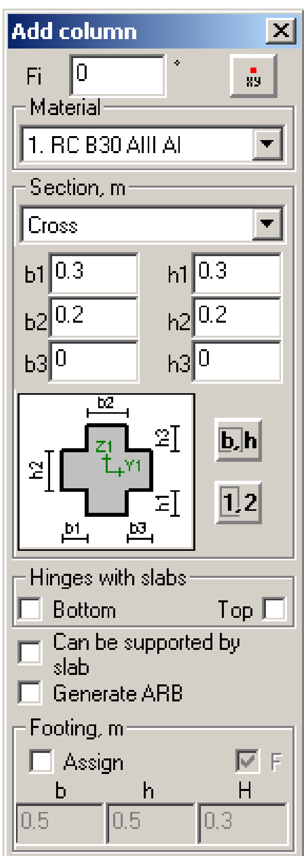


Figure 1.4.7 **Add column** dialog box

*To check the specified dimensions and section shape, click **Draw section with real values** button . Schematic presentation of the specified section will be displayed in the **Add column** dialog box. To go back to parametric presentation, click **Draw section with parameters** button.*

- With a single pointer (button on the toolbar), specify the node of intersection between line 4 and line A according to plan of the building.

To indicate columns on the model:

- To visualize numbers and parameters of columns, on the VIEW menu, click **Display options**.
- In the **Display options** dialog box, on the **Numbers and parameters** tab, define the following:
  - select the **Columns: Numbers and parameters** check box;
  - click **Apply** .

- To close the **Display options** dialog box, click **Close**.
- To do the same from the toolbar, click the **Columns: Numbers and parameters** button on the VISUALIZATION toolbar.

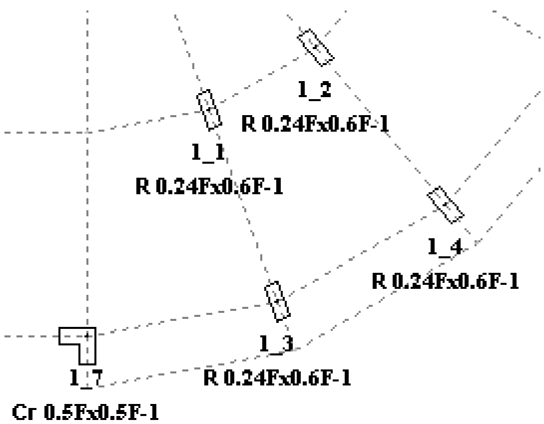


Figure 1.4.8 Notation for columns on the model

Storey No. and column No. are displayed for every column. In the lower row there are: section shape, overall dimensions of sections, fixation and number of material.

- Click the **Columns: Numbers and parameters** button once again to remove visualization of numbers and parameters of columns (the button should become inactive).

To define columns on fragment of a plan:
Define the complex shape column C3 located at intersection of lines **1a** and **A1** according to plan of the building.

- On the MODEL menu, point to **Coordinate system** and click **Fix**.
- On the model, define the node of intersection between lines **1a** and **A1**.
- On the model, define the node of intersection between lines **1a** and **D1**.

*The **Fix coordinate system** command is also available on the shortcut menu (this menu is presented when you right-click the mouse button).*

The coordinate system should be looked like the one on the Figure 1.4.9.

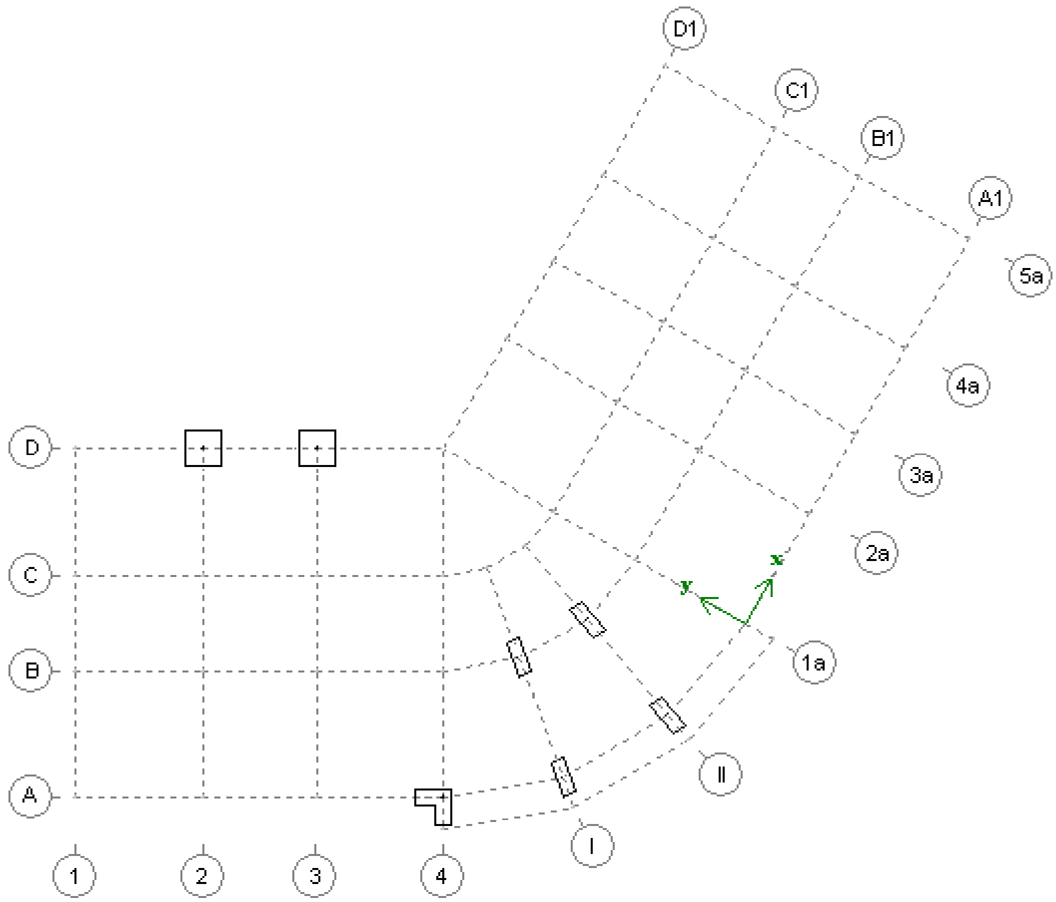


Figure 1.4.9 Fix coordinate system

*If you close the **Add column** dialog box by clicking and the **Add column** mode remains active (see the text in the Status bar), then to open the dialog box once again, it is necessary to close this mode and then activate it, that is, double-click **Add column** button on the toolbar.*

➪ In the **Add column** dialog box, define the following parameters:

- b1 = 0 m (section shape **Cross**);
- b2 = 0.2 m;
- b3 = 0.3 m;
- h1 = 0.3 m;
- h2 = 0.2 m;
- h3 = 0 m.

➪ Specify with the single pointer node of intersection between line **1a** and line **A1** according to plan of the building.

Define column C4 of complex section shape located at intersection of lines **4a** and **A1** according to plan of the building.

➪ In the **Add column** dialog box define the following parameters:

- b1 = 0.3 m (section shape **Cross**);
- b2 = 0.2 m;
- b3 = 0.3 m;
- h1 = 0 m;
- h2 = 0.2 m;
- h3 = 0.3 m.

➪ Specify with the single pointer node of intersection between line **4a** and line **A1** according to plan of the building.

Define column C3 of complex section shape located at intersection of lines **5a** and **A1** according to plan of the building.

➪ In the **Add column** dialog box define the following parameters:

- b1 = 0.3 m (section shape **Cross**);
- b2 = 0.2 m;
- b3 = 0 m;
- h1 = 0 m;
- h2 = 0.2 m;
- h3 = 0.3 m.

➪ Specify with the single pointer node of intersection between line **5a** and line **A1** according to plan of the building.

Define column C3 of complex section shape located at intersection of lines **5a** and **B1** according to plan of the building.

➪ In the **Add column** dialog box define the following parameters:

- b1 = 0.3 m (section shape **Cross**);
- b2 = 0.2 m;
- b3 = 0 m;
- h1 = 0.3 m;
- h2 = 0.2 m;
- h3 = 0 m.

➪ Specify with the single pointer node of intersection between line **5a** and line **B1** according to plan of the building.

Define column C4 of complex section shape located at intersection of lines **5a** and **C1** according to plan of the building.

- In the **Add column** dialog box define the following parameters:
  - b1 = 0.3 m (section shape **Cross**);
  - b2 = 0.2 m;
  - b3 = 0 m;
  - h1 = 0.3 m;
  - h2 = 0.2 m;
  - h3 = 0.3 m.
- Specify with the single pointer node of intersection between line **5a** and line **C1** according to plan of the building.

Define two columns C1 located at intersection of lines **4a** and **B1** and **4a** and **C1** according to plan of the building.

- In the **Add column** dialog box define the following parameters:
  - select the section shape **Rectangle**;
  - b = 0.4 m;
  - h = 0.4 m.

- With the single pointer specify node of intersection between line **4a** and line **B1** and then node of intersection between line **4a** and line **C1** according to plan of the building.

Note that local axes of the specified columns are directed according to the current location of coordinate system - the Y1-axis is parallel to the X-axis while the Z1-axis is parallel to the Y-axis.

- To restore initial location of coordinate system, on the MODEL menu, point to **Coordinate system** and click **Initial location**. (This command is also presented on the shortcut menu which appears on the screen when you right-click the mouse pointer.)

Defined columns will look like the columns presented in Figure 1.4.10.

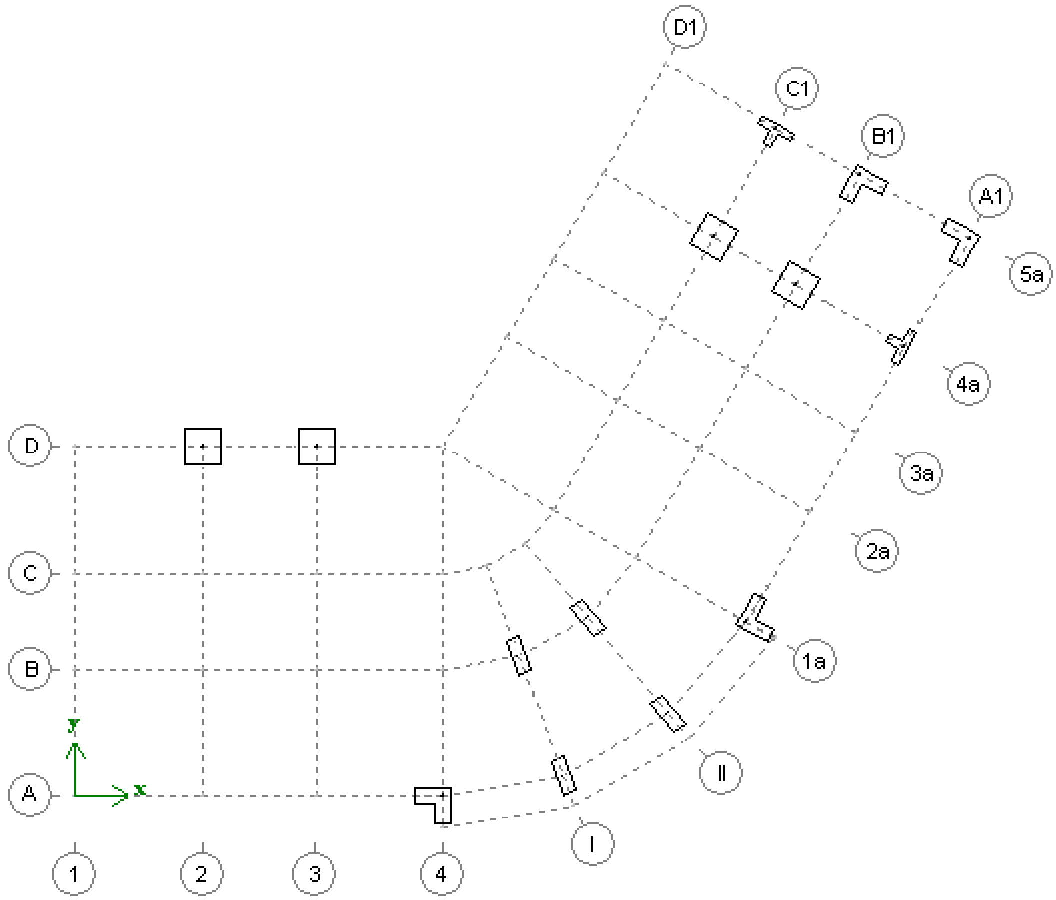


Figure 1.4.10 Columns of a regular storey

<u>To save data about the model:</u>

➪ On the FILE menu, click **Save** (button on the toolbar).

Step 5. Defining walls

<u>To define short wall (pylon):</u>

Pylons may be simulated with columns of rectangular section and with short walls (length up to 3 metres) as well.

Define parameters and location of pylon C2 located at intersection of lines **1** and **A** according to plan of the structure.

➪ To define parameters and location of pylon, on the MODEL menu, point to **Add** and click **Wall** (button on the toolbar).

➪ In the **Add wall** dialog box (see Fig.1.5.1), define the following parameters:

- thickness b = 0.24 m;
- material – RC B20 Al Al;
- other parameters remain by default.

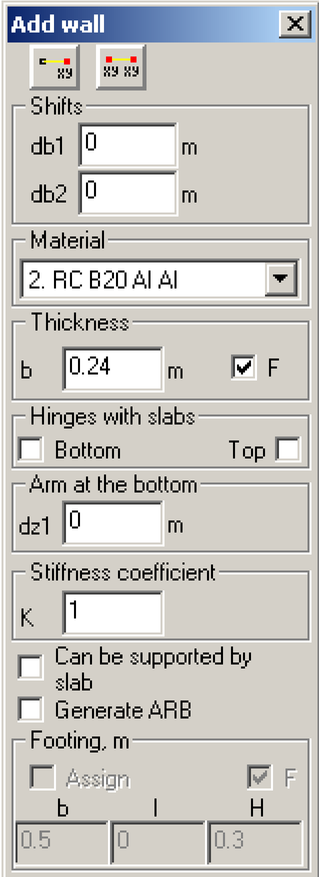


Figure 1.5.1 **Add wall** dialog box

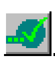
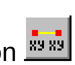
- Specify with the pointer node of intersection between line **1** and line **A** according to plan of the structure.
- In the **Add wall** dialog box, click the **Specify coordinates of node** button .
- In another window (see Fig.1.5.2), define the following parameters:
  - x = 0.6 m (by default, **Absolute** option is selected);
  - y = 0 m;
  - click **Apply** .

Figure 1.5.2 Dialog box to define coordinates of node

The short wall (pylon) will be added at node of intersection between line **1** and line **A**.

<u>To define short walls by coordinates:</u>
Define location of pylons C2 located at intersection of lines **1** and **B**, **1** and **D** according to plan of the structure.

- In the **Add wall** dialog box, click the **Specify coordinates of two nodes** button  .
- In another window (see Fig.1.5.3), define the following parameters:

- x1 = 0 m;
- y1 = 4 m;
- x2 = 0.6 m;
- y2 = 4 m;
- click **Apply** .

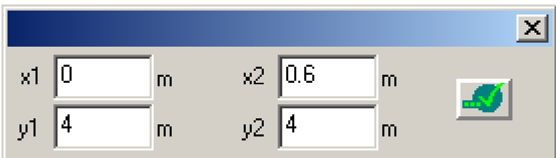


Figure 1.5.3 Dialog box to define coordinates of two nodes

The short wall (pylon) will be defined at the intersection of lines **1** and **B**.

- Specify with the pointer node of intersection between line **1** and line **D** according to plan of the structure.
- In the **Add wall** dialog box, click the **Specify coordinates of node** button .
- In another window, define the following parameters:
  - click **Relative** (by default, **Absolute** option is selected);
  - x = 0 m;
  - y = -0.6 m;
  - click **Apply** .

The specified walls will look as presented in Figure 1.5.4.

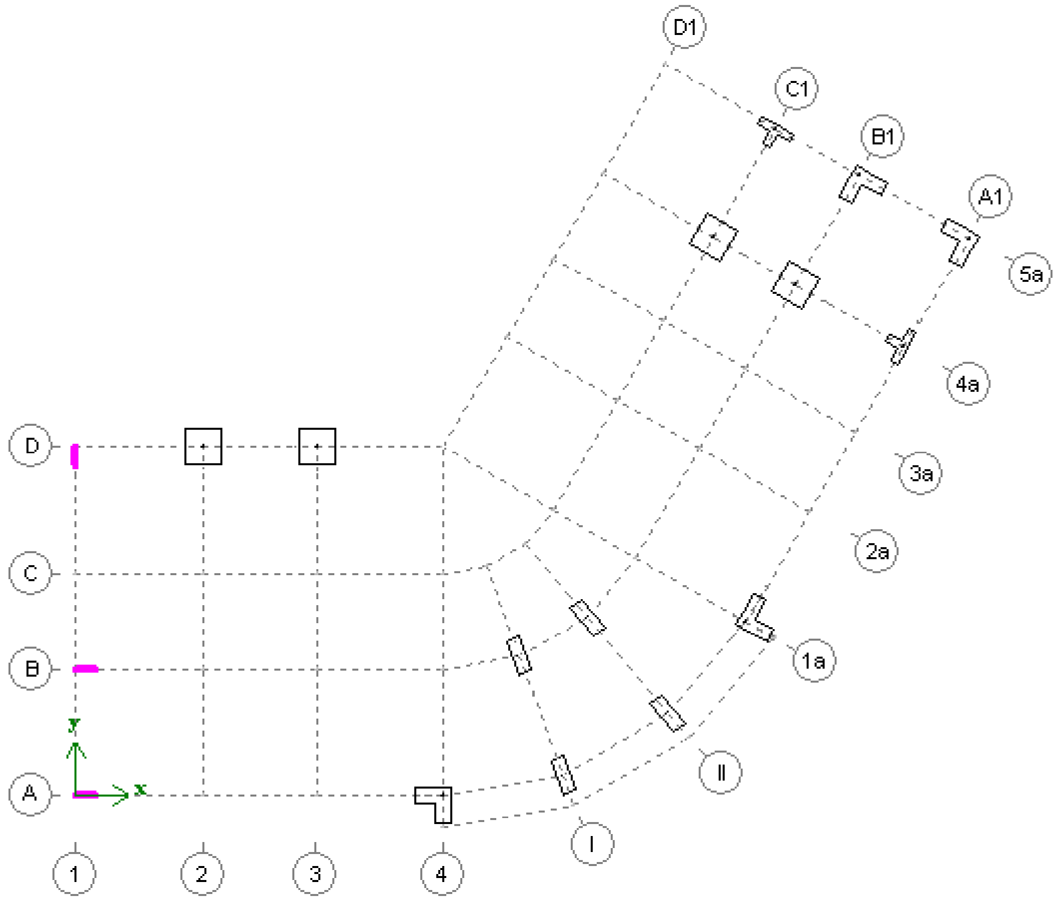


Figure 1.5.4 Short walls (pylons)

<u>To make mirror copy of short wall:</u>
Copy pylon C2 at the intersection of lines **1** and **B** according to plan of the structure.

- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).
- When this mode is active, select the wall at the intersection of lines **1** and **B**. Selected wall will be coloured red on the model.
- On the MODEL menu, point to **Edit** and click **Copy and move** (button on the toolbar).
- In the **Copy and move** dialog box, click the **Mirror** tab and define the following parameters:
  - click the **Mirror** tab (see Fig.1.5.5);
  - X1 = 5.8 m;
  - Y1 = -1 m;
  - X2 = 5.8 m;
  - Y2 = 1 m;
  - other parameters remain by default;
  - click **Apply** .

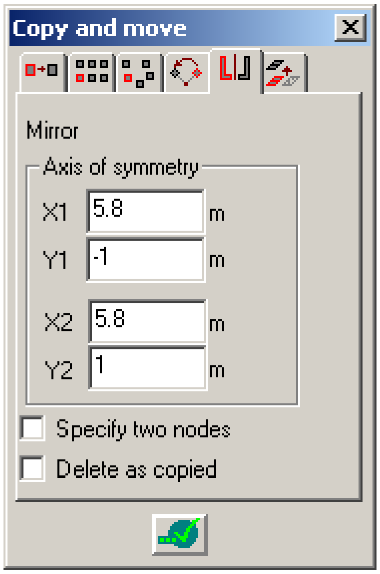


Figure 1.5.5 **Copy and move** dialog box (**Mirror copy** tab)

The new wall will be defined at the intersection of lines **4** and **B**.

<u>To copy short wall by two nodes and rotate the wall about the node:</u>
Copy short wall (pylon) C2 at the intersection between lines **4** and **B** according to plan of the structure.

- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).
- When this mode is active, select the wall at the intersection of lines **4** and **B**. Selected wall will be coloured red on the model.

*At this time **Copy and move** dialog box should remain open. If you close this dialog box, on the MODEL menu, point to **Edit** and click **Copy and move** (button on the toolbar).*

- In the **Copy and move** dialog box, do the following:
  - click the **Move** tab (see Fig.1.5.6);
  - select the **Specify two nodes** check box;
  - on the model, specify the first node — node of intersection of lines 4 and B according to plan of the structure;
  - on the model, specify the second node — node of intersection of lines 4 and C;

*The specified nodes will determine displacement vector. Coordinates of this vector will be displayed in the **Copy and move** dialog box (**Move** tab).*

  - click **Apply** .

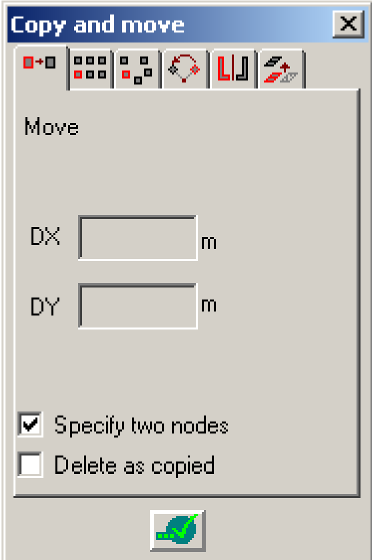


Figure 1.5.6 **Copy and move** dialog box (**Move** tab)

The new wall will appear at the intersection of lines **4** and **C**.

Rotate short wall (pylon) C2 at the intersection of lines **4** and **C** according to plan of the structure.

- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).
- When this mode is active, select this wall at the intersection of lines **4** and **C**.

- To move the origin, on the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin to intersection of lines **4** and **C** according to plan of the structure.
- In the **Copy and move** dialog box, do the following:
  - click the **Rotate** tab (see Fig.1.5.7);
  - select the **Delete as copied** check box;
  - define rotation angle Fi = -90 degrees;
  - click **Apply** .

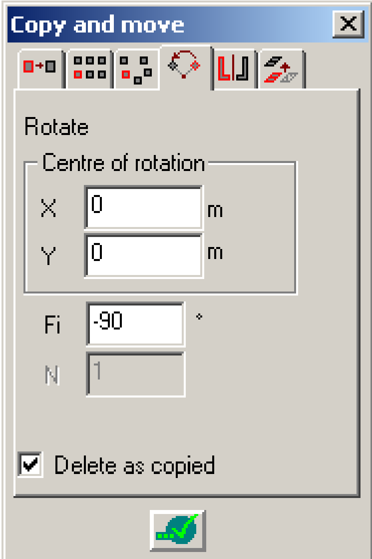


Figure 1.5.7 **Copy and move** dialog box (**Rotate** tab)

The wall at the intersection of lines **4** and **C** will be rotated.

<u>To copy short wall about the node and rotate wall about node:</u>

Copy short wall (pylon) C2 located at the intersection of lines **4** and **B** according to plan of the structure.

- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar). Then select the wall at the intersection of lines **4** and **B**.
- On the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin to intersection of lines **4** and **D** according to plan of the structure.
- In the **Copy and move** dialog box, click the **Rotate** tab and do the following:
  - clear the **Delete as copied** check box;
  - define rotation angle Fi = 60 degrees (number of copies remains by default as 1);
  - click **Apply**.

The specified walls will look as presented in Figure 1.5.8.

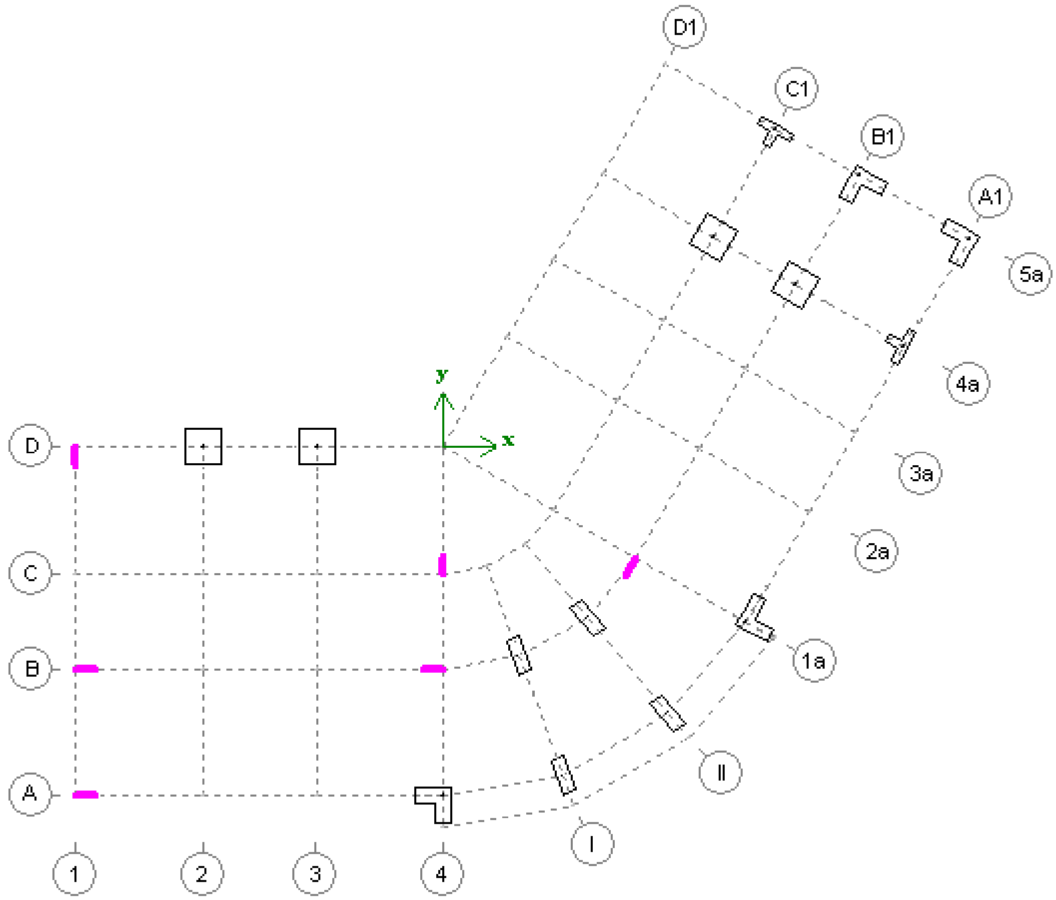


Figure 1.5.8 Copy short wall (pylon) about the node

Rotate short wall (pylon) C2 located at the intersection of lines **1a** and **B1** according to plan of the structure.

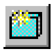
- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar). Then select the wall at the intersection of lines **1a** and **B1**.
- On the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin to intersection of lines **1a** and **B1** according to plan of the structure.
- In the **Copy and move** dialog box, click the **Rotate** tab and do the following:
  - select the **Delete as copied** check box;
  - define rotation angle Fi = 180 degrees;
  - click **Apply** .

The wall at intersection of lines **1a** and **B1** will be rotated.

- To restore initial location of coordinate system, on the MODEL menu, point to **Coordinate system** and click **Initial location**.

*To save the model, on the FILE menu, click **Save** (button on the toolbar).*

To define walls:

- To define parameters and location of walls, on the MODEL menu, point to **Add** and click **Wall** (button  on the toolbar).
- In the **Add wall** dialog box, define the following parameters:
  - material - RC B20 AI AI;
  - thickness b = 0.2 m;
  - other parameters remain by default.
- On the model, specify the first node — node of intersection of lines 2 and A according to plan of the structure;
- On the model, specify the second node — node of intersection of lines 3 and A.

The new wall appears along line A.

- In the same way, define all walls where nodes coincide with nodes of intersection of lines.
- On the model, specify node of intersection of lines 2 and A, then node of intersection of lines 2 and C.
- On the model, specify node of intersection of lines 3 and A, then node of intersection of lines 3 and C.
- On the model, specify node of intersection of lines 2 and C, then node of intersection of lines 3 and C.
- On the model, specify node of intersection of lines 1a and C1, then node of intersection of lines 1a and D1.
- On the model, specify node of intersection of lines 1a and D1, then node of intersection of lines 5a and D1.
- On the model, specify node of intersection of lines 2a and A1, then node of intersection of lines 3a and A1.
- On the model, specify node of intersection of lines 2a and A1, then node of intersection of lines 2a and C1.
- On the model, specify node of intersection of lines 2a and C1, then node of intersection of lines 3a and C1.
- On the model, specify node of intersection of lines 3a and A1, then node of intersection of lines 3a and C1.

The specified walls will look as presented in Figure 1.5.9.

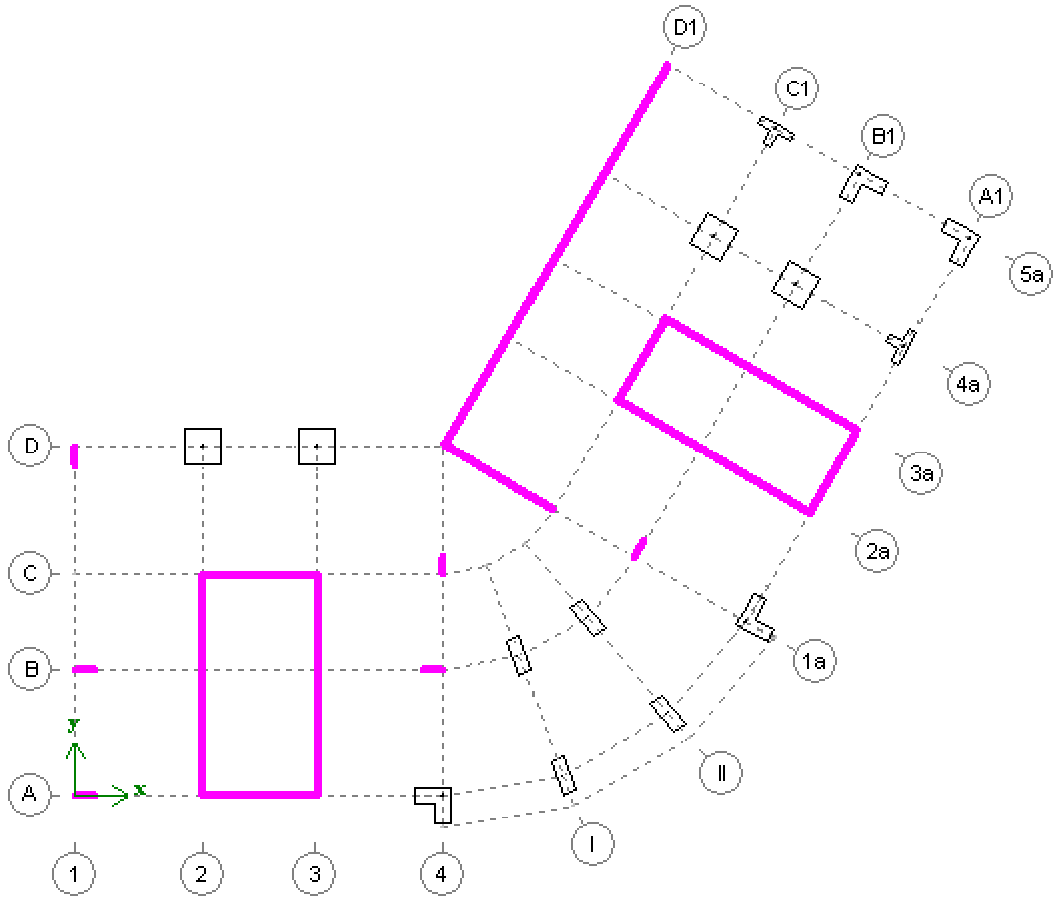


Figure 1.5.9 Walls

If the wall joins the column on the plan, then it is necessary to define the wall up to the column centre (not up to the column edge) so that the wall and the column will behave together in design model.

To mark walls on the model:

- To display numbers and parameters of walls on the model, on the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - click the **Numbers and parameters** tab ;
  - select the **Walls: Numbers and parameters** check box;
  - click **Apply** .
- To close the **Display options** dialog box, click **Close** .
- To do the same from the toolbar, just click the **Walls: Numbers and parameters** button on VISUALIZATION toolbar.

Storey No. and wall No. are displayed for every wall. In the lower row there are wall thickness, fixation and material code.

- To hide numbers and parameters of walls, click the **Walls: Numbers and parameters** button once again (the button should be inactive).

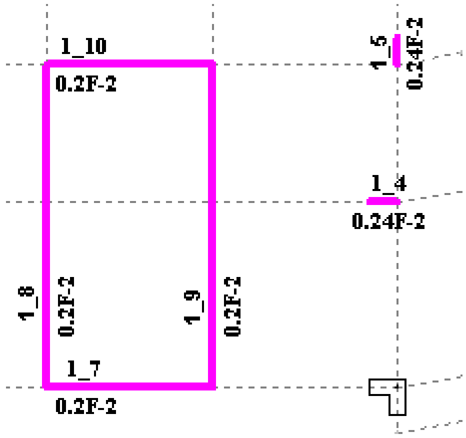


Figure 1.5.10 Notation for walls on the model

<u>To copy walls and edit wall length (shortening):</u>
Copy wall at line **C1** according to plan of the structure.

- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar). Then select the wall at the line **C1**.
- On the MODEL menu, point to **Coordinate system** and click **Fix**.
- Specify on the model the node of intersection between lines **2a** and **C1**.
- Specify on the model the node of intersection between lines **2a** and **D1**.

Coordinate system will be located as presented in Figure 1.5.11.

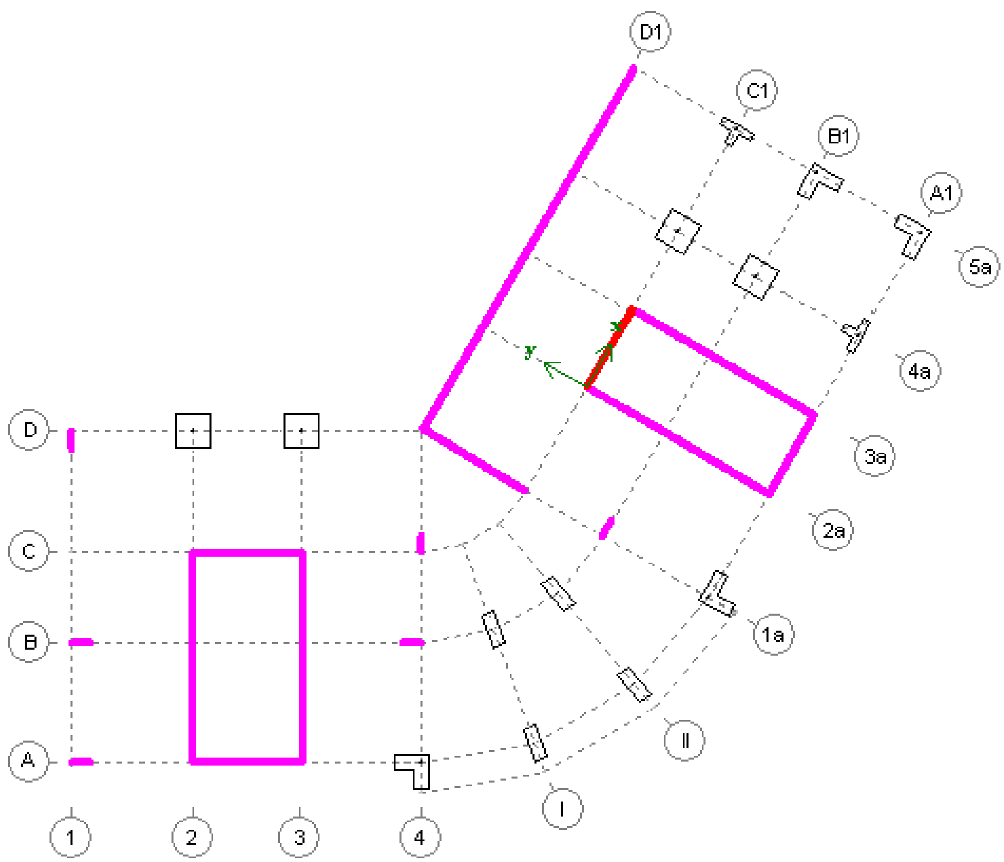


Figure 1.5.11 Fix coordinate system

- To copy the wall, on the MODEL menu, point to **Edit** and click **Copy and move** (button on the toolbar).
- In the **Copy and move** dialog box, on the **Move** tab , define the following parameters:
  - DX = 0 m;
  - DY = -1.2 m;
  - click **Apply** .
- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar). Then select the wall at the line **2a**.
- In the **Copy and move** dialog box, on the **Move** tab , define the following parameters:
  - DX = 1.45 m;
  - DY = 0 m;
  - click **Apply** .

The specified walls will look as presented in Figure 1.5.12.

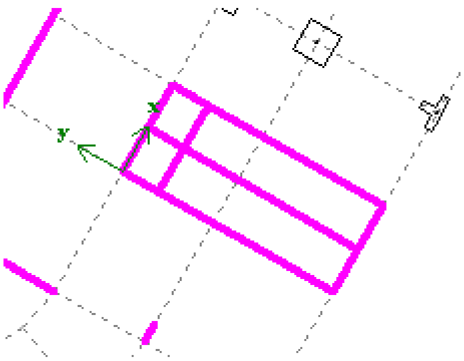


Figure 1.5.12 Walls (fragment of a plan)

- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar). Then select the wall that should be shortened (long wall between lines 2a and 3a).
- To edit length of selected wall, on the MODEL menu, point to **Edit** and click **Length** (button on the toolbar).
- In the **Edit length** dialog box (see Fig.1.5.13), define the following parameters:
  - select the **Specify two nodes** check box;
  - on the model, specify the first node — node of intersection of lines 2a and the wall (between lines B1and C1);
  - on the model, specify the second node — node of intersection of lines 3a and the wall (between lines B1and C1) as presented in Figure 1.5.14;

*The specified pair of nodes will determine the cutting line. Coordinates of this line will be presented in the **Edit length** dialog box.*

  - click **Apply** .

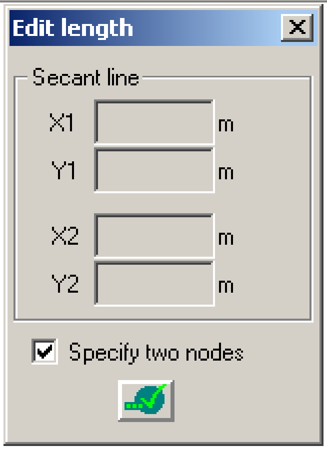


Figure 1.5.13 Specified nodes

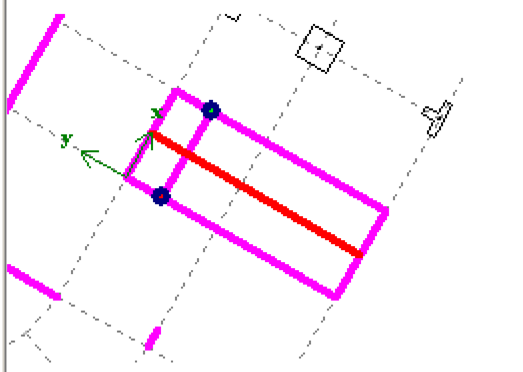


Figure 1.5.14 **Edit length** dialog box

The wall should be shortened as it presented in Figure 1.5.15.

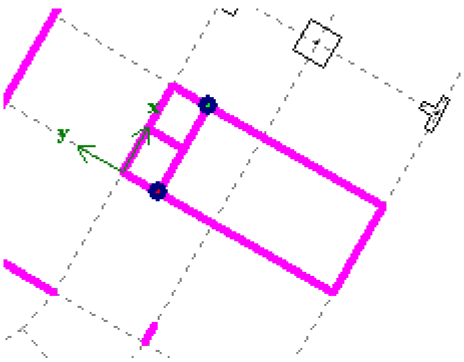


Figure 1.5.15 Walls (fragment of a plan)

*If you occasionally deleted the fragment of the wall and would like to restore it, on the EDIT menu, click **Undo** (button on the toolbar). Then click each of previously specified node, cancel node of cutting line and specify them on the model once again, but in reverse order. When you click **Apply**, the other fragment of the wall will be deleted.*

- To restore initial location of coordinate system, on the MODEL menu, point to **Coordinate system** and click **Initial location**.

<u>Multiple copy of the wall:</u>

- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar). Then select the wall at the line **C**.
- To copy the wall, on the MODEL menu, point to **Edit** and click **Copy and move** (button on the toolbar).
- In the **Copy and move** dialog box, do the following:
  - click the **Multiply** tab (see Fig.1.5.16);
  - define number of copies NY = 2;
  - DY = -1.5 m;
  - other parameters remain by default;

- click **Apply** .

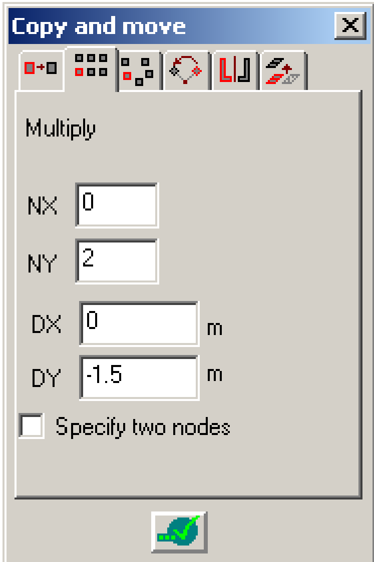


Figure 1.5.16 **Copy and move** dialog box (**Multiply** tab)

The specified walls will look as presented in Figure 1.5.17.

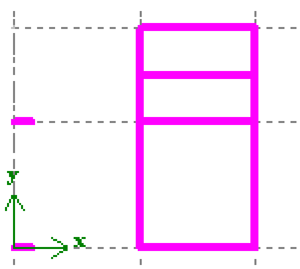


Figure 1.5.17 Walls (fragment of a plan)

<u>To edit length of wall in the Properties dialog box:</u>

- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar). Then select the wall No.1_19.
- On the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin to intersection of lines **2** and **B** according to plan of the structure.
- To display properties of selected element, on the **MODEL** menu, point to **Edit** and click **Properties** (button on the toolbar).
- In the **Wall No.1_19** dialog box (see Fig.1.5.18), do the following:
  - replace coordinate of the wall X2 = 3.6 m with coordinate X2 = 2 m;
  - other parameters remain by default;
  - click **Apply** .

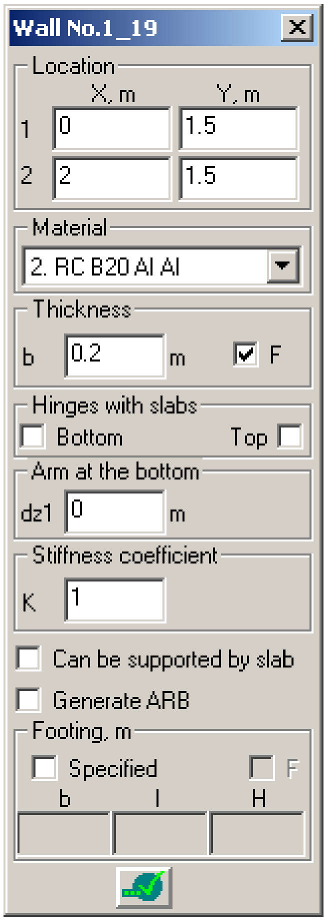


Figure 1.5.18 **Wall No.1_19** dialog box

The wall should be shortened as presented in Figure 1.5.19.

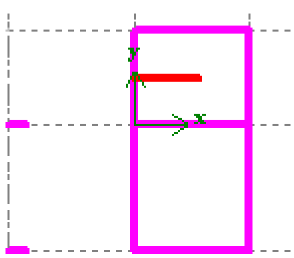


Figure 1.5.19 Walls (fragment of a plan)

- By default, **Inverse selection** option is active (button on the toolbar). To select another wall, on the MODEL menu, point to **Select**. Then point to **Action while pointing** and click **Select and cancel previous** (button on the toolbar).
- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar). Then select the wall No.1_20.
- In the **Wall No.1_20** dialog box, do the following:
  - replace coordinate of the wall X2 = 3.6 m with coordinate X2 = 2 m;
  - click **Apply** .

The second wall will be shortened.

To define the wall with coordinate axes:

- On the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin as presented in Figure 1.5.20.

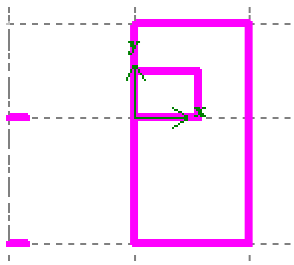


Figure 1.5.20 Move coordinate system

- On the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - click the **Additional** tab ;
  - select the **Axes of coordinates** check box;
  - click **Apply** .
- To close the **Display options** dialog box, click **Close** .
- To do the same from the toolbar, just click the **Axes of coordinates** button on VISUALIZATION toolbar.

Coordinate axes will be drawn from the current location of coordinate system (see Fig.1.5.21).

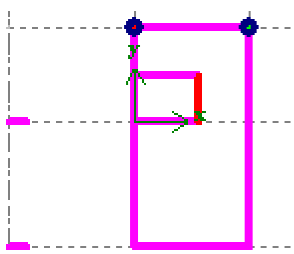


Figure 1.5.21 Axes of coordinates

Nodes where elements intersect with coordinate axes may be selected with a mouse pointer.

- On the MODEL menu, point to **Add** and click **Wall** (button on the toolbar).
- In the **Add wall** dialog box, define the following parameters:
  - thickness b = 0.2 m;
  - other parameters remain by default.
- Specify on the model origin of coordinate system and node of intersection between wall and line C.

The new wall will be added between lines 2 and 3.

- To hide presentation of coordinate axes, click the **Axes of coordinates** button on VISUALIZATION toolbar (the button should be inactive).

The specified walls should look as presented in Figure 1.5.22.

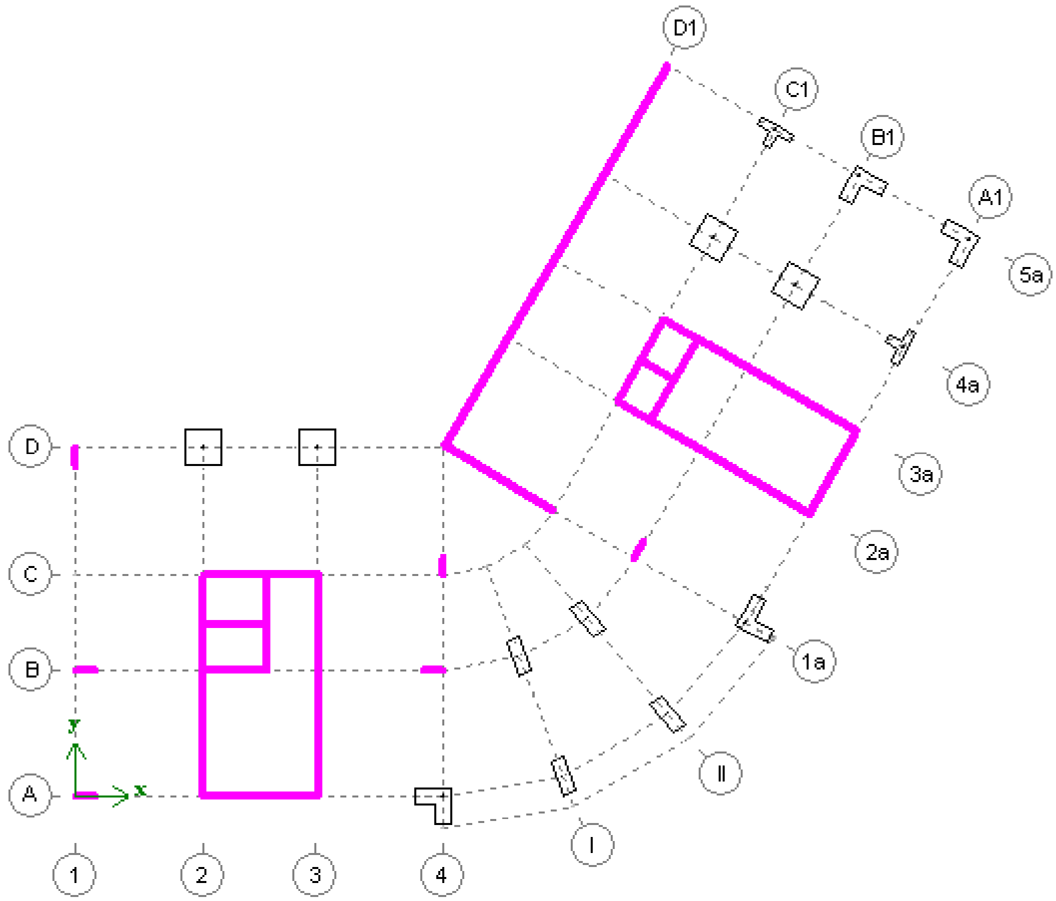


Figure 1.5.22 Walls of a regular storey

Step 6. Defining openings in walls

To define rectangular opening in a wall, copy opening and add it to database of openings:

- To define an opening D2 in a wall that is located between lines 2, 3 and lines B, C, on the **MODEL** menu, point to **Edit** and click **Openings in wall (partition)** (button on the toolbar).
- When this mode is active, specify the wall on the model.
- In the **Openings in walls** dialog box (see Fig.1.6.2), do the following:
  - under **Add opening**, click **Rectangular**;
  - in the **Opening** dialog box (see Fig.1.6.1), define the following parameters:
    - X = 0.3 m;
    - Y = 0 m;
    - a = 0.9 m;
    - b = 2.3 m;

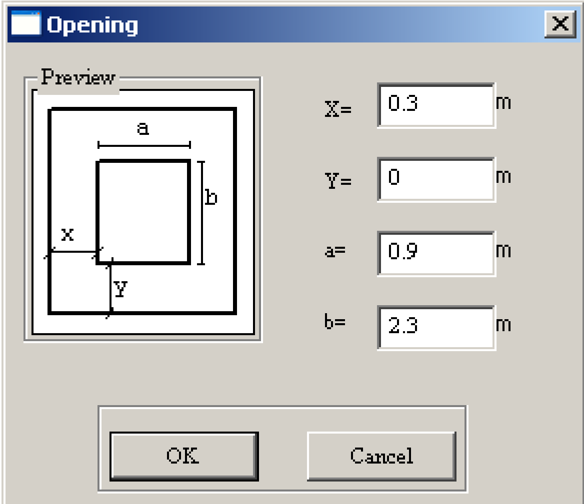


Figure 1.6.1 **Opening** dialog box

- click **OK** and the specified opening will be added to schematic presentation of a wall in the **Openings in walls** dialog box (see Fig.1.6.2);
- click on schematic presentation of a wall inside the opening contour and selected opening will be coloured red;
- to copy an opening, click **Copy**;
- in the **Copy openings** dialog box (see Fig.1.6.3), define the following parameters:
  - step along X = 1.5 m;
  - number along X = 1;
  - other parameters remain by default.

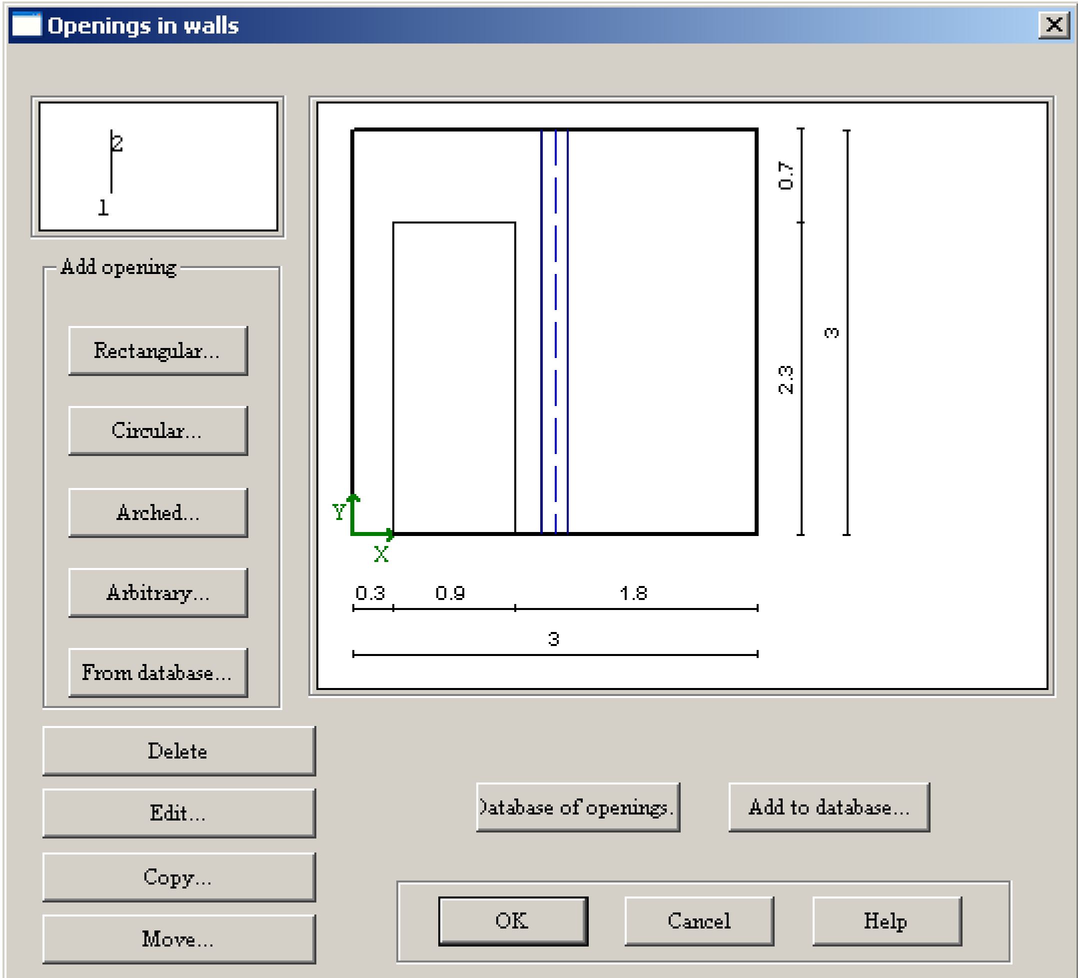


Figure 1.6.2 **Openings in walls** dialog box

- click **OK** and selected opening will be copied and added to schematic presentation of the wall in the **Openings in walls** dialog box;
- to add selected opening to database of openings, click **Add to database**;
- in the **Name of new opening** dialog box, define the name **Door D2** (by default, **Contur** name is displayed);
- click **OK** and selected opening will be added to the database and you will be able to use it for further work with other walls.
- In the **Openings in walls** dialog box, click **OK**.

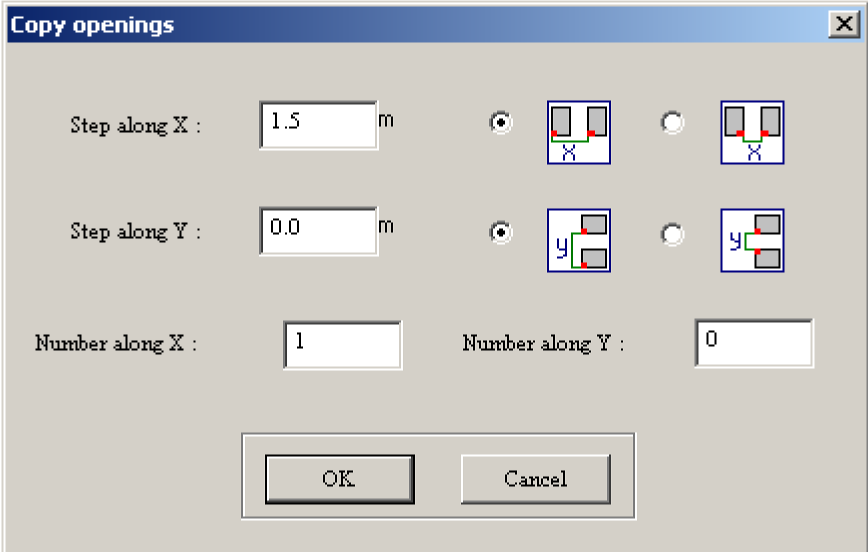


Figure 1.6.3 **Copy openings** dialog box

Openings defined in the wall will be presented on the plan (see Fig.1.6.4).

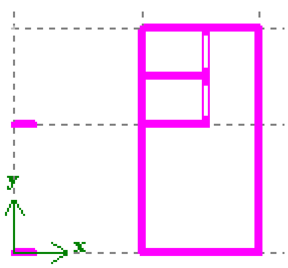


Figure 1.6.4 Openings in wall (fragment of a plan)

<u>To define rectangular and arbitrary openings in database of openings for walls:</u>

- To add rectangular opening D1 to the database of openings, on the **OPTIONS** menu, click **Database of openings in walls**.
- In the **Database of openings** dialog box (see Figure1.6.5), do the following:
  - under **Add opening**, click **Rectangular**;
  - in the **Opening** dialog box, define the following parameters:
    - a = 1.2 m;
    - b = 2.3 m;
  - click **OK** and specified opening will be added to the list of openings in the **Database of openings** dialog box;
  - click the line with name of the opening (by default, **Contur 1**) and the contour of the opening will be presented in the **Preview** area of the dialog box;
  - click the name of the opening once again and rename the opening as **Door D1**.

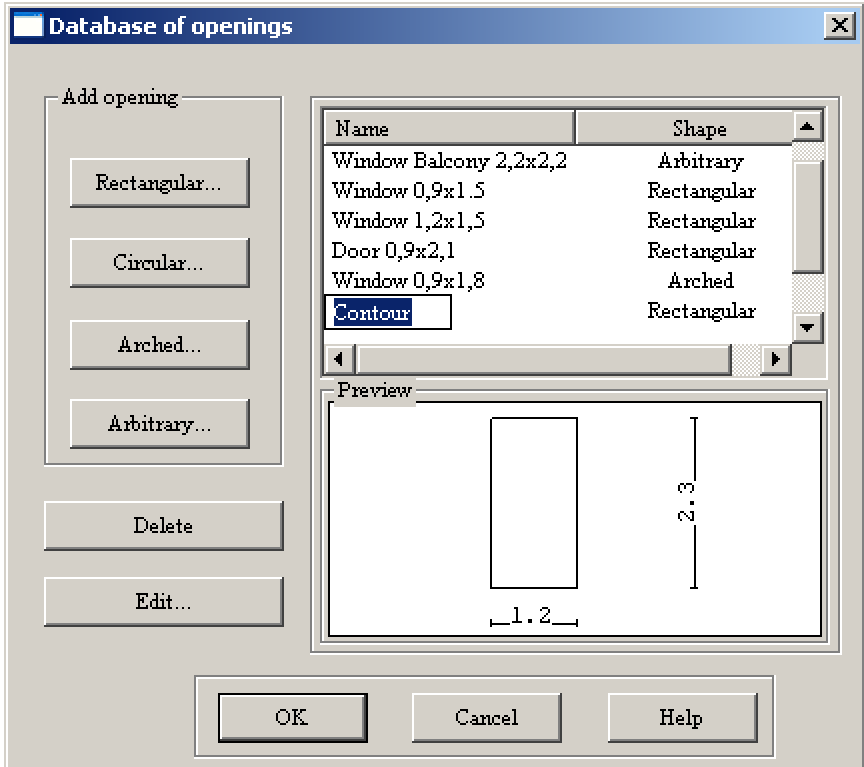


Figure 1.6.5 **Database of openings** dialog box

➪ To add rectangular opening W1, in the **Database of openings** dialog box, do the following:

- under **Add opening**, click **Rectangular**;
- in the **Opening** dialog box, define the following parameters:
  - a = 1.2 m;
  - b = 1.5 m;
- click **OK** and specified opening will be added to the list of openings in the **Database of openings** dialog box;
- click the line with name of the opening (by default, **Contur 1**) and the contour of the opening will be presented in the **Preview** area of the dialog box;
- click the name of the opening once again and rename the opening as **Window W1**.

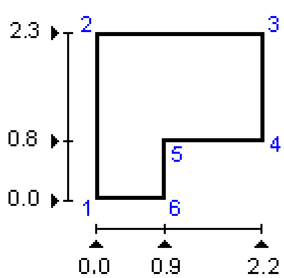


Figure 1.6.6 Numbers and coordinates of nodes of arbitrary opening

➪ To add arbitrary opening WB1, in the **Database of openings** dialog box, do the following:

- under **Add opening**, click **Arbitrary**;
- in the **Opening** dialog box (see Fig.1.6.7), define the following parameters:
  - select the **Display coordinates of nodes** check box;

- define X = 0 m (coordinates of the first node of the opening coincide with default values);
- Y = 0 m;
- click **Add after**;
- X = 0 m (the second node);
- Y = 2.2 m;
- click **Add after**;
- X = 2.2 m (the third node);
- Y = 2.3 m;
- click **Add after**;
- X = 2.2 m (the fourth node);
- Y = 0.8 m;
- click **Add after**;
- X = 0.9 m (the fifth node);
- Y = 0.8 m;
- click **Add after**;
- X = 0.9 m (the sixth node);
- Y = 0.0 m;
- click **Add after**;

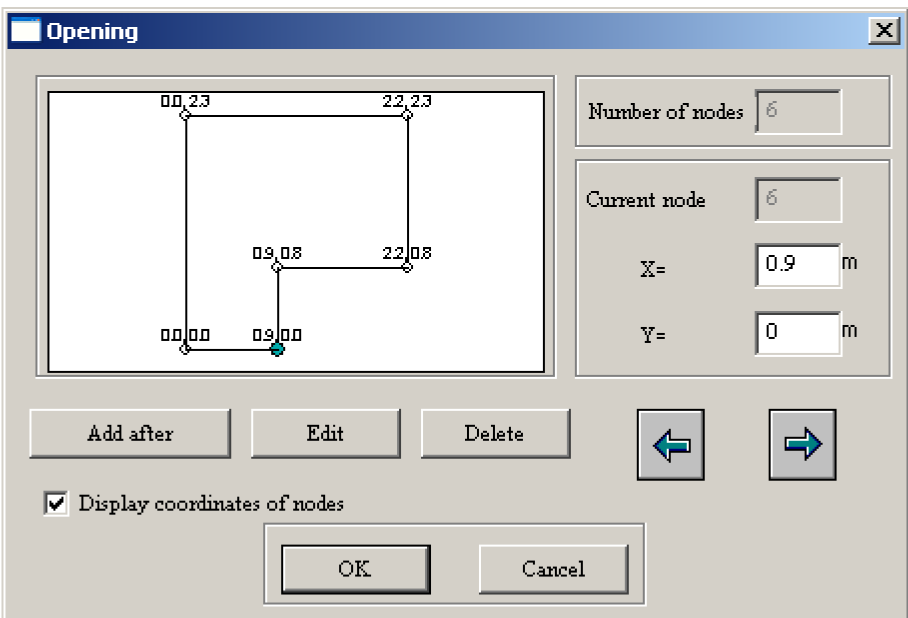


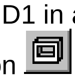
Figure 1.6.7 **Opening** dialog box

- click **OK** and specified opening will be added to the list of openings in the **Database of openings** dialog box;
- click the line with name of the opening (by default, **Contur 1**) and the contour of the opening will be presented in the **Preview** area of the dialog box;
- click the name of the opening once again and rename the opening as **Window Balcony WB1**.

➪ Click **OK**.

Database of openings will contain three new openings.

<u>To define openings from the database of openings for walls:</u>

➪ To define an opening D1 in a wall along line 2, on the MODEL menu, point to **Edit** and click **Openings in wall (partition)** (button  on the toolbar).

➪ When this mode is active, specify the wall on the model.

➪ In the **Openings in walls** dialog box, do the following:

*Notice orientation of the wall specified on the model. The 1st and the 2nd nodes of the wall are presented at the small image in the upper left corner of the **Openings in walls** dialog box. They represent the order in which nodes were specified when you defined the wall and determine location of local coordinate system of the wall. When you define an opening, its coordinates should be specified in local coordinate system of the wall.*

- under **Add opening**, click **From database**;
- in the **Add opening from database** dialog box (see Fig.1.6.8), do the following:
  - select Door D1 from the list;
  - specify X = 0.8 m.
- click **OK** and specified opening will be added to schematic presentation of the wall in the **Openings in walls** dialog box.

➪ Click **OK**.

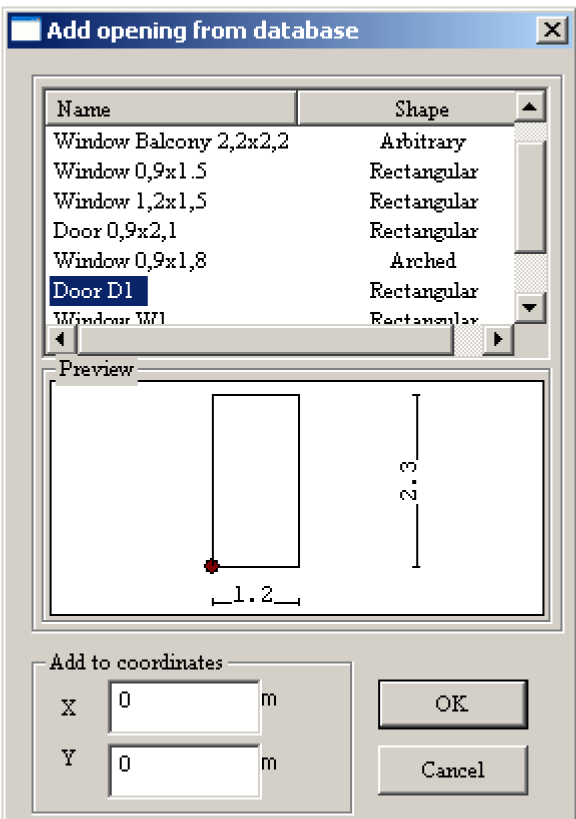


Figure 1.6.8 **Add opening from database** dialog box

➪ To define an opening D1 in a wall along line 3, specify the wall on the model. In this case the **Openings in wall (partition)** mode should be still active.

➪ In the **Openings in walls** dialog box, do the following:

- under **Add opening**, click **From database**;

- in the **Add opening from database** dialog box, do the following:
  - select Door D1 from the list;
  - specify X = 0.8 m
- click **OK** and specified opening will be added to schematic presentation of the wall in the **Openings in walls** dialog box.

➪ Click **OK**.

The openings specified in the walls should look as presented in Figure 1.6.9.

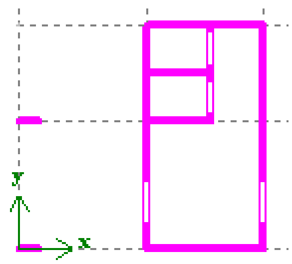


Figure 1.6.9 Openings in walls (fragment of plan)

➪ To define an opening D1 in a wall along line 2a, specify the wall on the model. In this case the **Openings in wall (partition)** mode should be still active.

➪ In the **Openings in walls** dialog box, do the following:

- under **Add opening**, click **From database**;
- in the **Add opening from database** dialog box, do the following:
  - select Door D1 from the list;
  - specify X = 4.3 m
- click **OK** and specified opening will be added to schematic presentation of the wall in the **Openings in walls** dialog box.

➪ Click **OK**.

*If the specified opening does not appear at place where it should be, on the EDIT menu, click **Undo** (button on the toolbar), then define an opening once again, but with another coordinate of insertion point, for example, X = 1.5 m.*

➪ To define an opening D1 in a wall along line 3a, specify the wall on the model. In this case the **Openings in wall (partition)** mode should be still active.

➪ In the **Openings in walls** dialog box, do the following:

- under **Add opening**, click **From database**;
- in the **Add opening from database** dialog box, do the following:
  - select Door D1 from the list;
  - specify X = 4.3 m
- click **OK** and specified opening will be added to schematic presentation of the wall in the **Openings in walls** dialog box.

➪ Click **OK**.

The openings specified in the walls should look as presented in Figure1.6.10.

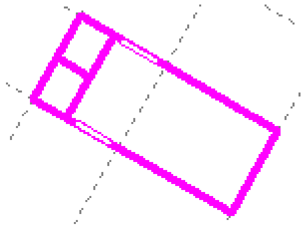


Figure 1.6.10 Openings in walls (fragment of plan)

To define openings from database of openings for walls, to copy and move an opening:

Openings may be copied only within the limits of one wall.

- To define openings W1 and WB1 in a wall along line D1 (see Fig.1.6.11), specify the wall on the model. In this case the **Openings in wall (partition)** mode should be still active.

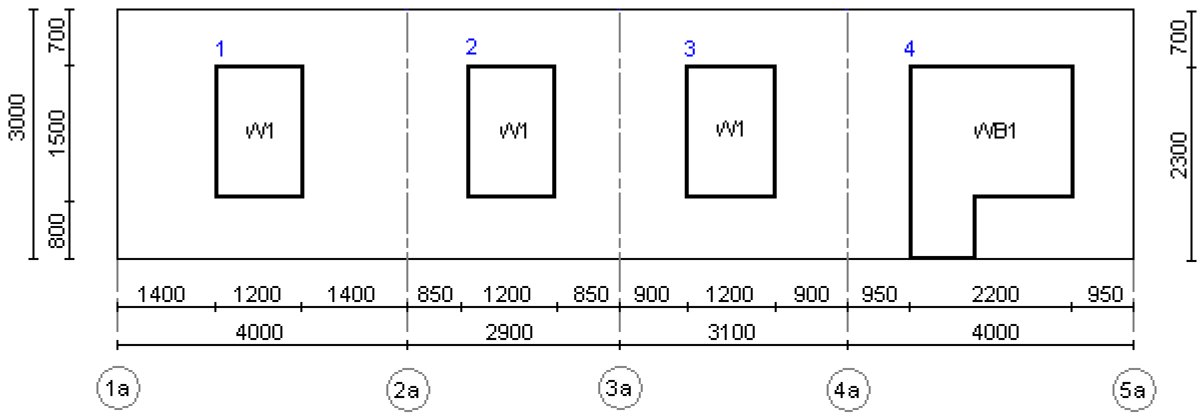


Figure 1.6.11 Numbers and dimensions of openings

- In the **Openings in walls** dialog box (see Fig.1.6.12), do the following:
  - under **Add opening**, click **From database**;
  - in the **Add opening from database** dialog box, do the following:
    - select Window W1 from the list;
    - specify X = 1.4 m;
    - Y = 0.8 m.

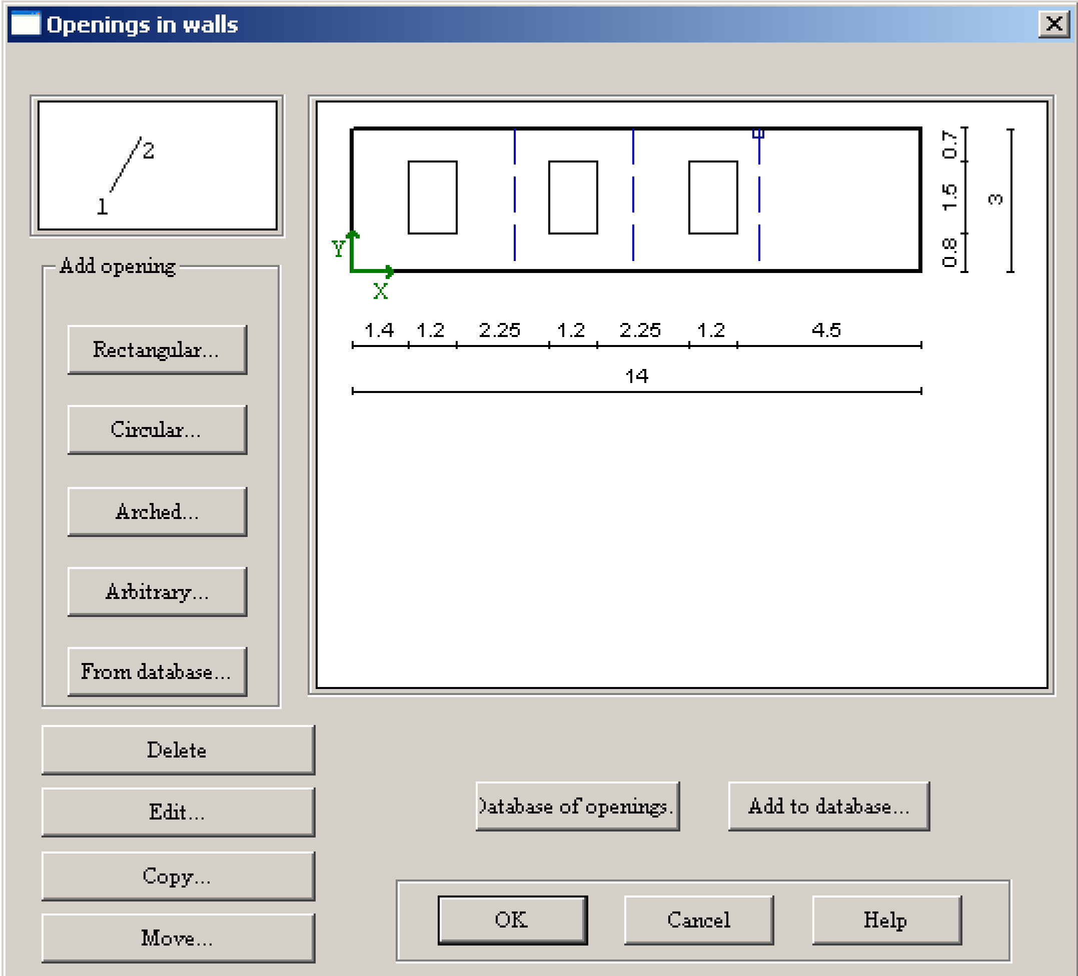


Figure 1.6.12 **Openings in walls** dialog box

- click **OK** and specified opening 1 will be added to schematic presentation of the wall in the **Openings in walls** dialog box;
- click on schematic presentation of a wall inside the opening 1 contour and selected opening will be coloured red;
- to copy an opening, click **Copy**;
- in the **Copy openings** dialog box, define the following parameters:
  - step along X = 3.45 m;
  - number along X = 2;
  - other parameters remain by default.
- click **OK** and specified openings 2 and 3 will be added to schematic presentation of the wall in the **Openings in walls** dialog box;
- click on schematic presentation of a wall inside the opening 1 contour (the first opening is still coloured red) to unselect it;
- click on schematic presentation of a wall inside the opening 3 contour and selected opening will be coloured red (see Fig.1.6.12);
- click **Move**;
- in the **Copy openings** dialog box (see Fig.1.6.13), define the following parameters:
  - step along X = -0.45 m;
  - other parameters remain by default.

- click **OK** and specified opening 3 will be moved to the left;
- under **Add opening**, click **From database**;
- in the **Add opening from database** dialog box, do the following:
  - select Window Balcony WB1 from the list;
  - specify X = 10.9 m
- click **OK** and specified opening 4 will be added to schematic presentation of the wall in the **Openings in walls** dialog box.

➪ Click **OK**.

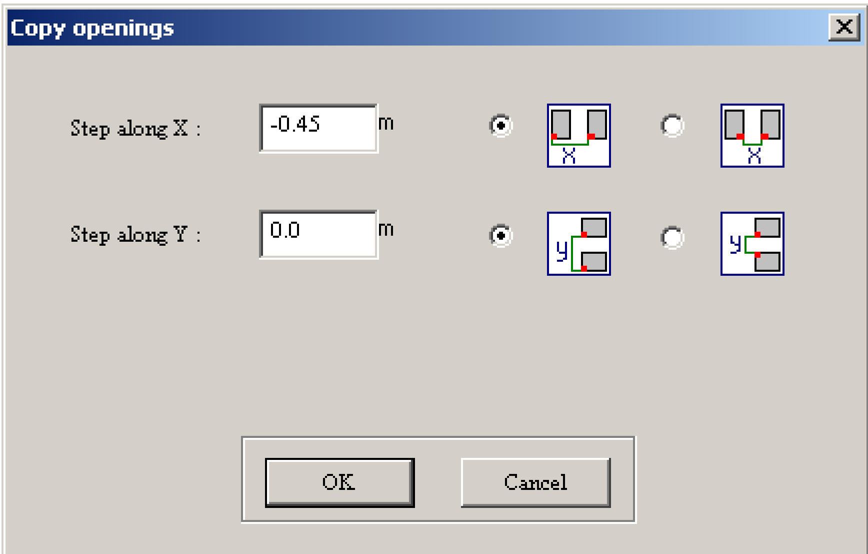


Figure 1.6.13 **Copy openings** dialog box

Openings defined for the wall will be presented on plan.

<u>To present 3D view of the model:</u>

➪ To present 3D view of the model, on the VIEW menu, point to **3D View** and then click **Current storey**.

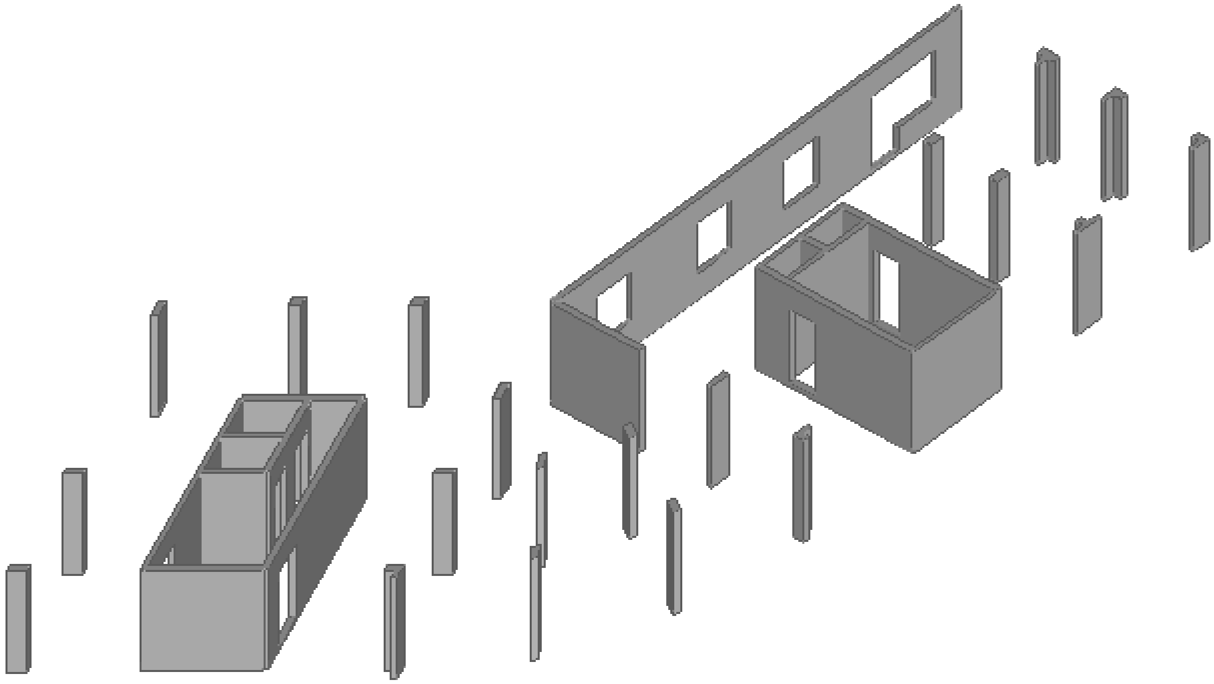
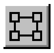


Figure 1.6.14 3D view of the model

➪ To return to the main view, on the VIEW menu, click **Main view** (button  on the toolbar).

➪ To save the model, on the FILE menu, click **Save** (button  on the toolbar).

Step 7. Defining floor slabs

<u>To define contour of floor slab:</u>

 *Any of specified contours including contour of floor slab may be edited later on. You could sketch contour by grid nodes and then edit it with account of balconies, adjoining openings, etc.*

 *Note that the floor slab is located at the upper level of the current storey **above** columns and walls of the current storey.*

➪ To define parameters and contour of floor slab, on the MODEL menu, point to **Add** and click **Add slab** (button  on the toolbar).

➪ In the **Add slab** dialog box (see Fig.1.7.1), define the following parameters:

- material – RC B30 AIII AIII;
- thickness b = 0.2 m;
- load in dead load case  0.3 tf/m^2;
- load in live load case  0.4 tf/m^2;
- other parameters remain by default.

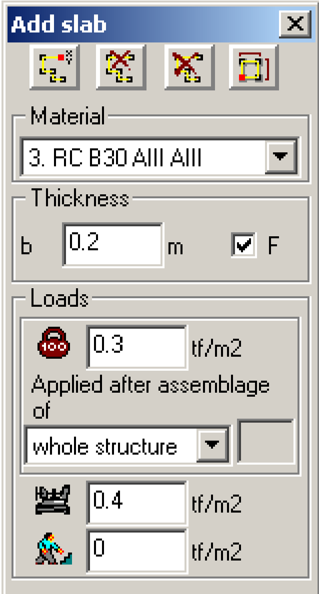
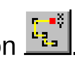
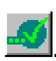


Figure 1.7.1 **Add slab** dialog box

Note that load uniformly distributed across whole slab is defined as parameter of slab.

- Specify with the pointer node of intersection between line **1** and line **A** according to plan of the structure.
- Specify with the pointer node of intersection between line **4** and line **A**.
- In the **Add slab** dialog box, click the **Specify coordinates of node** button .
- In another window (see Fig.1.7.2), define the following parameters:
  - click **Relative** (by default, **Absolute** option is selected);
  - specify x = 0 m;
  - y = -0.4 m;
  - click **Apply** .

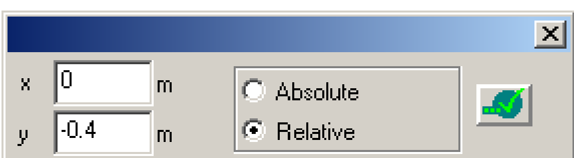


Figure 1.7.2 Floating box to define coordinates of node

The node located outside the grid nodes will be specified.

- Specify with the pointer node of intersection between line **I** and external line which coincides with the circle according to the plan of a structure.
- Specify with the pointer node of intersection between line **II** and external line which coincides with the circle.
- On the MODEL menu, point to **Coordinate system** and click **Fix**.
- On the model, specify node of intersection between lines 1a and A1.
- On the model, specify node of intersection between lines 1a and D1.

Coordinate system will be located as presented in Figure 1.7.3.

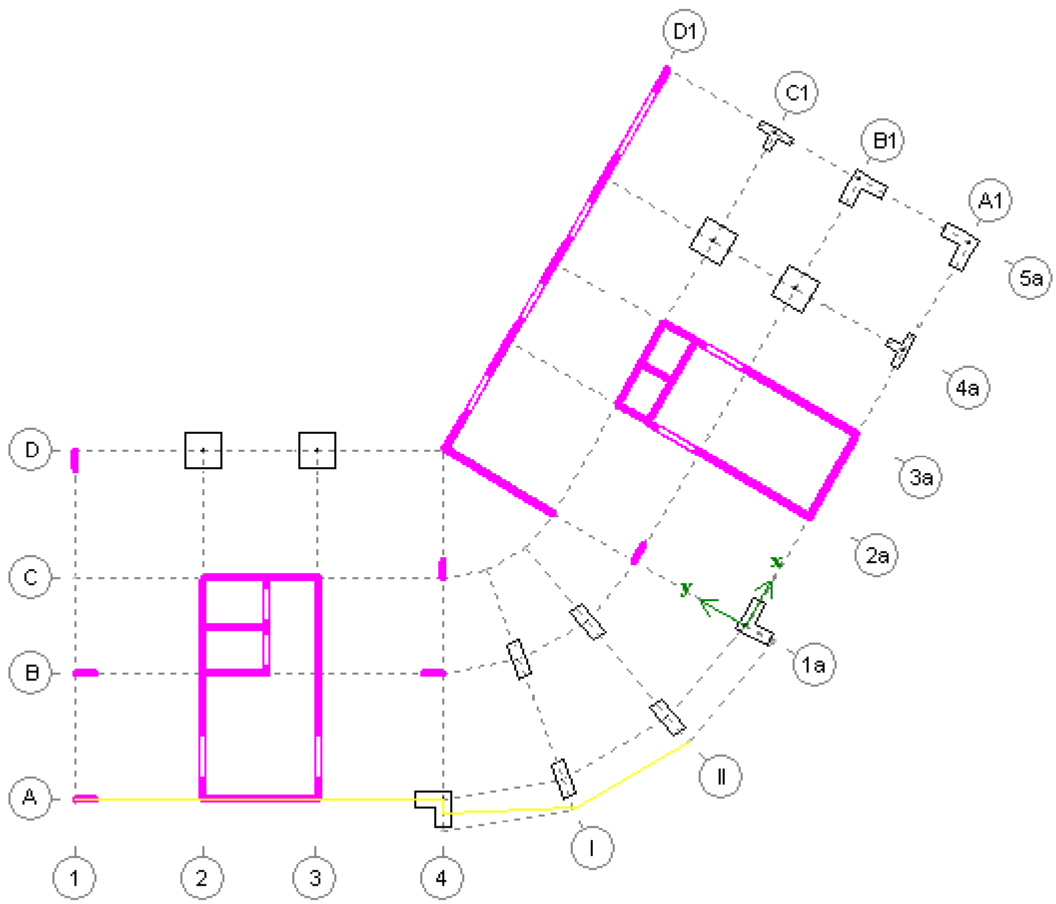


Figure 1.7.3 Fix coordinate system

- In the **Add slab** dialog box, click the **Specify coordinates of node** button .
- In another window, define the following parameters:
  - click **Absolute**;
  - specify x = 0 m;
  - y = -0.4 m;
  - click **Apply** .

The node located outside the grid nodes will be specified.

- Specify with the pointer node of intersection between line **1a** and line **A1** according to plan of the building.
- Specify with the pointer node of intersection between line **5a** and line **A1**.
- Specify with the pointer node of intersection between line **5a** and line **D1**.
- Specify with the pointer node of intersection between line **1a** and line **D1**.
- Specify with the pointer node of intersection between line **1** and line **D**.
- Specify with the pointer node of intersection between line **1** and line **A** (this is the first node that was specified) — contour of the slab will be closed.

- To restore initial location of coordinate system, on the MODEL menu, point to **Coordinate system** and click **Initial location**.

The specified slab will look as presented in Figure 1.7.4.

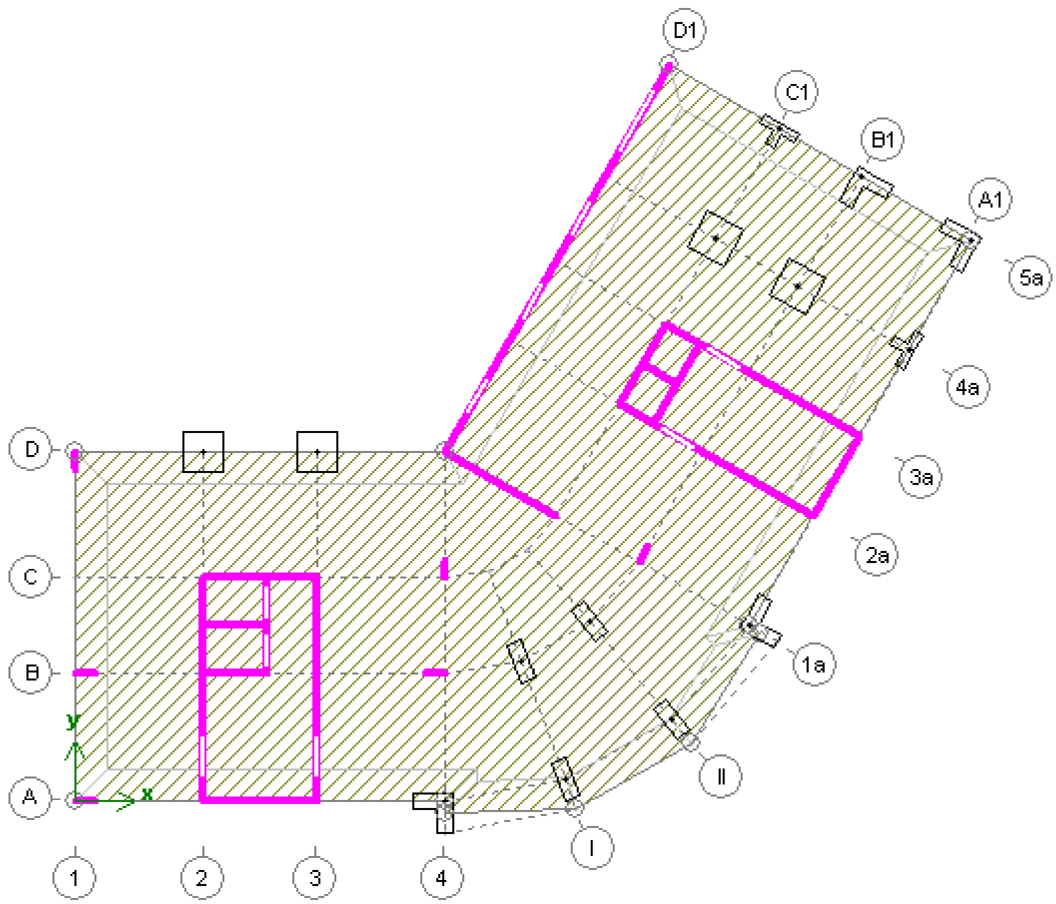


Figure 1.7.4 Floor slab

<u>To edit contour of floor slab – to add new nodes:</u>

- To edit contour of floor slab (add balcony), on the MODEL menu, point to **Edit** and click **Move or add contour node** (button on the toolbar).
- On the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin to intersection of lines 2 and D according to plan of the structure.

When you edit contour nodes (mode), the nodes are indicated with small circles.

- To add new node:
  - specify the contour side between two nodes (one of them — node of intersection between line 1 and D, the second — node of intersection between line 4 and D according to plan of the structure);
  - specify with the pointer node of intersection between line 2 and D.

New contour node will be added to the model.

➪ To add new node:

- specify the contour side between two nodes (one of them — node of intersection between line 2 and D, the second — node of intersection between line 4 and D according to plan of the structure);
- specify with the pointer node of intersection between line 3 and D.

New nodes of slab contour are presented in Figure 1.7.5.

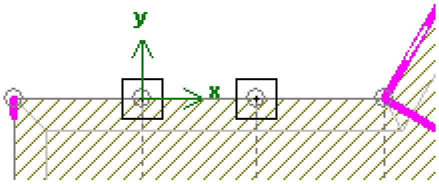
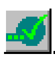


Figure 1.7.5 Floor slab (fragment)

➪ To add new node (two nodes that will modify the slab contour):

- specify the contour side between two nodes (one of them — node of intersection between line 2 and D, the second — node of intersection between line 3 and D according to plan of the structure);
- in the dialog box that was displayed when you activated the **Move or add contour node** command (see Fig.1.7.6), define the following parameters:
  - x = 0 m;
  - y = 1.2 m;
- click **Apply** .

New node located outside the grid nodes will be added.

Figure 1.7.6 Floating box to define coordinates of node

➪ To add new node:

- specify the contour side between two nodes (one of them — new node on line 2, the second — node of intersection between line 3 and D according to plan of the structure);
- in the dialog box that was displayed when you activated the **Move or add contour node** command, define the following parameters:
  - x = 3.6 m;
  - y = 1.2 m;
- click **Apply** .

New node located outside the grid nodes will be added. Modified slab contour is presented in Figure 1.7.7.

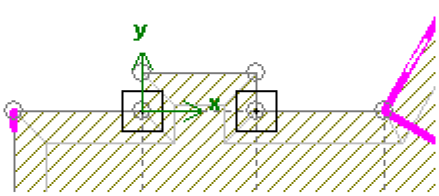


Figure 1.7.7 Floor slab (fragment)

To edit contour of floor slab – to add new nodes and move node:

- To edit contour of floor slab (add balcony), make sure that the mode **Move or add contour node** is still active.
- On the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin to intersection of lines 4a and D1 according to plan of the structure.
- To rotate coordinate system, on the MODEL menu, point to **Coordinate system** and click **Move and rotate**.
- In the **Move/rotate coordinate system** dialog box (see Fig.1.7.8), define the following parameters:
  - Fi = 60 degrees;
  - other parameters remain by default.
- Click **OK** and coordinate system will be rotated as necessary.

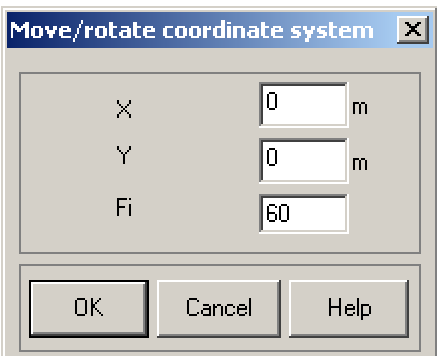


Figure 1.7.8 **Move/rotate coordinate system** dialog box

- To add new node:
  - specify the contour side between two nodes (one of them — node of intersection between line 1a and D1, the second — node of intersection between line 5a and D1 according to plan of the structure);
  - specify with the pointer node of intersection between line 4a and D1.

New contour node will be added to the model.

- To add new node:
  - specify the contour side between two nodes (one of them — node of intersection between line 4a and D1, the second — node of intersection between line 5a and D1 according to plan of the structure);
  - in the dialog box that was displayed when you activated the **Move or add contour node** command, define the following parameters:
    - x = 0 m;
    - y = 1.5 m;
  - click **Apply** .

New node located outside the grid nodes will be added.

- To move existing node:
  - specify with the pointer node of intersection between line 5a and D1 according to plan of the structure;
  - in the dialog box that was displayed when you activated the **Move or add contour node** command, define the following parameters:

- click **Relative** (by default, **Absolute** option is selected);
- specify x = 0 m;
- y = 0.8 m;

- click **Apply**.

Location of node is changed. Modified contour is presented in Figure 1.7.9.

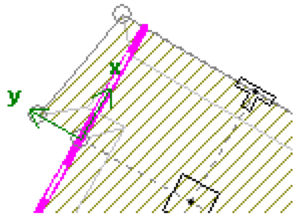


Figure 1.7.9 Floor slab (fragment)

➪ To restore initial location of coordinate system, on the MODEL menu, point to **Coordinate system** and click **Initial location**.

*If later on you would like to delete the node, on the MODEL menu, point to **Edit** and click **Delete contour node** (button on the toolbar). When this mode is active, specify the node that should be deleted.*

<u>To mark slabs on the model:</u>

➪ To display numbers and parameters of slabs on the model, on the VIEW menu, click **Display options**.

➪ In the **Display options** dialog box, do the following:

- click the **Numbers and parameters** tab ;
- select the **Slabs: Numbers and parameters** check box;
- click **Apply**.

➪ To close the **Display options** dialog box, click **Close** .

➪ To do the same from the toolbar, just click the **Slabs: Numbers and parameters** button on VISUALIZATION toolbar.

Storey No. and slab No. are displayed for every slab. In the lower row there are: slab thickness, fixation and number of material. If uniformly distributed load is defined across the whole slab, then in the third row there is its value for the current load case.

➪ Click the **Slabs: Numbers and parameters** button once again to remove visualization of numbers and parameters of slabs (the button should become inactive).

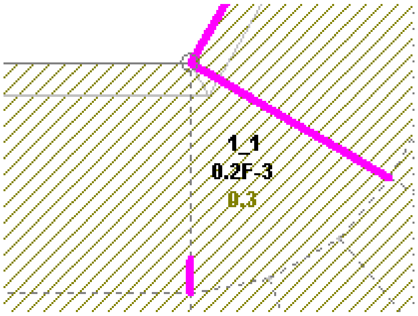


Figure 1.7.10 Notation for slabs on the model

Step 8. Defining openings in slabs

<u>To define openings in slab:</u>

- To define parameters and contour of floor slab, on the MODEL menu, point to **Add** and click **Add opening in slab** (button on the toolbar).
- Specify with the pointer node of intersection between line **2** and line **B** according to plan of the building.
- Specify with the pointer node of intersection between line **2** and line **C**.
- Specify on the model node of intersection between walls on line C.
- Specify on the model node of intersection between walls on line B.

Note that apart from grid nodes now it is possible to specify beginning and end nodes of walls, nodes of wall intersection and nodes of slab contours. Later on, number of nodes that may be specified with a pointer will be increased as new nodes will appear on the model.

- Specify with the pointer node of intersection between line **2** and line **B** (this is the first node that was specified) — contour of the slab will be closed.

The specified opening will look as presented in Figure 1.8.1.

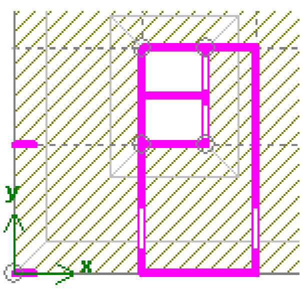


Figure 1.8.1 Floor slab with opening (fragment)

- To specify one more opening in slab, make sure that the mode **Add opening in slab** is still active.
- Specify with the pointer node of intersection between line **2a** and line **C1** according to plan of the building.
- Specify with the pointer node of intersection between line **3a** and line **C1**.
- Specify on the model node of intersection between walls on line **3a**.
- Specify on the model node of intersection between walls on line **2a**.

- Specify with the pointer node of intersection between line **2a** and line **C1** (this is the first node that was specified) — contour of the slab will be closed.

Figure 1.8.2 **Add opening** dialog box

One more opening will appear on the model.

<u>To define rectangular opening in slab by coordinates:</u>

- To specify rectangular opening in slab, make sure that the mode **Add opening in slab** is still active.
- On the MODEL menu, point to **Coordinate system** and click **Fix**.
- Specify with the pointer node of intersection between lines **2a** and **A1**.
- Specify with the pointer node of intersection between lines **2a** and **D1**.

The y-axis of coordinate system should coincide with line **2a**.

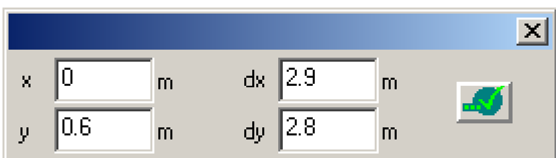
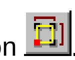
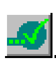


Figure 1.8.3 Floating box to define coordinates of node

- In the **Add opening** dialog box (see Fig.1.8.2), click the **Specify rectangular opening** button .
- In another window (see Fig.1.8.3), define the following parameters:
  - x = 0 m;
  - y = 0.6 m;
  - dx = 2.9 m;
  - dy = 2.8 m;
  - click **Apply** .

Rectangular opening will be added on the model.

- To restore initial location of coordinate system, on the MODEL menu, point to **Coordinate system** and click **Initial location**.

Floor slab with the specified openings should look as presented in Figure 1.8.4.

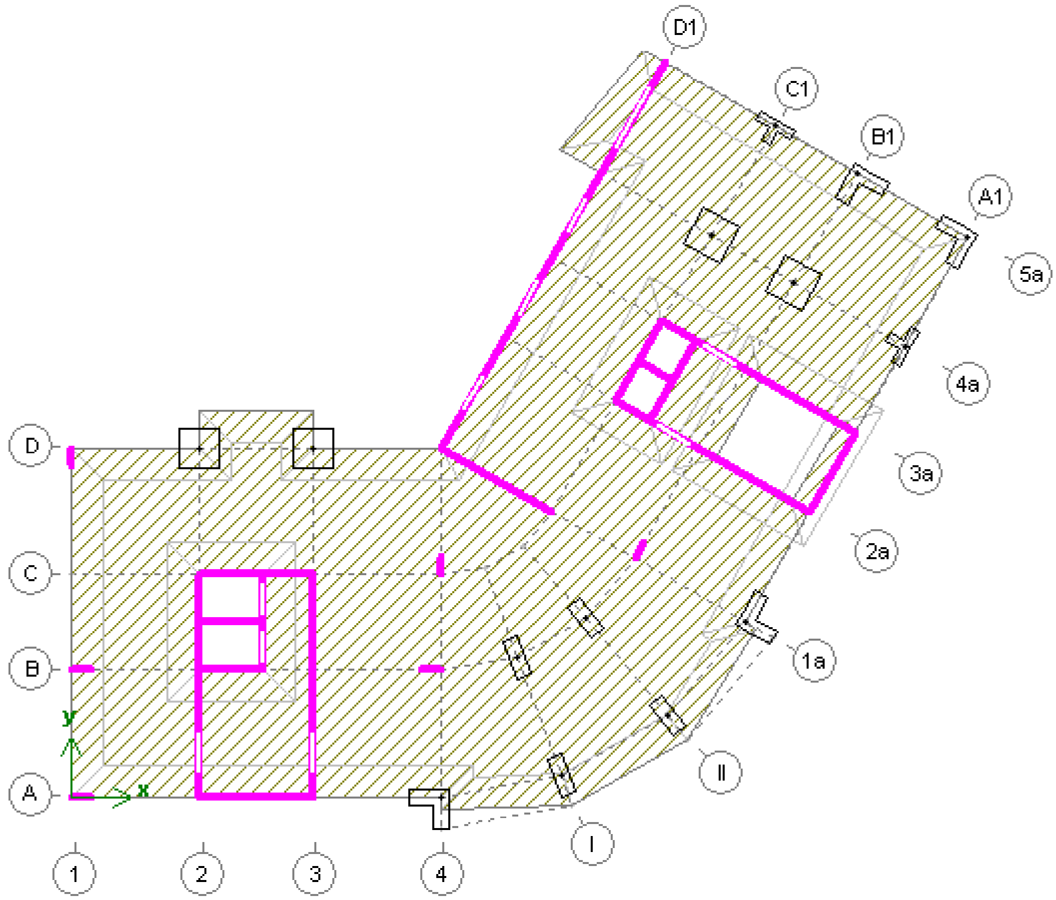


Figure 1.8.4 Floor slab of standard storey

Step 9. Defining beams

<u>To define multispan beam:</u>

- To define parameters and location of beam B1, on the MODEL menu, point to **Add** and click **Add beam** (button on the toolbar).
- In the **Add beam** dialog box (see Fig.1.9.1), define the following parameters:
  - material – RC B30 AIII AI;
  - thickness b = 0.4 m;
  - height h = 0.5 m;
  - other parameters remain by default.
- Specify with the pointer node of intersection between line **4a** and line **A1** according to plan of the building.
- Specify with the pointer node of intersection between line **4a** and line **D1**.

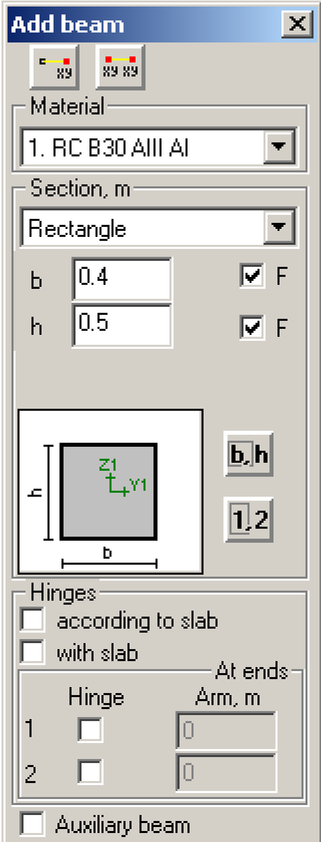


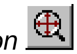
Figure 1.9.1 **Add beam** dialog box

Multispan beam will be added along the line 4a.

<u>To define auxiliary beams:</u>

Auxiliary beams are beams in slab that may have no support. In preliminary analysis such beams are simulated with load on slab and in FE model they are included as bar elements.

- To specify parameters and location of beam B2, make sure that the mode **Add beam** is still active.
- In the **Add beam** dialog box, define the following parameters:
  - thickness b = 0.2 m;
  - height h = 0.3 m;
  - select the **Auxiliary beam** check box;
  - other parameters remain by default.
- Specify with the pointer node of intersection between line **4** and slab contour near external line which coincides with the circle according to the plan of a structure.
- Specify with the pointer node of intersection between line **I** and external line which coincides with the circle.

*When you specify closely spaced nodes it is possible to zoom the image. To do this, on the VIEW menu, point to **Image** and click **Zoom to selection** (button*  *on the toolbar). To restore complete* *image on the screen, on the VIEW menu, point to **Image** and click **Fit in window** (button*  *on the toolbar).*

Auxiliary beam will be added at the edge of a slab.

- Define location of beam B2.
- Specify with the pointer node of intersection between line **I** and external line which coincides with the circle.
- Specify with the pointer node of intersection between line **II** and external line which coincides with the circle.
- Define location of the second beam B2.
- Specify with the pointer node of intersection between line **II** and external line which coincides with the circle.
- Specify with the pointer node of intersection between line **1a** and slab contour near external line which coincides with the circle.

The specified beams should look as presented in Figure 1.9.2.

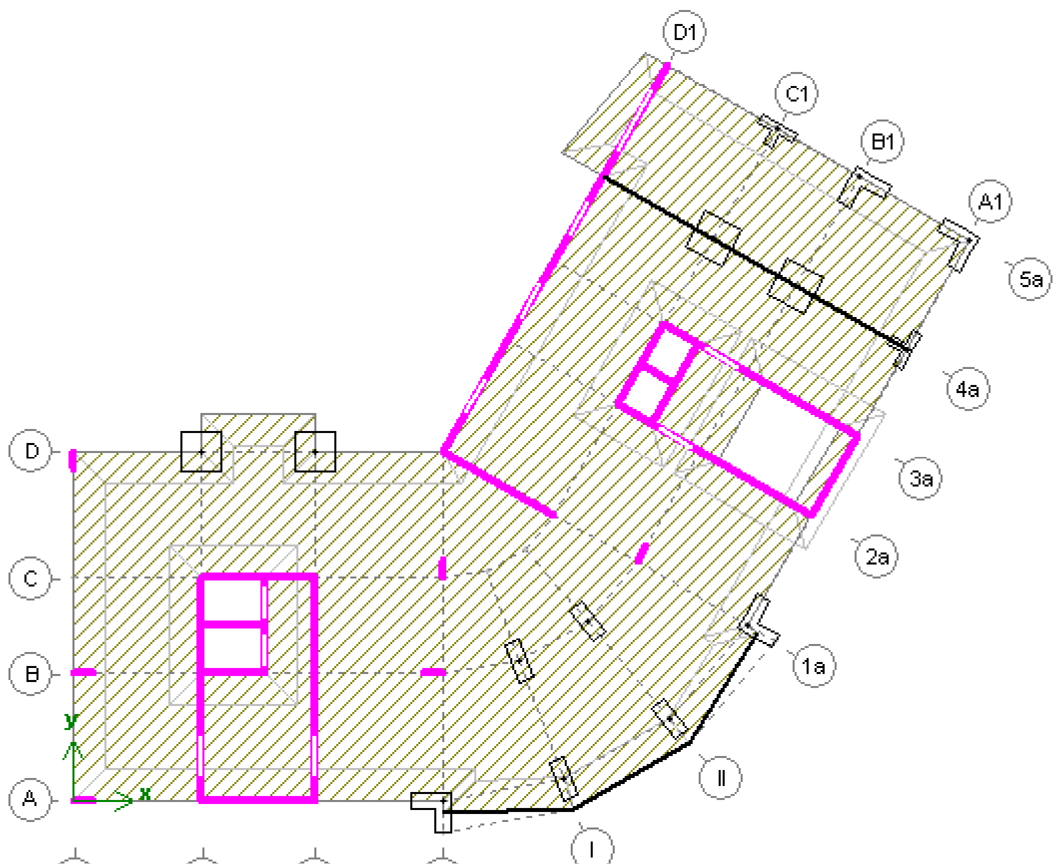


Figure 1.9.2 Beams of standard storey

<u>To mark beams on the model:</u>

- To display numbers and parameters of beams on the model, on the VIEW menu, click **Display options**.

- In the **Display options** dialog box, do the following:
  - click the **Numbers and parameters** tab ;
  - select the **Beams: Numbers and parameters** check box;
  - click **Apply** .
- To close the **Display options** dialog box, click **Close** .
- To do the same from the toolbar, just click the **Beams: Numbers and parameters** button on VISUALIZATION toolbar.

Storey No. and beam No. are displayed for every beam. In the lower row there are: section shape, dimensions, fixation and number of material. After analysis of a storey, multispan beam is divided into spans. Number of span is presented after beam number.

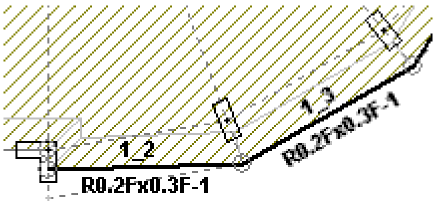


Figure 1.9.3 Notation for beams on the model

- Click the **Beams: Numbers and parameters** button once again to remove visualization of numbers and parameters of beams (the button should become inactive).

Step 10. Defining loads on slabs

<u>To take account of dead weight of elements:</u>

It is not necessary to define dead weight of elements as external load because it is automatically considered in analysis with the specified unit weight of materials. Load from dead weight of elements is included in dead load case.

<u>To define surface load:</u>

- Make sure that dead load case is selected as current one (**Dead load case** button on the toolbar should be active).
- To define dead uniformly distributed surface load, on the MODEL menu, point to **Add** and click **Surface load** (button on the toolbar).
- In the **Add surface load** dialog box (see Fig.1.10.1), define the following parameter:
  - value p = 0.16 tf/m2.

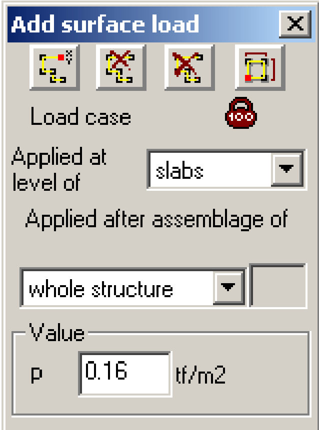


Figure 1.10.1 **Add surface load** dialog box

Note that value of uniformly distributed load is partially defined within the load distributed across the whole slab. That's why value of the surface load is defined as 0.46-0.3=0.16 tf/m2.

- Specify with the pointer node of intersection between line **2** and line **A** according to plan of the building.
- Specify with the pointer node of intersection between line **2** and line **B**.
- Specify on the model node of intersection between walls on line B.
- Specify on the model node of intersection between walls on line C.
- Specify with the pointer node of intersection between line **3** and line **C**.
- Specify with the pointer node of intersection between line **3** and line **A**.
- Specify with the pointer node of intersection between line **2** and line **A** (this is the first node that was specified) — contour of the surface load will be closed.

The surface load will look as presented in Figure 1.10.2.

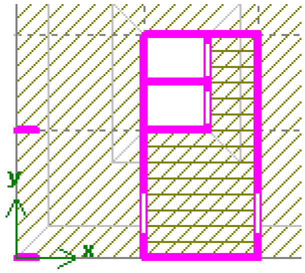


Figure 1.10.2 Floor slab with load (fragment)

- To define dead uniformly distributed surface load with the same load value on the area between lines 2a, 3a, B1, C1, make sure that the mode **Add surface load** is still active.
- Specify on the model node of intersection between walls on line 2a.
- Specify on the model node of intersection between walls on line 3a.
- Specify on the model node where the opening joins the wall on line 3a.
- Specify on the model node where the opening joins the wall on line 2a.
- Specify on the model node of intersection between walls on line 2a (this is the first node that was specified) — contour of the surface load will be closed.

The surface load will look as presented in Figure 1.10.3.

Figure 1.10.3 Notation for loads on the model

To mark loads on the model:

- To display numbers and parameters of beams on the model, on the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - click the **Numbers and parameters** tab ;
  - select the **Load values** check box;
  - click **Apply** .
- To close the **Display options** dialog box, click **Close** .
- To do the same from the toolbar, just click the **Load values** button on VISUALIZATION toolbar.

For every load you will see its value for the load case that is selected current at the moment.

To define linear loads:

- To define dead linearly distributed load, on the MODEL menu, point to **Add** and click **Linear load** (button on the toolbar).
- In the **Add linear load** dialog box (see Fig.1.10.4), define the following parameter:
  - value p = 0.22 tf/m.
- Specify with the pointer node of intersection between line **1** and line **A** according to plan of the building.
- Specify on the model the nearest (bottom) node of short wall that is located at the intersection of line **1** and **D**.

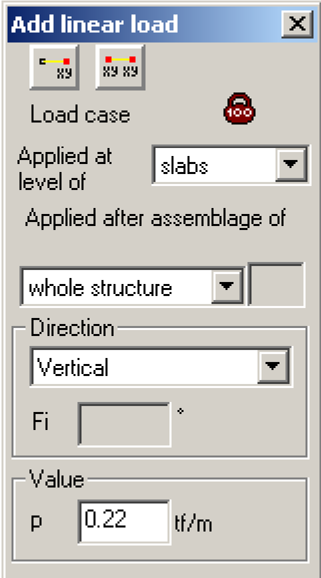
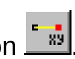
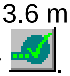


Figure 1.10.4 **Add linear load** dialog box

Linear load along the line 1 will be added.

- To define dead linearly distributed load with the same load value along lines A, D, A1, 5a and line that coincides with circle, make sure that the mode **Add linear load** is still active.
- Specify on the model right node of short wall that is located at the intersection of line 1 and A.
- Specify with the pointer node of intersection between line **2** and line **A**.
- Specify with the pointer node of intersection between line **3** and line **A**.
- In the **Add linear load** dialog box, click the **Specify coordinates of node** button .
- In another window, define the following parameters:
  - click **Relative** (by default, **Absolute** option is selected);
  - specify x = 3.6 m;
  - click **Apply** .

This linear load was defined up to the column edge. As columns are presented schematically on plan (not to scale) and you could see it when you change the image scale, then coordinates of the second node of load were defined numerically 4-(0.3+0.2/2)=3.6 m.

Two linear loads along the line **A** should look as presented in Figure 1.10.5.

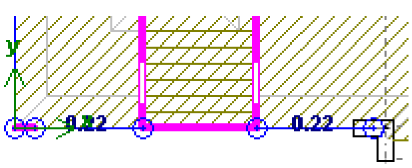


Figure 1.10.5 Floor slab with load (fragment)

- Specify with the pointer node of intersection between line **1** and line **D**.
- Specify with the pointer node of intersection between line **4** and line **D**.

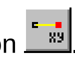
Linear load along the line **D** will be added.

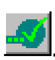
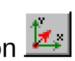
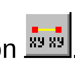
- Specify on the model the first node of auxiliary beam No.1_2.
- Specify on the model the second node of auxiliary beam No.1_2.
- Specify on the model the first node of auxiliary beam No.1_3.
- Specify on the model the second node of auxiliary beam No.1_3.
- Specify on the model the first node of auxiliary beam No.1_4.
- Specify on the model the second node of auxiliary beam No.1_4.

Three linear loads along the line that coincides with circle will be added.

- On the MODEL menu, point to **Coordinate system** and click **Fix**.
- Specify with the pointer node of intersection between lines **1a** and **A1**.
- Specify with the pointer node of intersection between lines **1a** and **D1**.

The y-axis of coordinate system should coincide with the line **1a**.

- In the **Add linear load** dialog box, click the **Specify coordinates of node** button .
- In another window, define the following parameters:
  - x = 0.4 m (by default, **Absolute** option is selected);
  - click **Apply** .

- Specify with the pointer node of intersection between line **2a** and line **A1**.
- Specify with the pointer node of intersection between line **3a** and line **A1**.
- In the open dialog box for coordinates of node, define the following parameters:
  - click **Relative** (by default, **Absolute** option is selected);
  - x = 2.7 m;
  - click **Apply** .
- On the MODEL menu, point to **Coordinate system** and click **Move** (button  on the toolbar). Then move the origin to intersection of lines 4a and A1 according to plan of the structure.
- In the **Add linear load** dialog box, click the **Specify coordinates of two nodes** button .
- In another window, define the following parameters:
  - x1 = 0.4 m;
  - y1 = 0 m;
  - x2 = 3.6 m;
  - y2 = 0 m;
  - click **Apply** .

Three linear loads along line A1 will be added.

- On the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin to intersection of lines 5a and A1 according to plan of the structure.
- In the open dialog box for coordinates of two nodes, define the following parameters:
  - x1 = 0 m;
  - y1 = 0.4 m;
  - x2 = 0 m;
  - y2 = 3.6 m;
  - click **Apply** .
- On the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin to intersection of lines 5a and B1 according to plan of the structure.
- In the open dialog box for coordinates of two nodes, define the following parameters:
  - x1 = 0 m;
  - y1 = 0.1 m;
  - x2 = 0 m;
  - y2 = 2.6 m;
  - click **Apply** .
- On the MODEL menu, point to **Coordinate system** and click **Move** (button on the toolbar). Then move the origin to intersection of lines 5a and C1 according to plan of the structure.
- In the open dialog box for coordinates of two nodes, define the following parameters:
  - x1 = 0 м;
  - y1 = 0.4 м;
  - x2 = 0 м;
  - y2 = 4 м;
  - click **Apply** .

Three linear loads along the line 5a will be added.

*At this moment all linear loads defined on floor slab should have load value 0.22 tf/m. If there are loads with value 0.1 tf/m on the model (in case you accidentally did not change default value), then to select these loads, on the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar). To change the load value for 0.22 tf/m, on the MODEL menu, point to **Edit** and click **Element properties** (button on the toolbar).*

- To define dead linearly distributed load with load value 0.2 tf/m along the side of opening in slab, make sure that the mode **Add linear load** is still active.
- In the **Add linear load** dialog box, define the following parameter:
  - value p = 0.2 tf/m.
- Specify on the model node where the opening joins the wall on line 2a.
- Specify on the model node where the opening joins the wall on line 3a.

Linear load will be added along the side of opening in slab.

- ➪ To define dead linearly distributed load with load value 0.14 tf/m along balconies, make sure that the mode **Add linear load** is still active.
- ➪ In the **Add linear load** dialog box, define the following parameter:
  - value p = 0.14 tf/m.

- ➪ Specify with the pointer node of intersection between line **2** and line **D**.
- ➪ Specify on the model node of balcony on line 2.

- ➪ Specify on the model node of balcony on line 2.
- ➪ Specify on the model node of balcony on line 3.

- ➪ Specify on the model node of balcony on line 3.
- ➪ Specify with the pointer node of intersection between line **3** and line **D**.

- ➪ Specify with the pointer node of intersection between line **4a** and line **D1**.
- ➪ Specify on the model node of balcony on line 4a.

- ➪ Specify on the model node of balcony on line 4a.
- ➪ Specify on the model node of balcony on line 5a.

- ➪ Specify on the model node of balcony on line 5a.
- ➪ Specify with the pointer node of intersection between line **5a** and line **D1**.

Six linear loads along balconies will be added.

- ➪ To restore initial location of coordinate system, on the MODEL menu, point to **Coordinate system** and click **Initial location**.

Floor slab with the specified loads should look as presented in Figure 1.10.6.

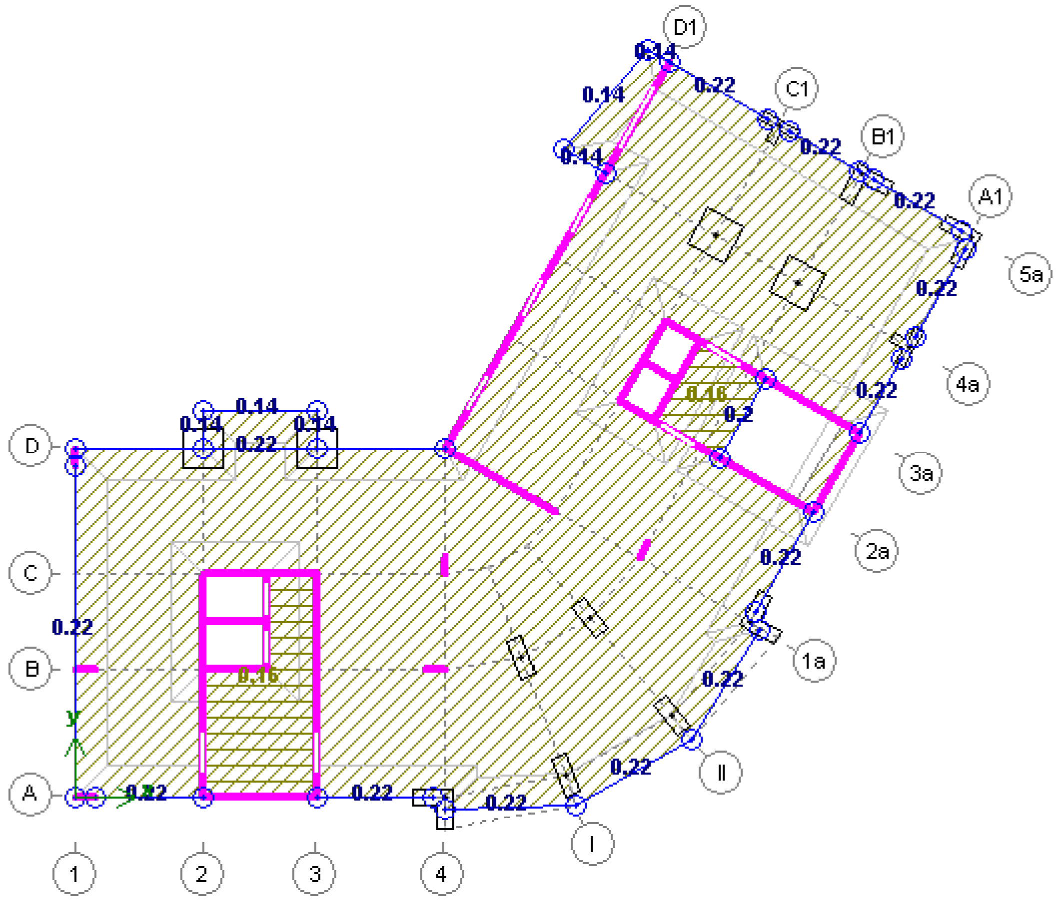


Figure 1.10.6 Floor slab of standard storey

<u>To present 3D view of the model:</u>

- To present 3D view of the model, on the VIEW menu, point to **3D View** and then click **Current storey**.
- To display numbers and parameters of slabs on the model, on the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - click the **Elements** tab , select the **Loads at slab level** check box;
  - click the **Numbers and parameters** tab ;
  - select the **Load values** check box;
  - click **Apply** .
- To close the **Display options** dialog box, click **Close** .
- To do the same from the toolbar, just click either **Loads at slab level** button or **Load values** button on VISUALIZATION toolbar.
- To return to the main view, on the VIEW menu, click **Main view** (button on the toolbar).

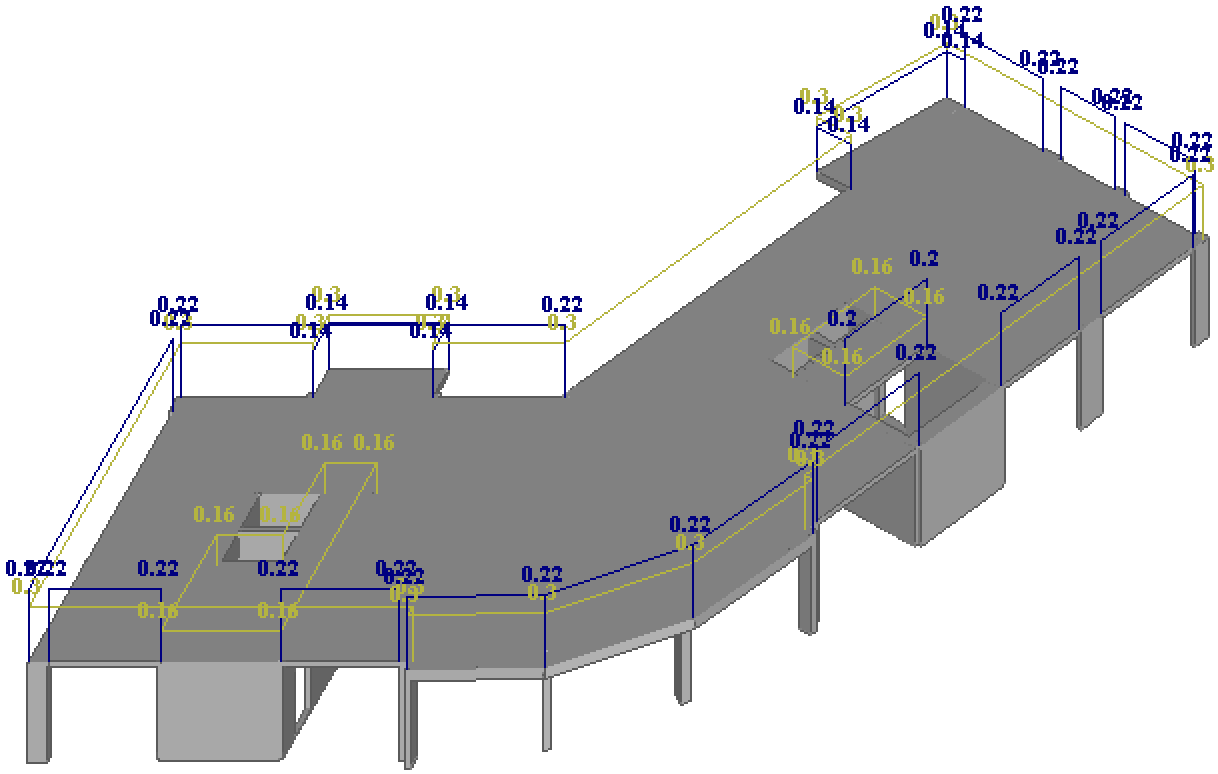


Figure 1.10.7 3D view of the specified model

<u>To define load factors:</u>

In BUILDING module it is accepted to define normative values of loads with load factor equal to 1 (Yf=1). Analysis results (loads, displacements, forces) are also presented with their normative values. Load factors and combination coefficients are defined according to selected building code. To define them, on the LOAD CASE menu, click ***Coefficients for load combinations*** *(see Fig.1.10.8). The specified factors and coefficients are considered in expert appraisal of reinforced concrete sections of elements and are exported to other modules where it is necessary to know design values of loads.*

When you check or determine sections of RC elements, calculate percentage of reinforcement, the set of load case combinations is automatically generated in the BUILDING module. This set is generated according to selected building code, to load cases available for the problem, to specified load factors, coefficients for load combinations and safety factor for purpose of structure. In this case, it is considered that earthquake and wind load cases are sign variable and the fact that pairs of earthquake and wind load cases directed opposite one another cannot be applied simultaneously.

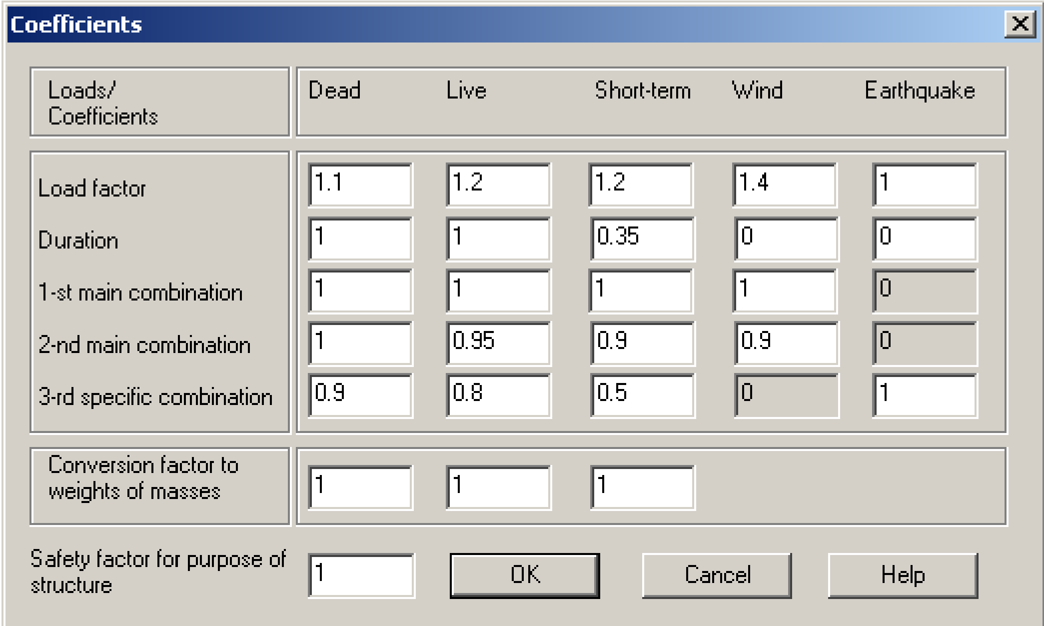
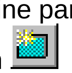


Figure 1.10.8 **Coefficients** dialog box

Step 11. Defining partitions

<u>To define partitions:</u>

Though partitions are defined as structural elements in the BUILDING module, but in both preliminary and FE analyses structural elements of partitions are simulated with the load on slab, not with element of the model. That's why, the partition is considered as a load and is applied onto the floor slab.

- To define parameters and location of partition, on the MODEL menu, point to **Add** and click **Partition** (button  on the toolbar).
- In the **Add partition** dialog box (see Fig.1.11.1), define the following parameters:
  - material 4. Brick 125-100
  - thickness b = 0.12 m;
  - other parameters remain by default.

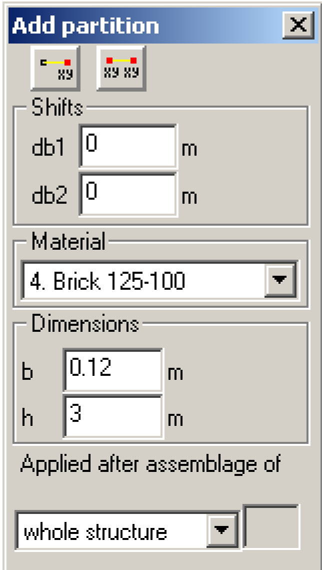


Figure 1.11.1 **Add partition** dialog box

Unlike height of columns and walls that is automatically taken as equal to storey height, height of partition is defined (by default, its value is equal to 3 m). That's why it is possible to define partitions of any height but the user should check this value if the storey height will be edited later.

- Specify with the pointer node of intersection between line **4a** and line **B1** according to plan of the building.
- Specify with the pointer node of intersection between line **5a** and line **B1**.

- To define one more partition, make sure that the mode **Add partition** is still active.
- Specify with the pointer node of intersection between line **4a** and line **C1**.
- Specify with the pointer node of intersection between line **5a** and line **C1**.

Two partitions will be added along lines B1 and C1.

<u>To define openings in partitions from the database of openings:</u>

- To define an opening D2 in partition along line B1, on the on the MODEL menu, point to **Edit** and click **Opening in wall (partition)** (button on the toolbar).
- When this mode is active, specify the partition on the model.
- In the **Openings in walls** dialog box, do the following:

Notice orientation of the partition specified on the model. The 1st and the 2nd nodes of partition are presented in the ***Openings in walls*** *dialog box at the small image to the left. They represent the order in which nodes were specified when you defined the partition and determine location of local coordinate system of the partition. When you define an opening, its coordinates should be specified in local coordinate system of the partition.*

  - under **Add opening**, click **From database**;

- in the **Add opening from database** dialog box, do the following:
  - select **Door D2** from the list;
  - specify X = 0.8 m.
- click **OK** and the opening will be added to schematic presentation of partition in the **Openings in walls** dialog box;

➪ Click **OK**.

➪ To define an opening D2 in partition along line C1, make sure that the mode **Add opening in wall (partition)** is still active. Then specify the partition on the model.

➪ In the **Openings in walls** dialog box, do the following:

- under **Add opening**, click **From database**;
- in the **Add opening from database** dialog box, do the following:
  - select **Door D2** from the list;
  - specify X = 0.8 m.
- click **OK** and the opening will be added to schematic presentation of partition in the **Openings in walls** dialog box;

➪ Click **OK**.

Openings will be added in both partitions.

To mark partitions on the model:

➪ To display numbers and parameters of walls on the model, on the VIEW menu, click **Display options**.

➪ In the **Display options** dialog box, do the following:

- click the **Numbers and parameters** tab ;
- select the **Partitions: Numbers and parameters** check box;
- click **Apply** .

➪ To close the **Display options** dialog box, click **Close** .

➪ To do the same from the toolbar, just click the **Partitions: Numbers and parameters** button on VISUALIZATION toolbar.

Storey No. and partition No. are displayed for every partition. In the lower row there are thickness and height of partition as well as number of material.

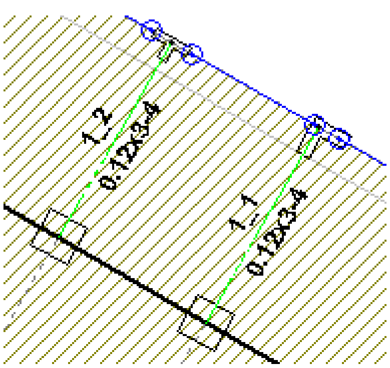


Figure 1.11.2 Notation for partitions on the model

- To hide numbers and parameters of partitions, click the **Partitions: Numbers and parameters** button once again (the button should be inactive).

<u>To present 3D view of the model:</u>

- To present 3D view of the model, on the VIEW menu, point to **3D View** and then click **Current storey**.

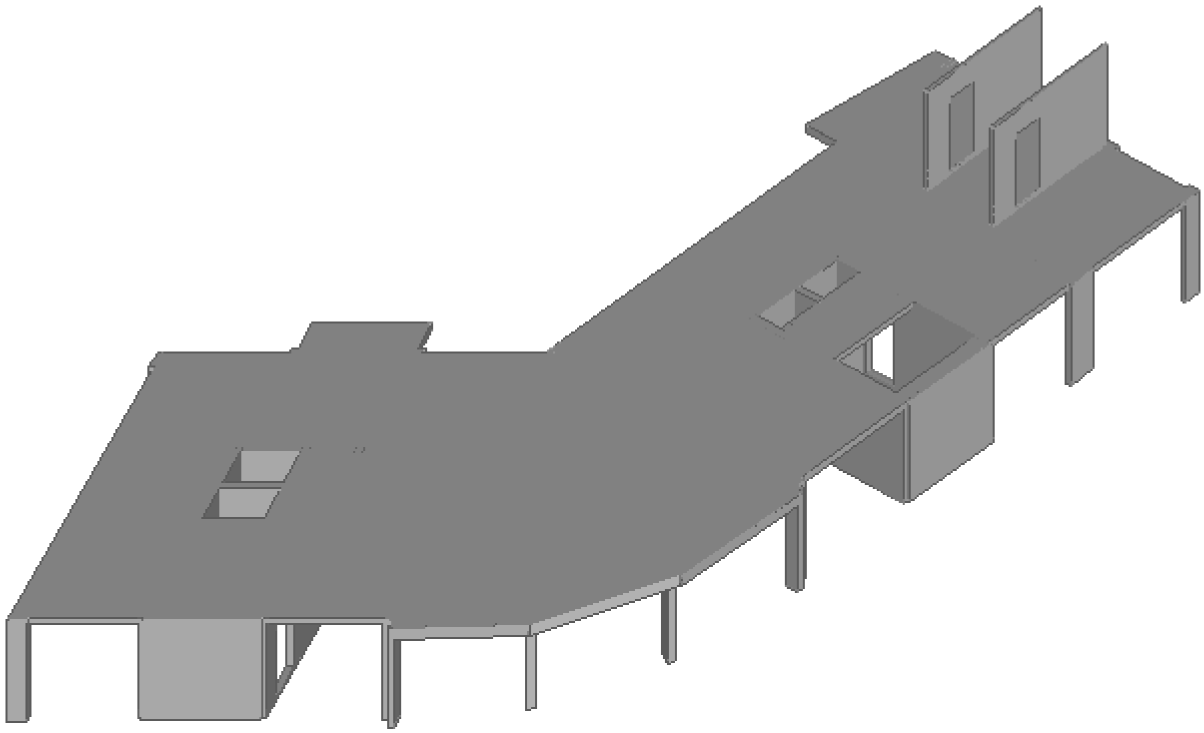


Figure 1.11.3 3D view of the specified model

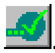
- To return to the main view, on the VIEW menu, click **Main view** (button on the toolbar).

Step 12. Analysis of storey

*There are two types of analysis in BUILDING module – preliminary (simplified) analysis and Finite Element Analysis (FEA). In preliminary (simplified) analysis you generate the model of the structure, gather loads on it, determine and check sections of reinforced concrete (RC) elements. Preliminary analysis consists of several analyses. To perform preliminary analysis, on the ANALYSIS menu, use **Analysis of current storey** and **Analysis of whole structure** commands. Diagnostic procedure is performed during analysis of the generated model. The errors are displayed in the box. When you click a row with description of an error, the element in which this error occurred is coloured red on the model.*

- To check the specified model and analyse the current storey, on the ANALYSIS menu, click **Analysis of current storey** (button on the toolbar).

- When analysis is complete, on the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - click the **Results of preliminary analysis** tab ;

- select the **For storey** check box;
- click **Apply** .

➪ To close the **Display options** dialog box, click **Close**  .

➪ To do the same from the toolbar, just click the **Analysis results for storey** button  on VISUALIZATION toolbar.

Analysis results should look as presented in Figure 1.12.1.

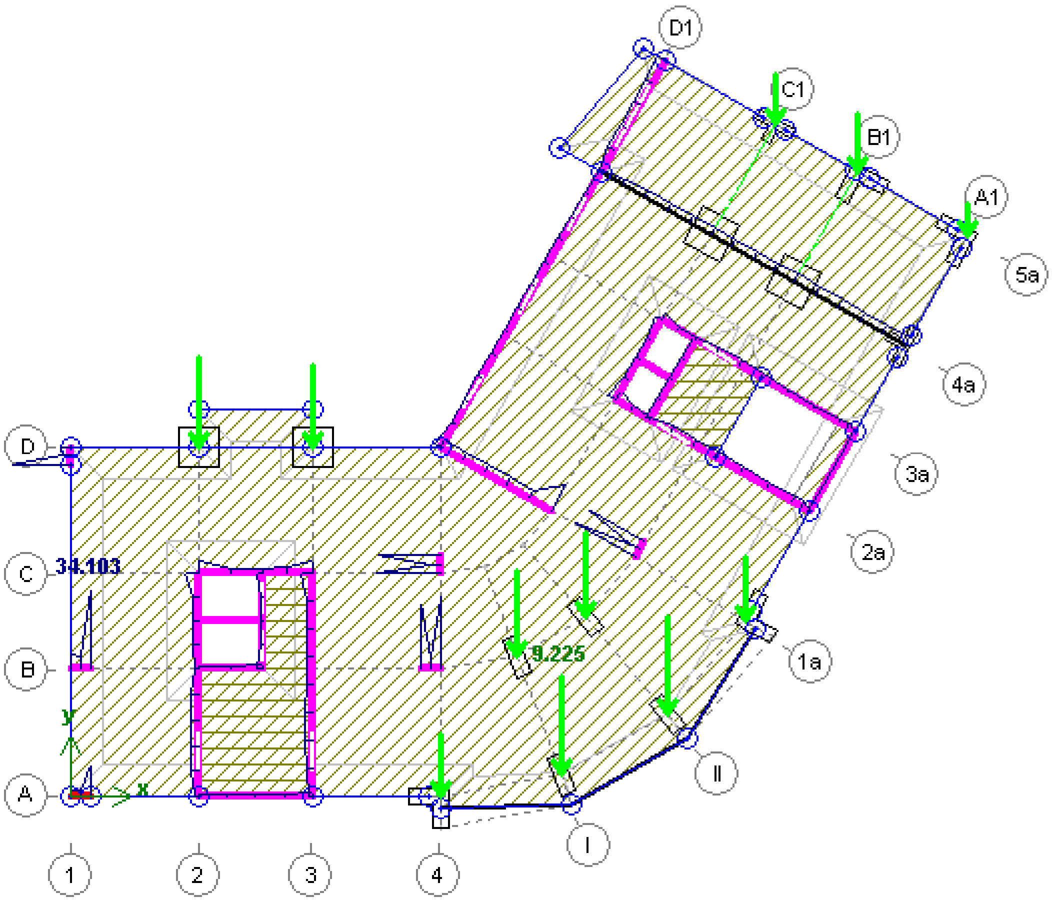
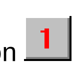


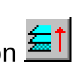
Figure 1.12.1 Floor slab of a standard storey with analysis results for a storey

Step 13. Making copy of the storey

<u>To copy the current storey:</u>

 *If several storeys have the same shape and load, then it is recommended to create one storey, analyse it to check the structural model and then copy it to other storeys.*

Make sure that the first storey is assigned as a current one - button  on the toolbar should be active.

➪ To copy the current storey, on the STOREYS menu, click **Copy storey** (button  on the toolbar).

➪ In the **Copy current storey** dialog box (see Fig.1.13.1), define the following parameters:

- from storey No. 2;

- to storey No. 13.

➪ Click **OK**.

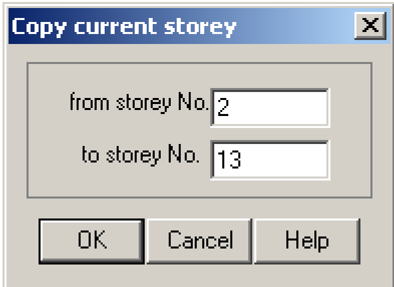


Figure 1.13.1 **Copy current storey** dialog box

The specified model will consists of the same 13 storeys.

Note that storey numbers on the toolbar beginning from number 2 are now coloured red. It means that there are specified elements on these storeys.

<u>To edit number of current storey:</u>

➪ To edit number of current storey, on the STOREYS menu, click **Current storey**.

➪ In the **Current storey** dialog box (see Fig.1.13.2), do the following:

- select **13** from the list;

➪ Click **OK**.

➪ To do the same from the toolbar, just click the storey number buttons  on the toolbar.

Storey No.13 will become a current one.

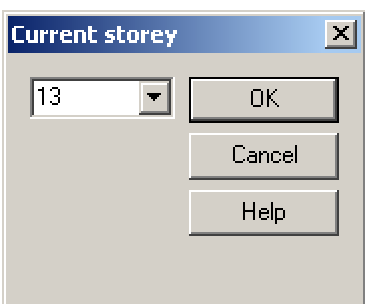
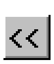


Figure 1.13.2 **Current storey** dialog box

➪ Click  and then  buttons on the toolbar.

Storey No.1 will be current again.

<u>To edit height of current storey:</u>

➪ To edit height of current storey No.1, on the STOREYS menu, click **Parameters of storey**.

➪ In the **Parameters of current storey** dialog box (see Fig.1.13.3), define the following parameters:

- name **Basement**;

- storey height 2.4 m;

➪ Click **OK**.

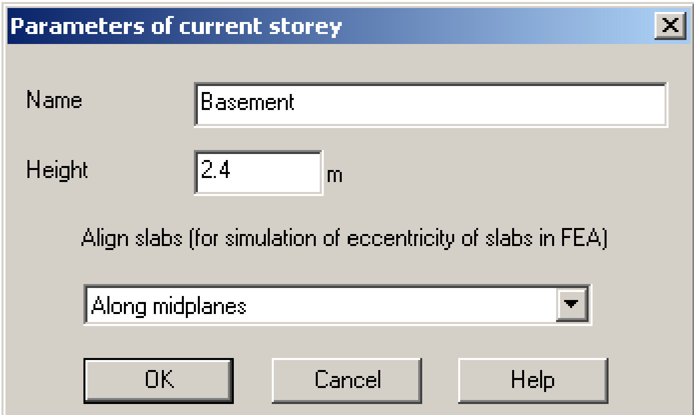


Figure 1.13.3 **Parameters of current storey** dialog box

Height of storey No.1 will be changed.

In BUILDING module, the storeys are always numbered beginning from No.1. It should be considered when you define a basement. Number of current storey, storey name, height and upper level of current storey are displayed in the upper left corner of the screen.

<u>To present 3D view of the model:</u>

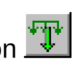
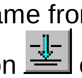
➪ To present 3D view of the model, on the VIEW menu, point to **3D View** and then click **Whole structure** (button  on the toolbar).

➪ To display numbers and parameters of slabs on the model, on the VIEW menu, click **Display options**.

➪ In the **Display options** dialog box, do the following:

- on the **Results of preliminary analysis** tab , select the **For storey** check box;
- click **Apply** .

➪ To close the **Display options** dialog box, click **Close**  .

➪ To do the same from the toolbar, just click either **Loads at slab level** button  or **Analysis results for storey** button  on VISUALIZATION toolbar.

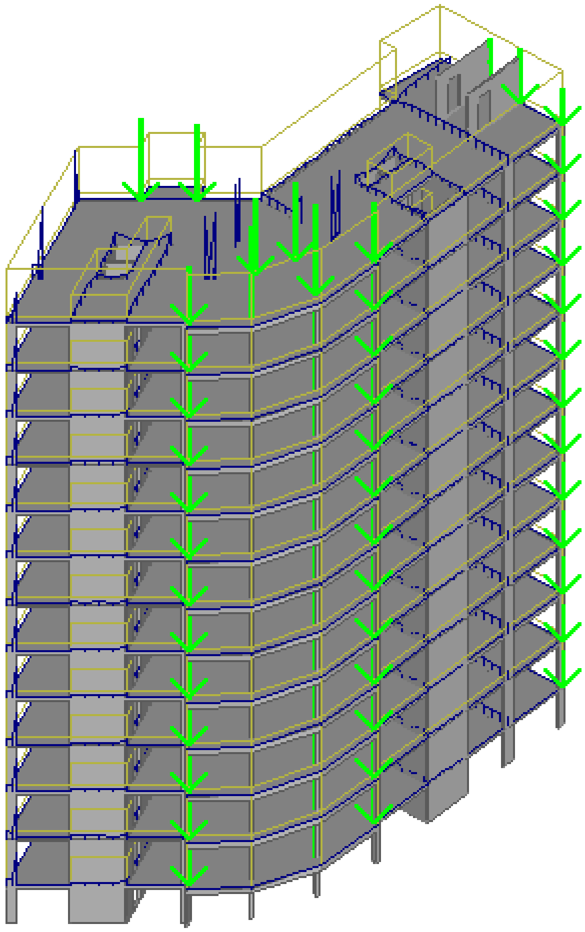


Figure 1.13.4 3D view of the model

➪ To return to the main view, on the VIEW menu, click **Main view** (button on the toolbar).

Step 14. Editing the storey

<u>To delete from the storey elements selected by criteria:</u>

➪ On storey No.1 (basement), delete all columns and walls in order to define new location of walls in the basement.

➪ To select all columns and walls, on the MODEL menu, point to **Select** and click **Select elements by criteria** (button on the toolbar).

➪ In the **Select elements by criteria** dialog box, do the following:

- on the **Columns** tab , all parameters should remain by default;
- click **Apply** ;
- under **Action**, click **Select** in order that previous selection remains on the model;
- select the **Walls** tab ;
- click **Apply** .

➪ To close **Select elements by criteria** dialog box, click the **Close** button .

Selected columns and walls should look as presented in figure 1.14.1.

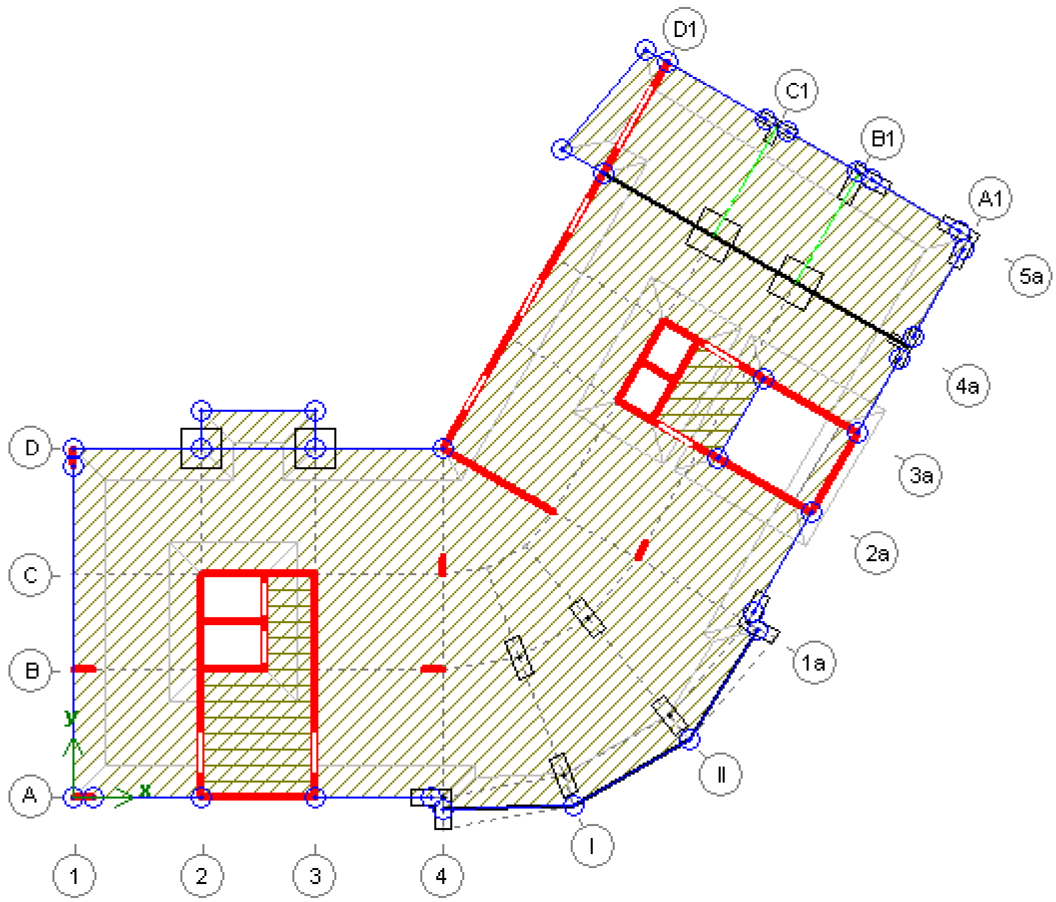
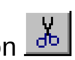


Figure 1.14.1 Selected elements on the model of current storey

➪ To delete selected elements, on the MODEL menu, click **Delete elements** (button  on the toolbar).

<u>To define walls for the basement:</u>

➪ To define parameters and location of external walls, on the MODEL menu, point to **Add** and click **Wall** (button  on the toolbar).

➪ In the **Add wall** dialog box, define the following parameters:

- material – RC B20 AI AI;
- thickness b = 0.24 m;
- other parameters remain by default.

➪ Specify the first node on the model — node of intersection between line **1** and line **A** according to plan of the building.

➪ Specify the first node on the model — node of intersection between line **4** and line **A**.

New wall will be added along line A.
In the same way specify all external walls of the basement.

➪ Specify with the pointer node of intersection between line **4** and line **A**, then — node of intersection between line **I** and line **A**.

- Specify with the pointer node of intersection between line **I** and line **A**, then — node of intersection between line **II** and line **A**.
- Specify with the pointer node of intersection between line **II** and line **A**, then — node of intersection between line **1a** and line **A1**.
- Specify with the pointer node of intersection between line **1a** and line **A1**, then — node of intersection between line **5a** and line **A1**.
- Specify with the pointer node of intersection between line **5a** and line **A1**, then — node of intersection between line **5a** and line **D1**.
- Specify with the pointer node of intersection between line **5a** and line **D1**, then — node of intersection between line **4** and line **D**.
- Specify with the pointer node of intersection between line **4** and line **D**, then — node of intersection between line **1** and line **D**.
- Specify with the pointer node of intersection between line **1** and line **D**, then — node of intersection between line **1** and line **A**.

The specified walls should look as presented in Figure 1.14.2.

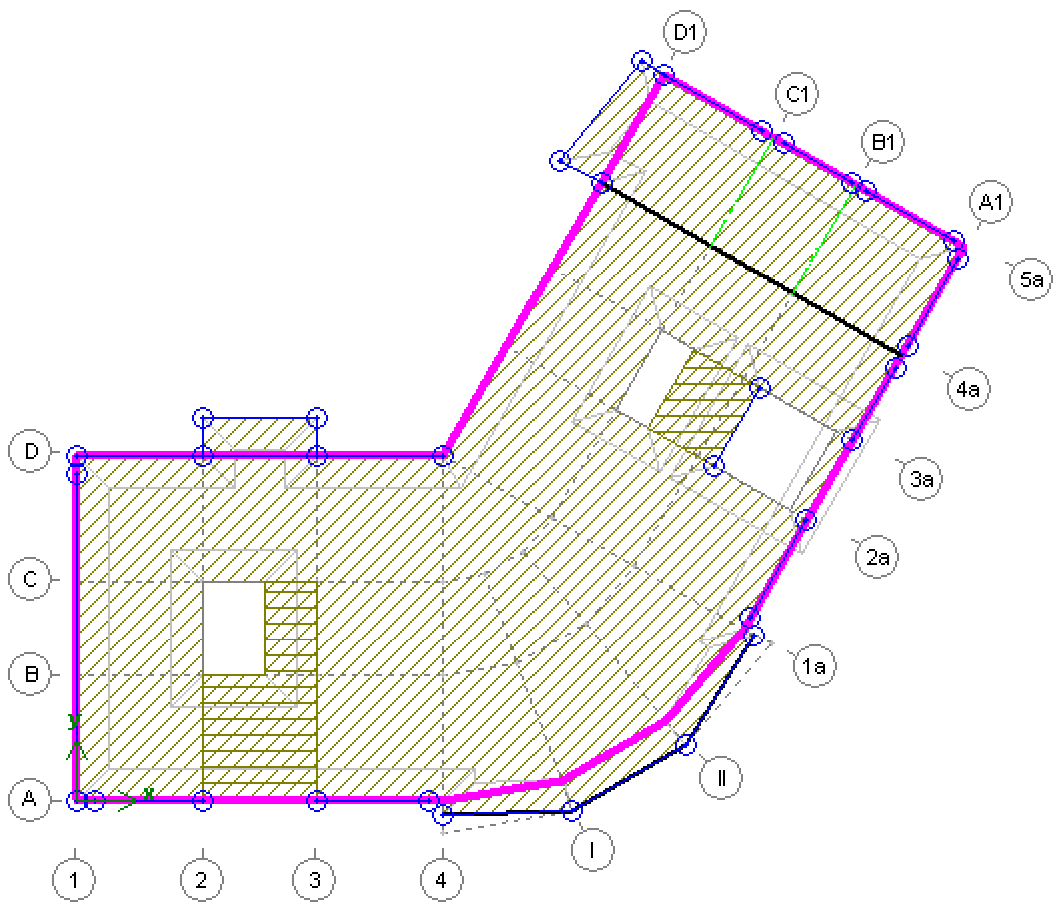


Figure 1.14.2 Specified walls on the model of current storey

- To visualize columns and walls of upper storey on the model, on the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - on the **Additional** tab , select the **Elements of upper storey** check box;

- click **Apply**.

➪ To close the **Display options** dialog box, click **Close** .

When you visualize columns and walls of upper or lower storey, there will be several additional nodes that may be specified with the mouse pointer when you define elements of current storey.

➪ To define parameters and location of internal walls of basement, make sure that the mode **Add wall** is still active.

➪ Specify with the pointer node of intersection between line **1** and line **B**, then — node of intersection between line **4** and line **B**.

➪ Specify with the pointer node of intersection between line **2** and line **A**, then — node of intersection between line **2** and line **C**.

➪ Specify with the pointer node of intersection between line **3** and line **A**, then — node of intersection between line **3** and line **C**.

➪ Specify with the pointer node of intersection between line **2** and line **C**, then — node of intersection between line **3** and line **C**.

➪ Specify on the model node where an opening joins the wall on line B, then — node where an opening joins the wall on line C.

➪ Specify on the model node where middle of an opening joins the wall on line 2, then — node where middle of an opening joins the wall parallel to line 3.

➪ To define walls with shifts, in the **Add wall** dialog box, define the following parameters:

- shift db1 = 0.4 m;
- shift db2 = 0.6 m;
- other parameters remain by default.

➪ Specify with the pointer node of intersection between line **4** and line **A**, then — node of intersection between line **4** and line **C**.

Note that beginning and end of the wall were shifted away from the specified nodes at the specified distances db1, db2.

➪ Define two more walls with the same shifts.

➪ Specify with the pointer node of intersection between line **I** and line **A**, then — node of intersection between line **I** and line **B**.

➪ Specify with the pointer node of intersection between line **II** and line **A**, then — node of intersection between line **II** and line **B**.

➪ In the **Add wall** dialog box, define the following parameters:

- shift db1 = 0.4 m;
- shift db2 = 0 m.

➪ Specify with the pointer node of intersection between line **1a** and line **A1**, then — node of intersection between line **1a** and line **B1**.

➪ In the **Add wall** dialog box, define the following parameters:

- shift db1 = 0 m;
- shift db2 = 0 m.

➪ Specify with the pointer node of intersection between line **1a** and line **C1**, then — node of intersection between line **1a** and line **D1**.

➪ Specify with the pointer node of intersection between line **1a** and line **B1**, then — node of intersection between line **5a** and line **B1**.

➪ Specify with the pointer node of intersection between line **1a** and line **C1**, then — node of intersection between line **5a** and line **C1**.

➪ Specify with the pointer node of intersection between line **2a** and line **A1**, then — node of intersection between line **2a** and line **C1**.

➪ Specify with the pointer node of intersection between line **3a** and line **A1**, then — node of intersection between line **3a** and line **C1**.

➪ Specify on the model node where an opening joins the wall on line 2a, then — node where an opening joins the wall on line 3a.

➪ Specify on the model node where middle of an opening joins the wall on line C1, then — node where middle of an opening joins the wall parallel to line 3a.

The specified walls should look as presented in Figure 1.14.3.

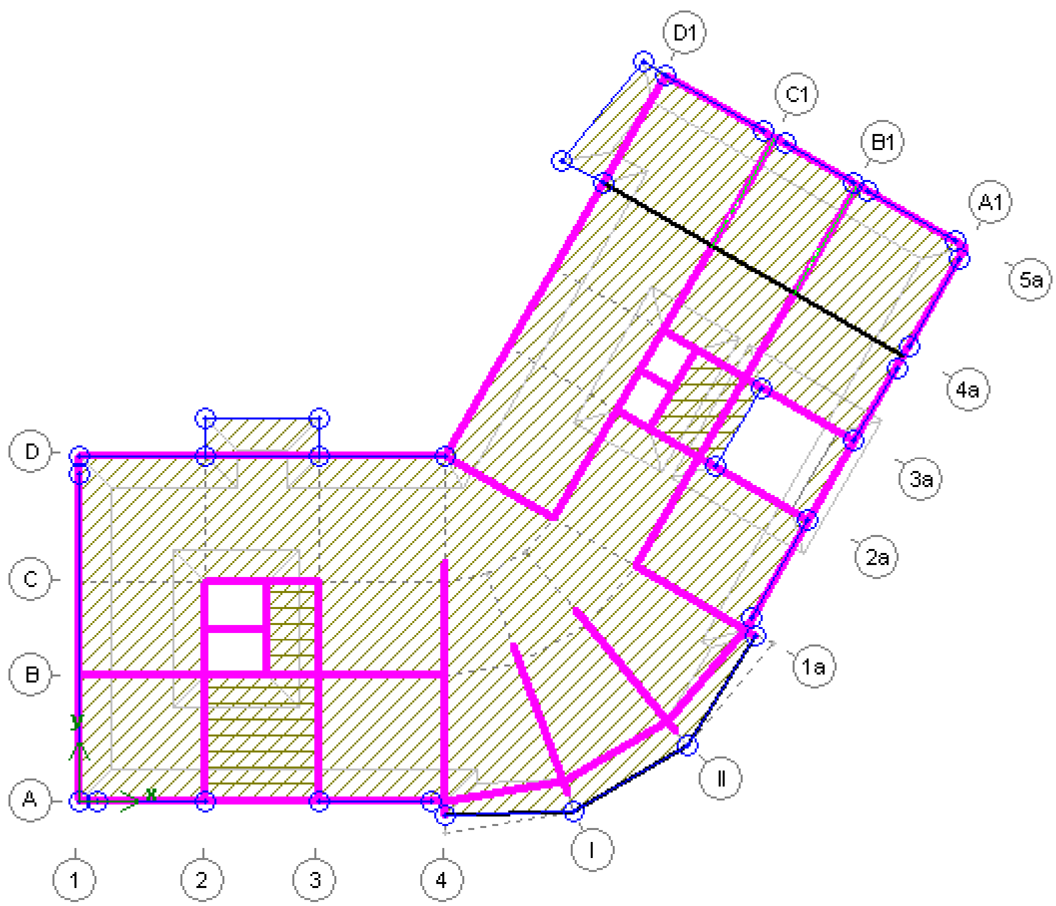


Figure 1.14.3 Specified walls on the model of current storey

*To check the changed model of the storey and analyse it, on the ANALYSIS menu, click **Analysis of current storey** (button on the toolbar).*

To delete from the storey loads selected by criteria:

- To change number of the current storey, on the STOREYS menu, click **Current storey**.
- In the **Current storey** dialog box, do the following:
  - select **13** from the list;
  - click **OK**.
- To do the same from the toolbar, click the and buttons on the toolbar.

Storey No.13 will become a current one.

- On the storey No.13 (the upper storey), delete all partitions and linear loads.
- To select all partitions and linear loads, on the MODEL menu, point to **Select** and click **Select elements by criteria** (button on the toolbar).
- In the **Select elements by criteria** dialog box, do the following:
  - select the **Partitions** tab ;
  - click **Apply** ;
  - select the **Linear loads** tab ;
  - under **Action**, click **Select** in order not to unselect partitions that were selected;
  - click **Apply** .
- To close the **Select elements by criteria** dialog box, click the **Close** button .
- To delete selected elements, on the MODEL menu, click **Delete elements** (button on the toolbar).

To delete from the storey openings and surface load:

- To select all surface loads and two openings between lines 2a and 3a on storey No.13, on the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).
- When this mode is active, specify on the model arbitrary point inside the contour of the first opening. Selected opening will be coloured red.
- Specify on the model arbitrary point inside the contour of the second opening.
- Specify on the model arbitrary point inside the contour of surface load.
- In the **Select element** dialog box (see Fig.1.14.4), make the selection more exact:
  - select **Surface load** from the list.
  - click **OK**.
- To delete selected elements, on the MODEL menu, click **Delete elements** (button on the toolbar).

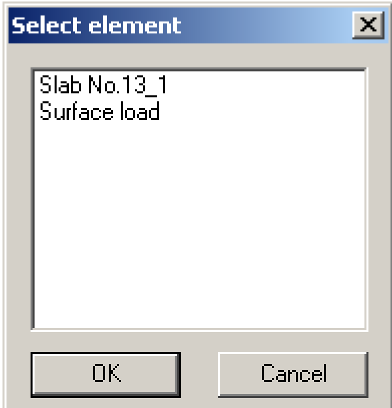


Figure 1.14.4 **Select element** dialog box

To edit load uniformly distributed across whole slab:

- To select the slab, make sure that the mode **Select elements** is still active.
- Specify the slab on the model. Selected slab will be coloured red.
- To edit load uniformly distributed across whole slab, on the MODEL menu, point to **Edit** and click **Element properties** (button on the toolbar).
- In the **Slab No.13_1** dialog box, define the following parameters:
  - load from dead load case 0.2 tf/m^2;
  - other parameters remain by default;
  - click **Apply** .

Value of uniformly distributed load will be modified.

- To turn off the **Element properties** mode, click the button on the toolbar once more.
- To turn off the **Select elements** mode, click the button on the toolbar once more.
- To unselect all elements, on the MODEL menu, point to **Edit** and click **Unselect** (button on the toolbar).

*To check the changed model of the storey and analyse it, on the ANALYSIS menu, click **Analysis of current storey** (button on the toolbar).*

To copy elements from one storey to another:

- To select group of walls between lines 2 and 3 from storey No.13 to storey No.14, on the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).
- To hide visualization of the specified loads and analysis results, click the **Loads** button (VISUALIZATION toolbar) and **Analysis results of storey** button.
- On the MODEL menu, point to **Select** and click **Group pointer** (button on the toolbar).
- Specify on the model rectangle that will include walls between lines 2 and 3. Group of walls will be coloured red.

- On the MODEL menu, point to **Select** and click **Single pointer** (button on the toolbar).
- Specify the opening located in the rectangle selected with group pointer in order to unselect it.

Selected walls should look as presented in Figure 1.14.5.

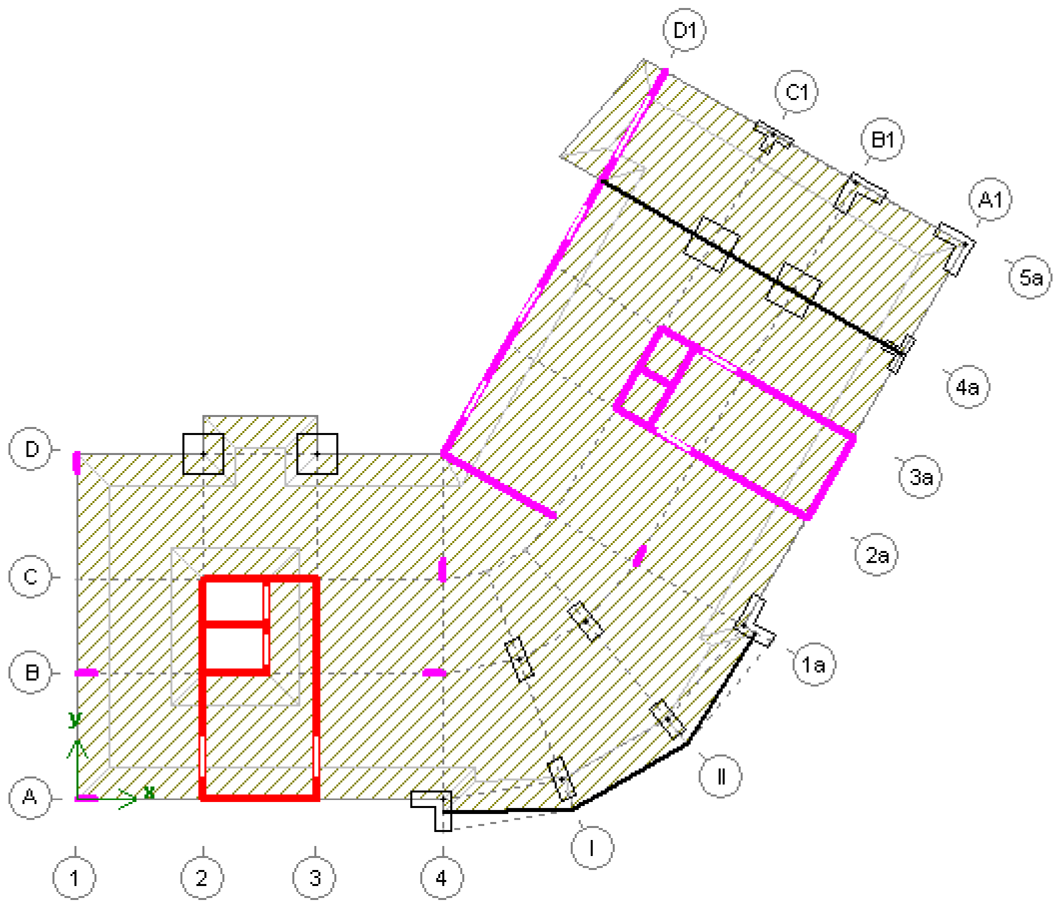


Figure 1.14.5 Selected elements on the model of current storey

- To copy selected wall from one storey to another, on the MODEL menu, point to **Edit** and click **Copy and move** (button on the toolbar).
- In the **Copy and move** dialog box, do the following:
  - select the **Copy to other storeys** tab (see Fig.1.14.6);
  - specify from storey No. 14;
  - to storey No. 14;
  - click **Apply** .

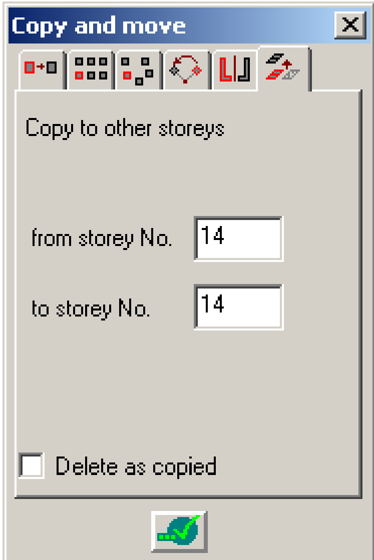


Figure 1.14.6 **Copy and move** dialog box (**Copy to other storeys** tab)

Selected walls will be copied to storey No.14.

- To change number of the current storey, on the STOREYS menu, click **Current storey**.
- In the **Current storey** dialog box, do the following:
  - select **14** from the list;
  - click **OK**.

To edit height of current storey:

- To edit height of the current storey No. 14, on the STOREYS menu, click **Parameters of storey**.
- In the **Parameters of current storey** dialog box, define the following parameters:
  - storey height 2.8 m;
  - click **OK**.

Height of storey No.14 will be modified.

To define contour of floor slab:

- To define parameters and contour of floor slab for storey No.14, on the MODEL menu, point to **Add** and click **Slab** (button on the toolbar).
- In the **Add slab** dialog box, define the following parameters:
  - material – RC B30 AIII AIII;
  - thickness b = 0.2 m;
  - load from dead load case 0.2 tf/m^2;
  - load from live load case 0.4 tf/m^2;
  - other parameters remain by default.
- Specify with the pointer the following nodes (one after another):
  - node of intersection between line **2** and line **A**;

- node of intersection between line **2** and line **C**;
- node of intersection between line **3** and line **C**;
- node of intersection between line **3** and line **A**;
- node of intersection between line **2** and line **A** (this is the first node that was specified) — contour of the slab will be closed.

The specified slab will look as presented in Figure 1.14.7.

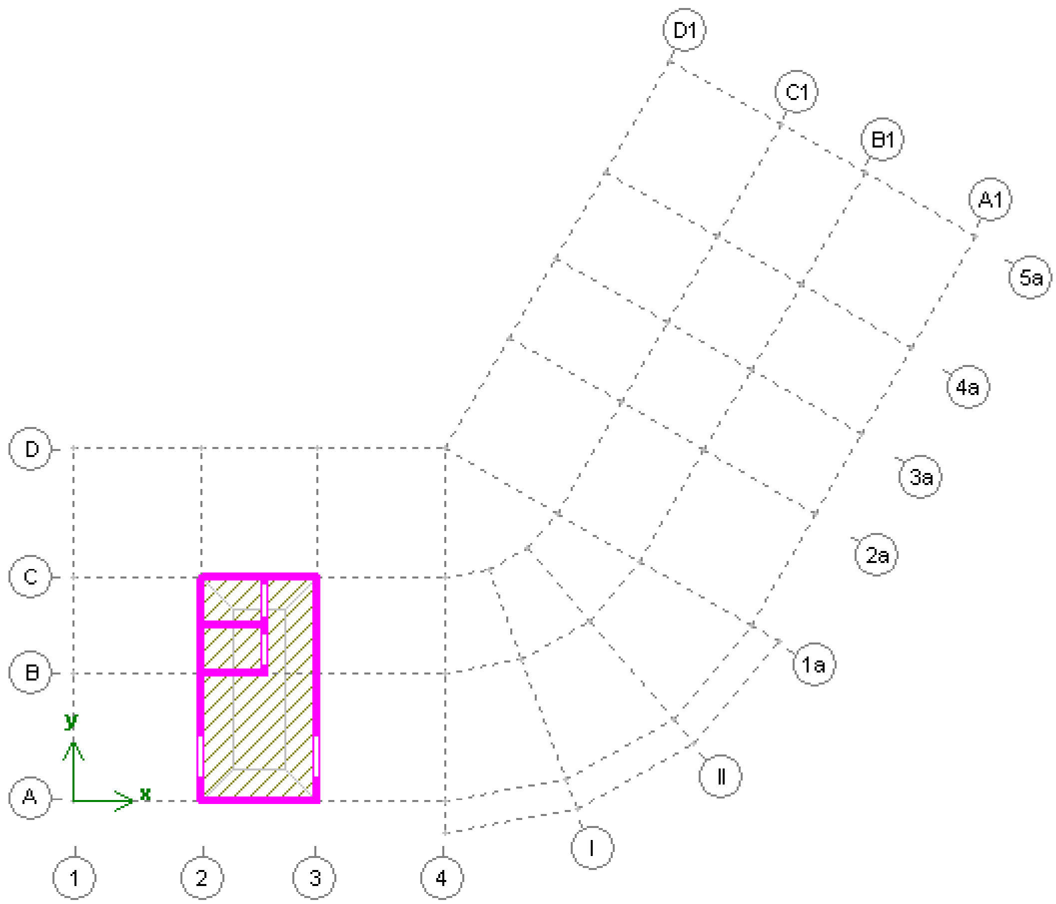
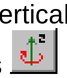


Figure 1.14.7 Floor slab

To present 3D view of the model:

- To present 3D view of the model, on the VIEW menu, point to **3D View** and then click **Whole structure** (button  on the toolbar).
- To switch from axonometric projection to perspective projection, on the VIEW menu, point to **Projection** and click **Perspective** (button  on the toolbar).
- To move the model down on the screen, click PAGE UP several times.
- To move the image away, click the DOWN ARROW key several times.
- To rotate the model about its vertical axis, use the key combinations CTRL+RIGHT ARROW or CTRL+LEFT ARROW (buttons  and  on the toolbar).

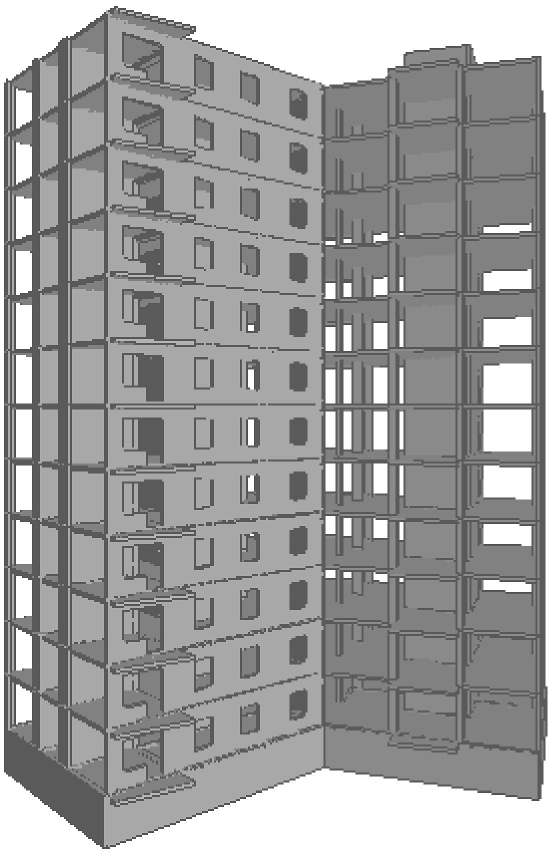


Figure 1.14.8 3D view of the model (perspective)

- To return to the main view, on the VIEW menu, click **Main view** (button on the toolbar).

Step 15. Defining mat foundations

*Mat foundations may be defined only at storey No.1. If you select **calculated stiffness** option, then moduli of subgrade reaction C1, C2 for mat foundations will be calculated according to the specified soil properties.*

To define contour of mat foundation:

- To define storey No.1 as current one, click the buttons and on the toolbar.
- To hide visualization of slabs, click the **Slabs** button (the button should become inactive).
- To define parameters and contour of mat foundation, on the MODEL menu, point to **Add** and click **Mat foundation** (button on the toolbar).
- In the **Add mat foundation** dialog box (see Fig.1.15.1), define the following parameters:
  - material – 3. Reinforced concrete B30 AIII AIII;
  - shift db = 0.5 m;
  - thickness b = 0.5 m;
  - under **Bedding**, select **Natural bed (calculated stiffness)** from the list.
  - load from dead load case 0.1 tf/m^2;

- load from live load case 0.1 tf/m^2.

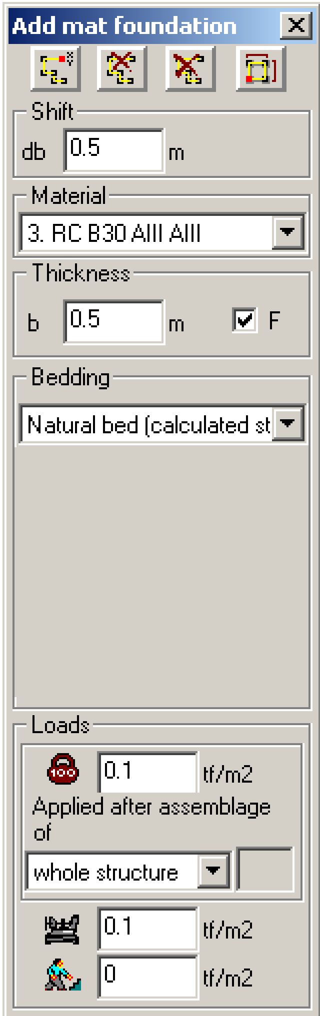


Figure 1.15.1 **Add mat foundation** dialog box

- Specify with the pointer node of intersection between line **1** and line **A** according to plan of the building.
- Specify with the pointer node of intersection between line **4** and line **A**.
- Specify with the pointer node of intersection between line **I** and line **A**.
- Specify with the pointer node of intersection between line **II** and line **A**.
- Specify with the pointer node of intersection between line **1a** and line **A1**.
- Specify with the pointer node of intersection between line **5a** and line **A1**.
- Specify with the pointer node of intersection between line **5a** and line **D1**.
- Specify with the pointer node of intersection between line **4** and line **D**.
- Specify with the pointer node of intersection between line **1** and line **D**.
- Specify with the pointer node of intersection between line **1** and line **A** (this is the first node that was specified) — contour of the mat foundation will be closed.

The specified mat foundation will look as presented in Figure 1.15.2.

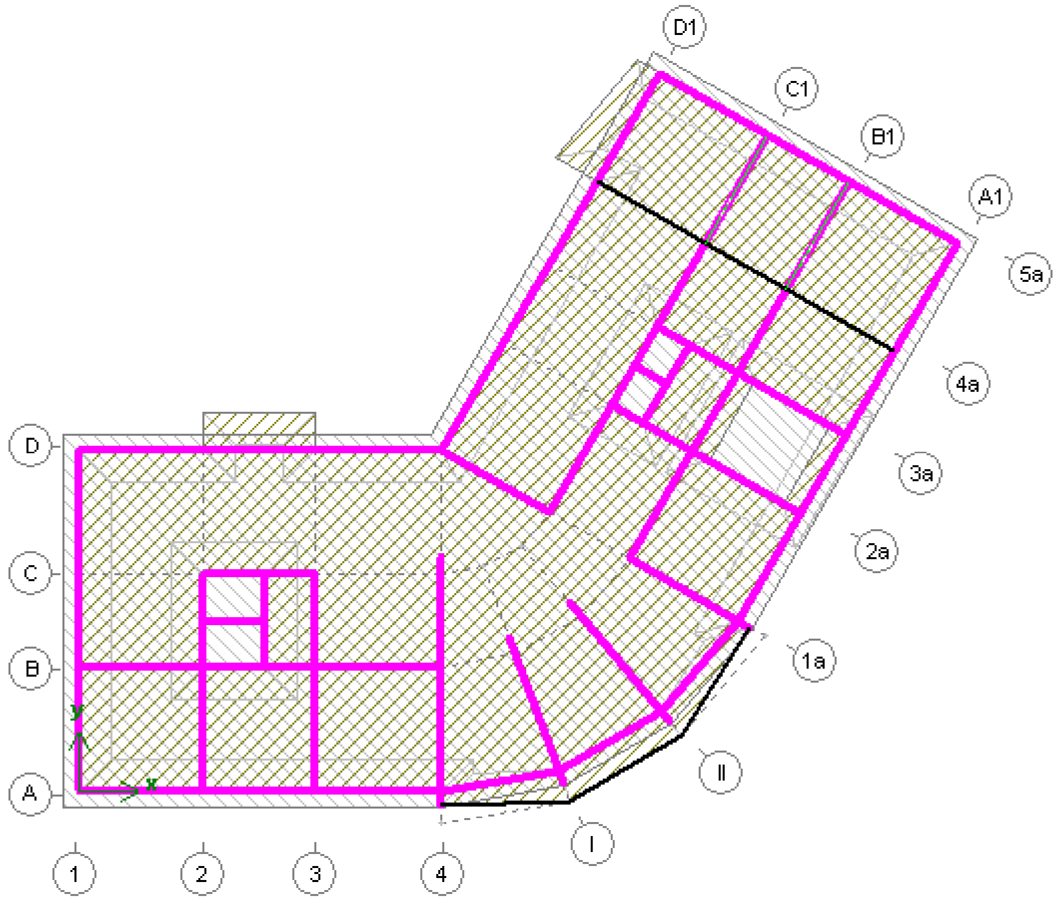


Figure 1.15.2 Mat foundation

Note that contour sides of mat foundation were shifted outside from the specified nodes at the specified distance db.

To mark mat foundations on the model:

- To display numbers and parameters of mat foundations on the model, on the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - click the **Numbers and parameters** tab ;
  - select the **Mat foundations: Numbers and parameters** check box;
  - click **Apply** .
- To close the **Display options** dialog box, click **Close** .
- To do the same from the toolbar, just click the **Mat foundations: Numbers and parameters** button on VISUALIZATION toolbar.

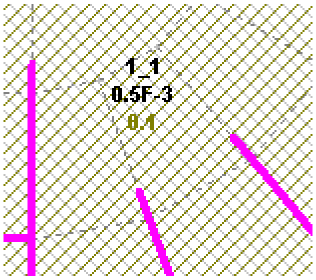


Figure 1.15.3 Notation for mat foundations on the model

Storey No. (always 1) and mat foundation No. are displayed for every mat foundation. In the lower row there are: thickness of mat foundation, fixation and number of material. If uniformly distributed load is defined across the whole mat foundation, then in the third row there is its value for the current load case.

- Click the **Mat foundations: Numbers and parameters** button once again to remove visualization of numbers and parameters of mat foundations (the button should become inactive).

*If necessary, it is possible to define openings in mat foundation. To do this, on the MODEL menu, point to **Add** and click **Opening in mat foundation** (button on the toolbar).*

Step 16. Defining sectional elevations

<u>To define sectional elevations:</u>

For export to ELEVATION (WALL) module, elevations that pass along line of walls should be defined separately.

- To define storey No.2 as current one, click the button 2 on the STOREYS toolbar.
- To define parameters and profile for sectional elevation, on the MODEL menu, point to **Add** and click **Sectional elevation** (button on the toolbar).
- In the **Define sectional elevation** dialog box (see Fig.1.16.1), remain all parameters by default (name of sectional elevation 1).

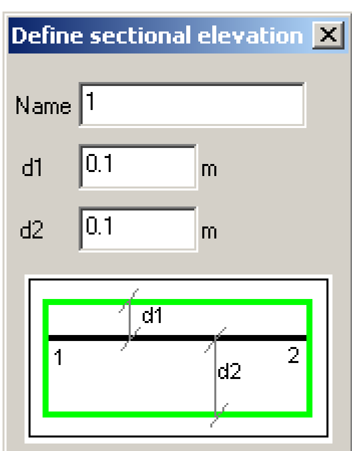


Figure 1.16.1 **Define sectional elevation** dialog box

For every sectional elevation, the program displays its name and dimensions for allowed displacements of elements from the plane of sectional elevation (by default, 0.1m from every side of elevation).

- Specify with the pointer node of intersection between line **1** and line **A** according to plan of the building.
- Specify with the pointer node of intersection between line **4** and line **A**.

Sectional elevation 1 will be added.

Define sectional elevation 2.

- Specify with the pointer node of intersection between line **2** and line **A**.
- Specify with the pointer node of the balcony beyond the intersection between line **2** and line **D**.

Define sectional elevation 3.

- Specify with the pointer node of intersection between line **1a** and line **D1**.
- Specify with the pointer node of intersection between line **5a** and line **D1**.

Specified sectional elevations should look as presented in Figure 1.16.2.

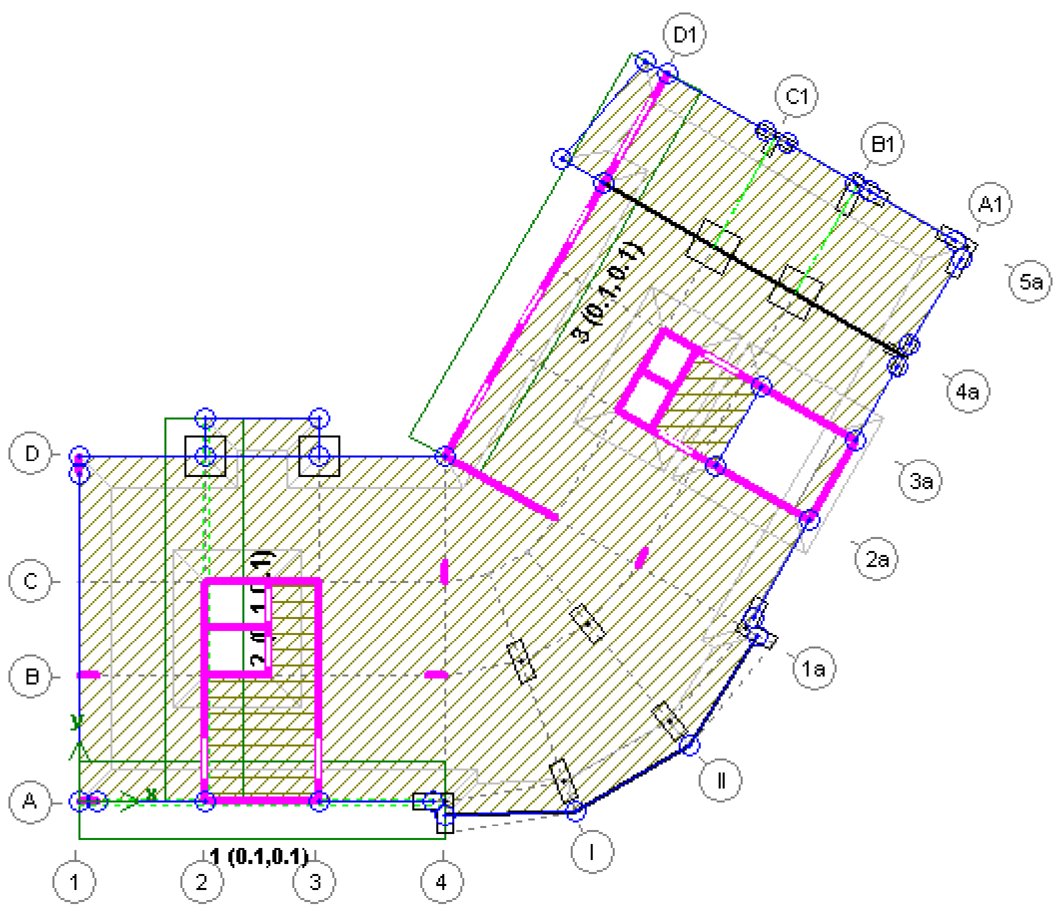


Figure 1.16.2 Sectional elevations

- To hide visualization of elevation parameters, click the **Sectional elevations** button on VISUALIZATION toolbar (the button should become inactive).

*It is recommended to export data to ELEVATION (WALL) module after finite element analysis. FEA results for sectional elevations and groups of walls are exported to ELEVATION (WALL) module as a **map of imported displacements** at nodes and lines where neighbouring elements join the model. In this case elements that join the sectional elevation model are not exported to design module. Their presence at nodes of design model is simulated with group of initial (forced) displacements – linear translations and rotation angles. Behaviour of fragment in the whole design model of the structure is also considered properly.*

<u>To unite walls for export to ELEVATION (WALL) module:</u>

For export to ELEVATION (WALL) module, selected walls may be united into 'Group of walls as sectional elevation'. There are some advantages of such approach – when you create and edit groups, analysis results are not cancelled; walls may be united between the specified storeys (united walls should belong to the same plane) as well as from top to bottom along the whole height of the structure.

- To display numbers and parameters of walls (see Fig.1.16.3) on the model, on the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - click the **Numbers and parameters** tab ;
  - select the **Walls: Numbers and parameters** check box;
  - click **Apply** .
- To close the **Display options** dialog box, click **Close** .

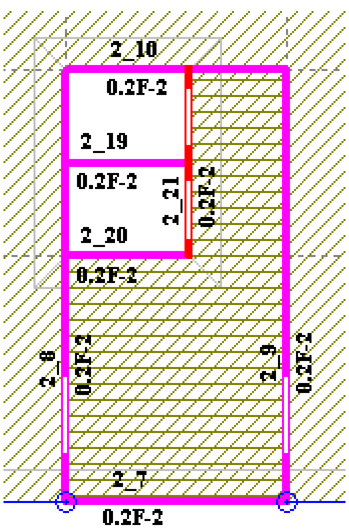


Figure 1.16.3 Notation for walls on the model

- To select the wall No.2_21 between lines 2, 3 and lines B, C, on the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).
- To select all walls located above and below selected wall, on the MODEL menu, point to **Select** and click **Walls on the same chords as selected ones**.

- In the **Select walls on the same chords with selected ones** dialog box (see Fig.1.16.4), remain all parameters with default values and click **OK**.

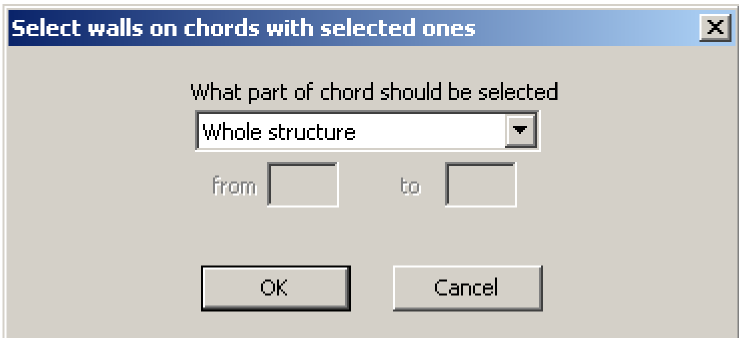


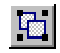
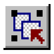
Figure 1.16.4 **Select walls on chords with selected ones** dialog box

- To activate the mode in which it is possible to edit and delete selected elements of all storeys, on the MODEL menu, point to **Select / Commands for** and click **Selected elements of all storeys** (button on the toolbar).
- To create 'Group of walls as sectional elevation' along the whole height of the structure, on the MODEL menu, point to **Edit** and click **Unite walls for export to ELEVATION** (button on the toolbar).
- In the **Unite walls for export** dialog box, do the following:
  - to create new group Grp No.1, click the **Create group** button (see Fig.1.16.5);
  - to add walls selected on the model to the current group Grp No.1, click the **Add selected wall to group** button .

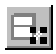
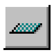
Figure 1.16.5 **Unite walls for export** dialog box

Group of walls along the whole height of the structure is created. Walls united into the group are coloured light blue on the model.

- Create 'Group of walls as sectional elevation' between the specified storeys (from the 3rd to the 5th storey).
- To define storey No.3 as current one, click button 3 on the STOREYS toolbar.
- To select the wall No.3_14 on line 2a between lines A1, C1, on the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).
- To define storey No.4 as current one, click appropriate button on the STOREYS toolbar.

- To select the wall No.4_14 on line 2a between lines A1, C1.
- To define storey No.5 as current one, click appropriate button on the STOREYS toolbar.
- To select the wall No.5_14 on line 2a between lines A1, C1.
- In the **Unite walls for export** dialog box, do the following:
  - to create new group Grp No.2, click the **Create group** button 
  - to add walls selected on the model to the current group Grp No.2, click the **Add selected wall to group** button  .

Group of walls between the specified storeys is created. Walls united into the group are coloured light blue on the model.

- To turn off the mode for unification of walls, on the MODEL menu, point to **Edit** and click **Unite walls for export to ELEVATION** (button  on the toolbar) – the button should become inactive.
- To restore mode where it is not possible to edit and delete selected elements of all storeys, on the MODEL menu, point to **Select / Commands for** and click **Selected elements of current storey only** (button  on the toolbar).

Step 17. Defining earthquake and wind loads

<u>To define earthquake and wind loads:</u>

- To define earthquake and wind loads, on the **MODEL** menu, point to **Add** and click **Earthquake and wind loads** (button  on the toolbar).
- In the **Earthquake and wind loads** dialog box (see Fig.1.17.3), do the following:
  - select the **Earthquake 1** check box;
  - define direction 0 degrees;
  - select the **Earthquake 2** check box;
  - define direction 90 degrees;
  - to define parameters for the default building code (SNIP II-7-81 amendments 2000), click **Parameters**;
  - in the **SNIP II-7-81 (amend. 2000)** dialog box, define the following parameters:
    - seismicity of the site 7;
    - soil category III;
    - other parameters remain by default;
    - click **OK**.

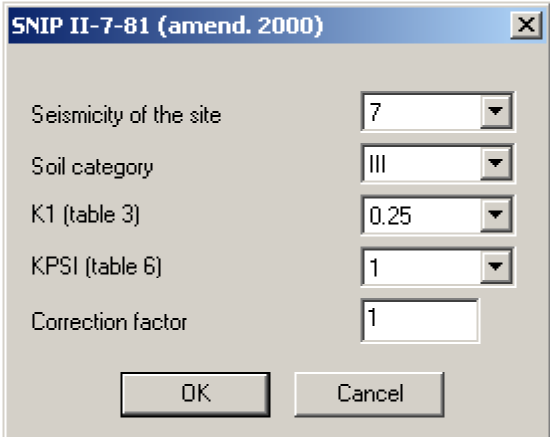


Figure 1.17.1 **SNIP II-7-81 (amend.2000)** dialog box

- In the **Earthquake and wind loads** dialog box, select the **Wind 1** check box;
- define direction 90 degrees;
- select the **Wind 2** check box;
- define direction 135 degrees;
- to define parameters for the default building code (SNIP 2.01.07-85), click **Parameters**;
- in the **SNIP 2.01.07-85** dialog box (see Fig.1.17.2), define the following parameters:
  - wind region II;
  - site type B;
  - other parameters remain by default;
  - click **OK**.

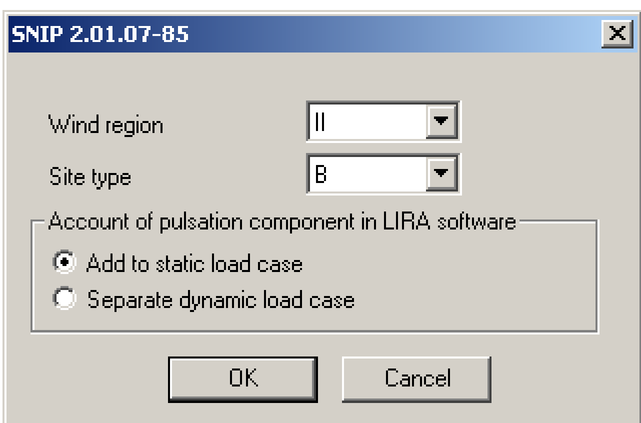


Figure 1.17.2 **SNIP 2.01.07-85** dialog box

➪ In the **Earthquake and wind loads** dialog box, click **OK**.

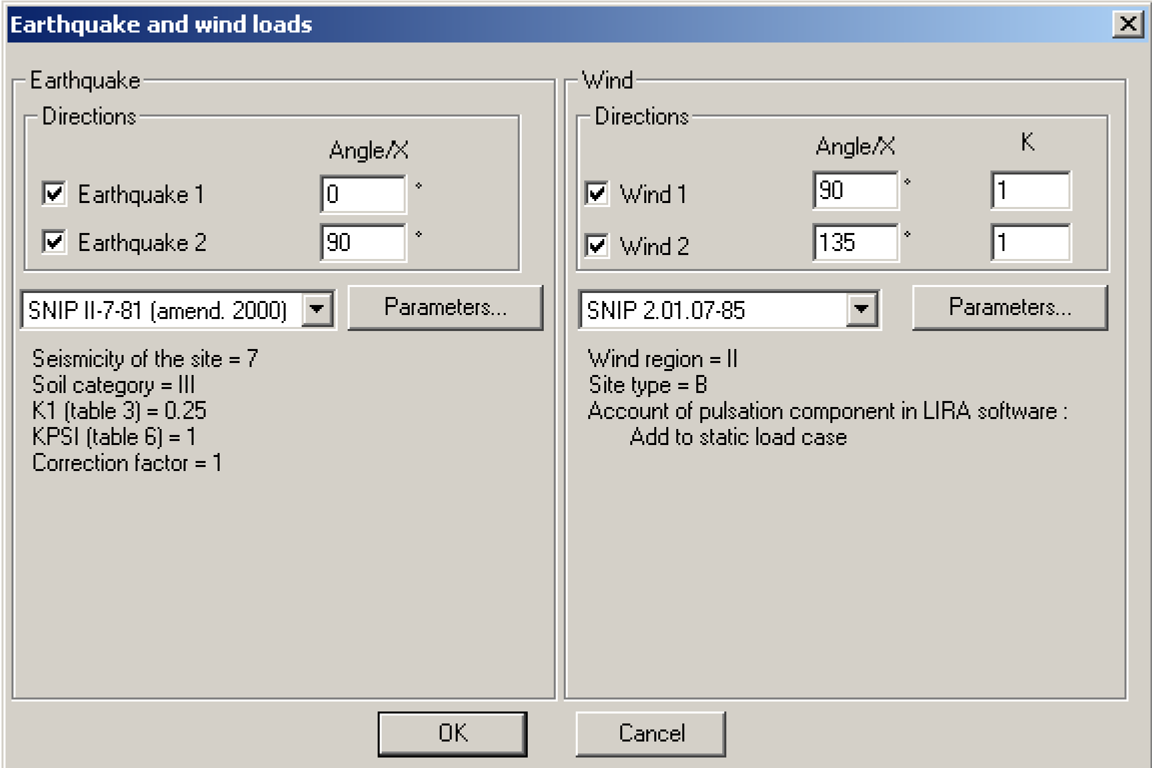


Figure 1.17.3 **Earthquake and wind loads** dialog box

<u>To define load factors:</u>

*To define load factors in a separate table, on the LOAD CASE menu, click **Coefficients**.*

<u>To mark earthquake and wind loads on the model:</u>

*For every specified earthquake and wind load you will see direction — red arrow displayed in the left bottom corner of the screen. To select the current load case, use the LOAD CASE menu. For example, to select load case Wind 2, on the LOAD CASE menu, click **2nd Wind** (button on the toolbar).*

Step 18. Analysis of the whole structure

<u>To analyse the whole structure:</u>

*There are two types of analysis in BUILDING module – preliminary (simplified) analysis and Finite Element Analysis (FEA). In preliminary (simplified) analysis you generate the model of the structure, gather loads on it, determine and check sections of reinforced concrete (RC) elements. As this analysis is preliminary one, it is always necessary to perform FEA after it and FEA results should be considered as final results. Preliminary analysis consists of several analyses. To perform preliminary analysis, on the ANALYSIS menu, use **Analysis of current storey** and **Analysis of whole structure** commands. Diagnostic procedure is performed during analysis of the generated model. The errors are displayed in the box. When you click a row with description of an error, the element in which this error occurred is coloured red on the model. Parameters of elastic foundation (moduli of subgrade reaction C1, C2) are not calculated in preliminary analysis.*

- To perform preliminary analysis, on the ANALYSIS menu, click **Analysis of whole structure** (button on the toolbar).
- To define storey No.1 as current one, click the button on the STOREYS toolbar.
- On the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - on the **Results of preliminary analysis** tab, select the **For whole structure** check box;
  - click **Apply** .
- Close the **Display options** dialog box.
- To do the same, you could click the **Results of preliminary analysis for structure** button on VISUALIZATION toolbar.

Analysis results should look as presented in Figure 1.18.1.

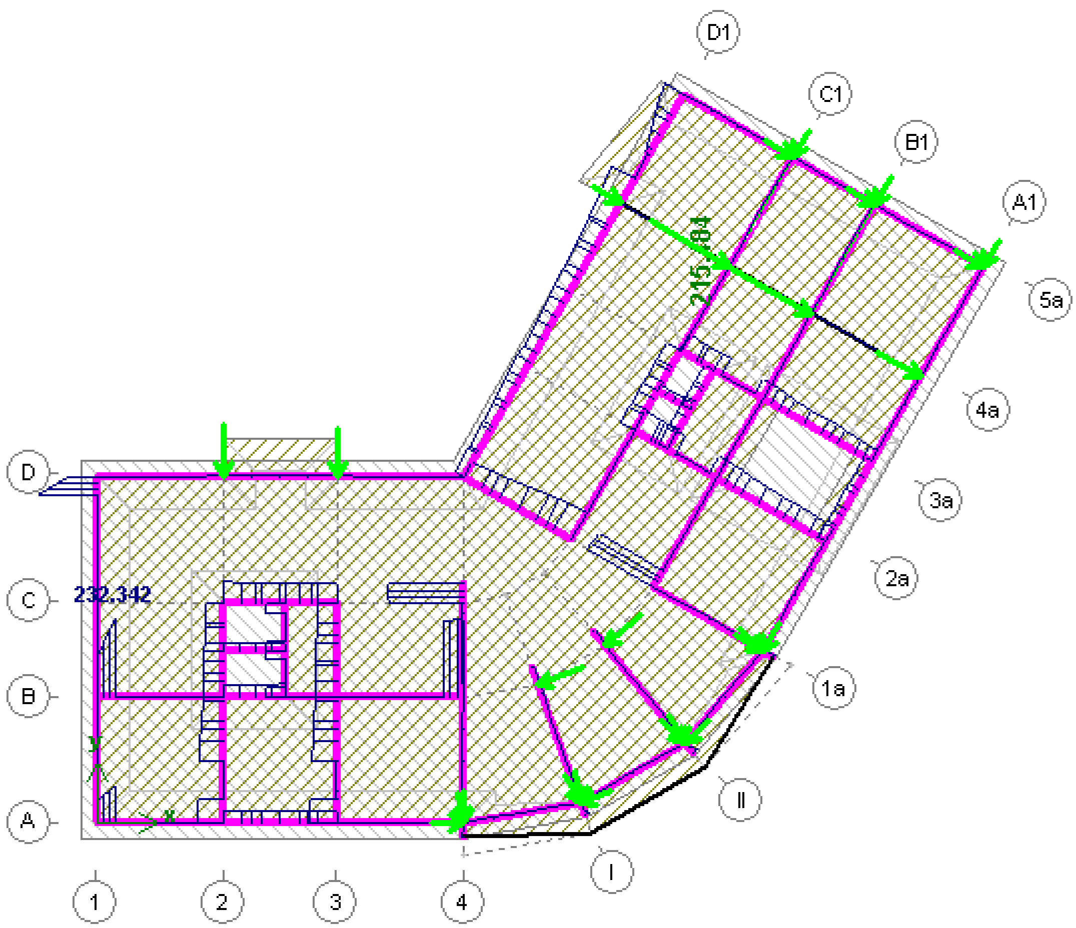


Figure 1.18.1 Analysis results for the whole structure (storey No.1, dead load case)

To visualize analysis results:

- To define storey No.2 as current one, click the button on the STOREYS toolbar.

Analysis results should look as presented in Figure 1.18.2

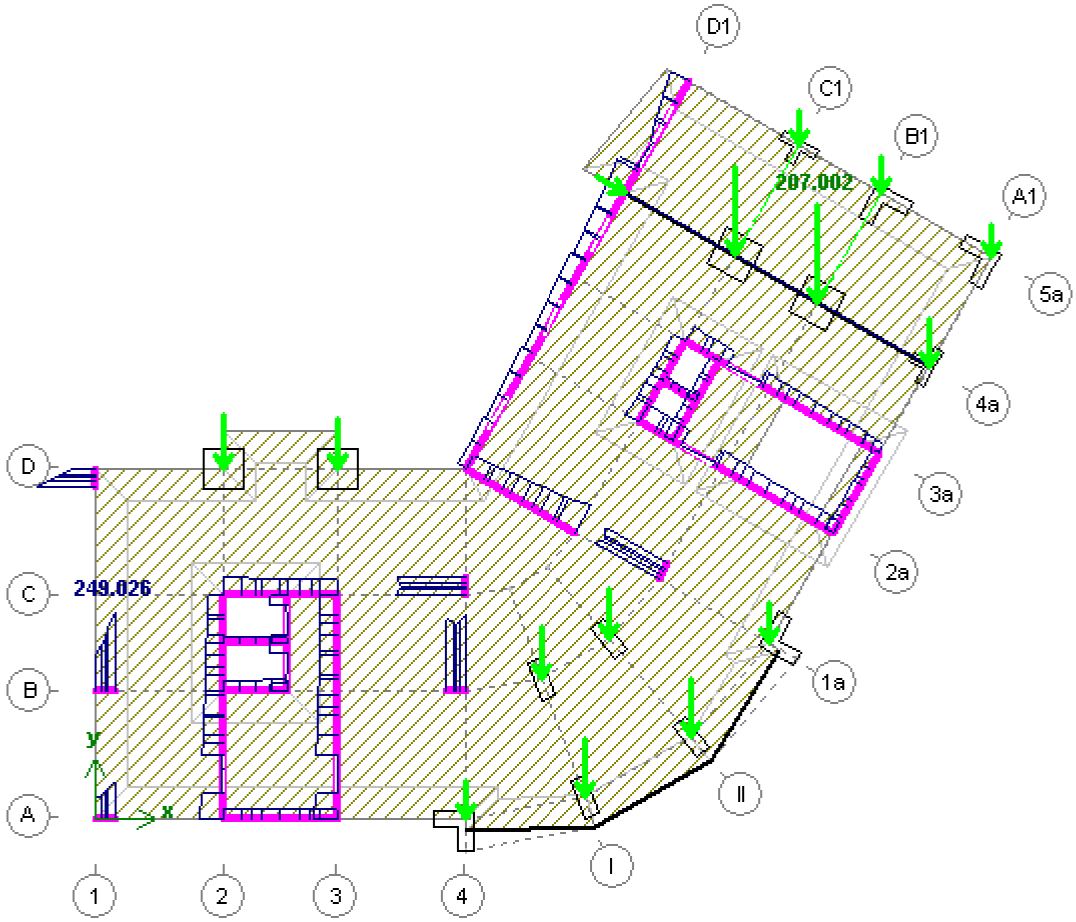


Figure 1.18.2 Analysis results for the whole structure (storey No.2, dead load case)

➪ To define the 1st wind load case as the current one, click button on the LOAD CASES toolbar.

Analysis results should look as presented in Figure 1.18.3

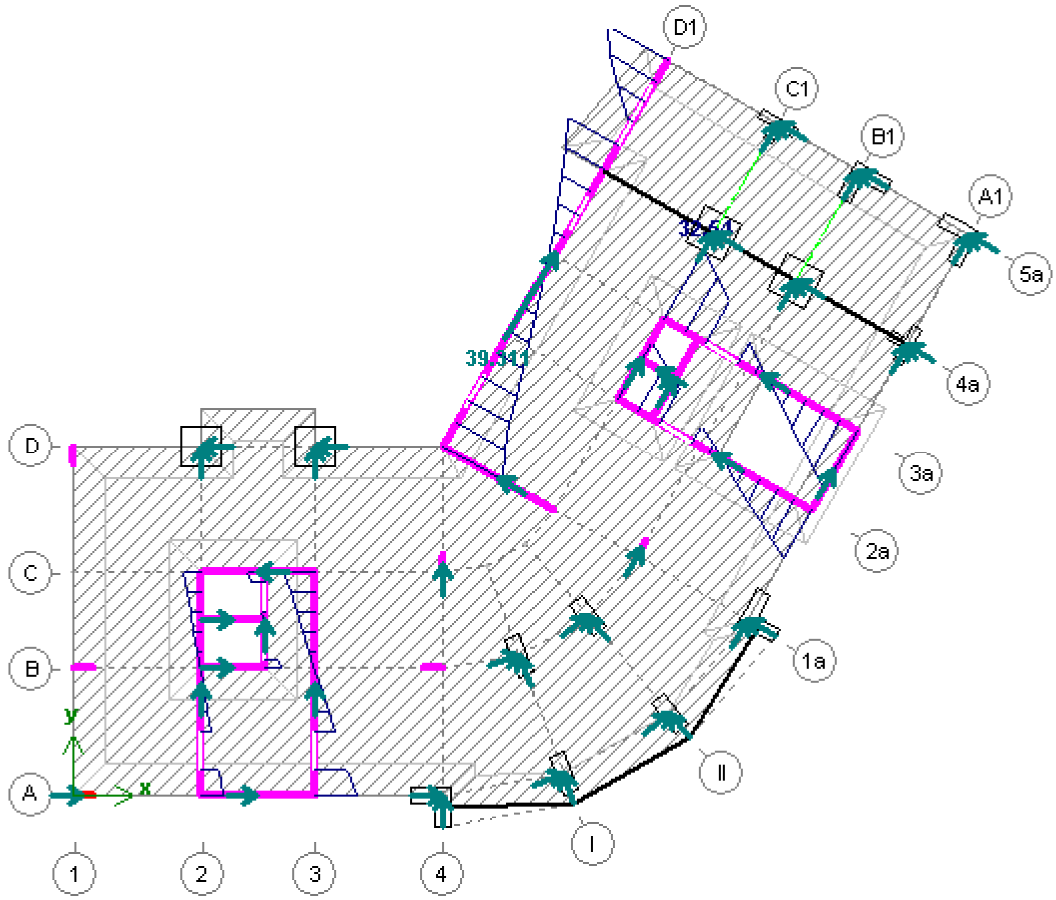


Figure 1.18.3 Analysis results for the whole structure (storey No.2, 1st wind load case)

<u>To present 3D view of the model:</u>

- To present 3D view of the model, on the VIEW menu, point to **3D View** and then click **Whole structure** (button on the toolbar).
- To visualize storey No.2, on the VIEW menu, click **Structure fragment**.
- In the **Storeys** dialog box (see Fig.1.18.4), define the following parameters:
  - from storey No. 2;
  - to storey No.2.
  - click **OK**.

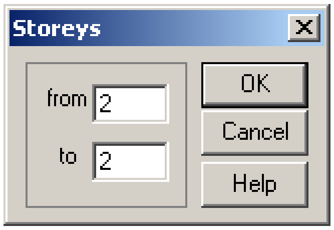


Figure 1.18.4 **Storeys** dialog box

- Hide visualization of floor slabs and partitions and show analysis results of the whole structure (use the **Display options** command on the VIEW menu).

- In the **Display options** dialog box, do the following:
  - on the **Elements** tab, clear the **Partitions** check box;
  - clear the **Slabs** check box;
  - select the **Results of preliminary analysis** tab;
  - select the **For whole structure** check box;
  - click **Apply**.
- To close the **Display options** dialog box, click **Close**.
- To do the same from the toolbar, just use the **Partitions**, **Slabs** and **Analysis results for the whole structure** buttons on VISUALIZATION toolbar.

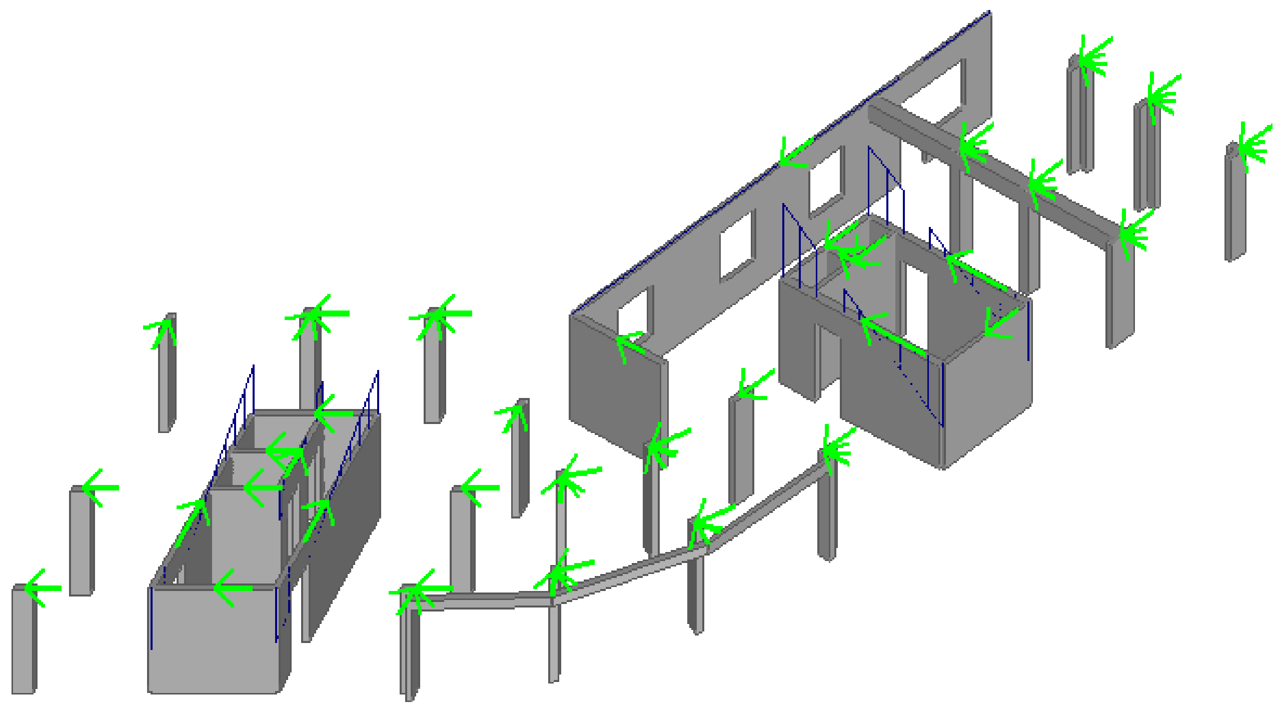


Figure 1.18.5 3D view of the model (storey No.2, 1st wind load case)

- To define the dead load case as the current one, click button on the LOAD CASES toolbar.

Analysis results should look as presented in Figure 1.18.6

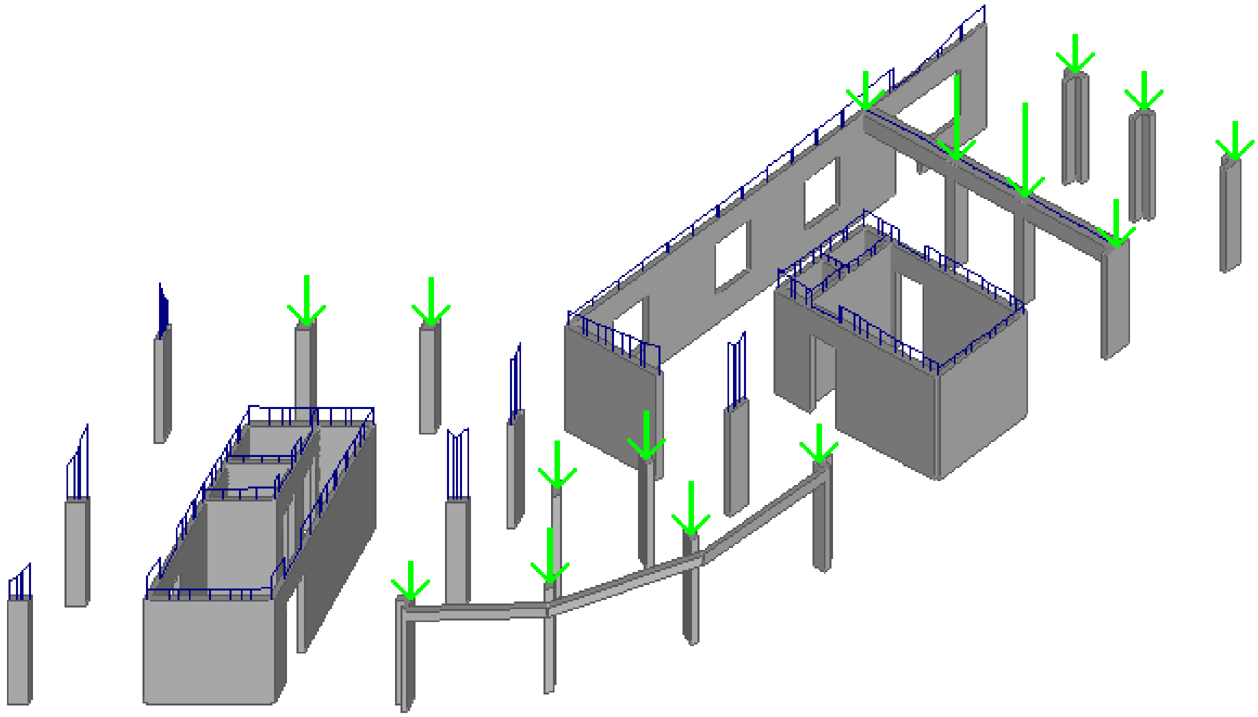
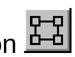
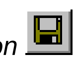


Figure 1.18.6 3D view of the model (storey No.2, dead load case)

➪ To return to the main view, on the VIEW menu, click **Main view** (button  on the toolbar).

To save analysis results:

 *When you save the model with the **Save** command on the FILE menu (button  on the toolbar), analysis results are saved to *.chg file as well as the model. File may be saved even without analysis results. To do this, before the save procedure, on the ANALYSIS menu, click **Cancel analysis results**.*

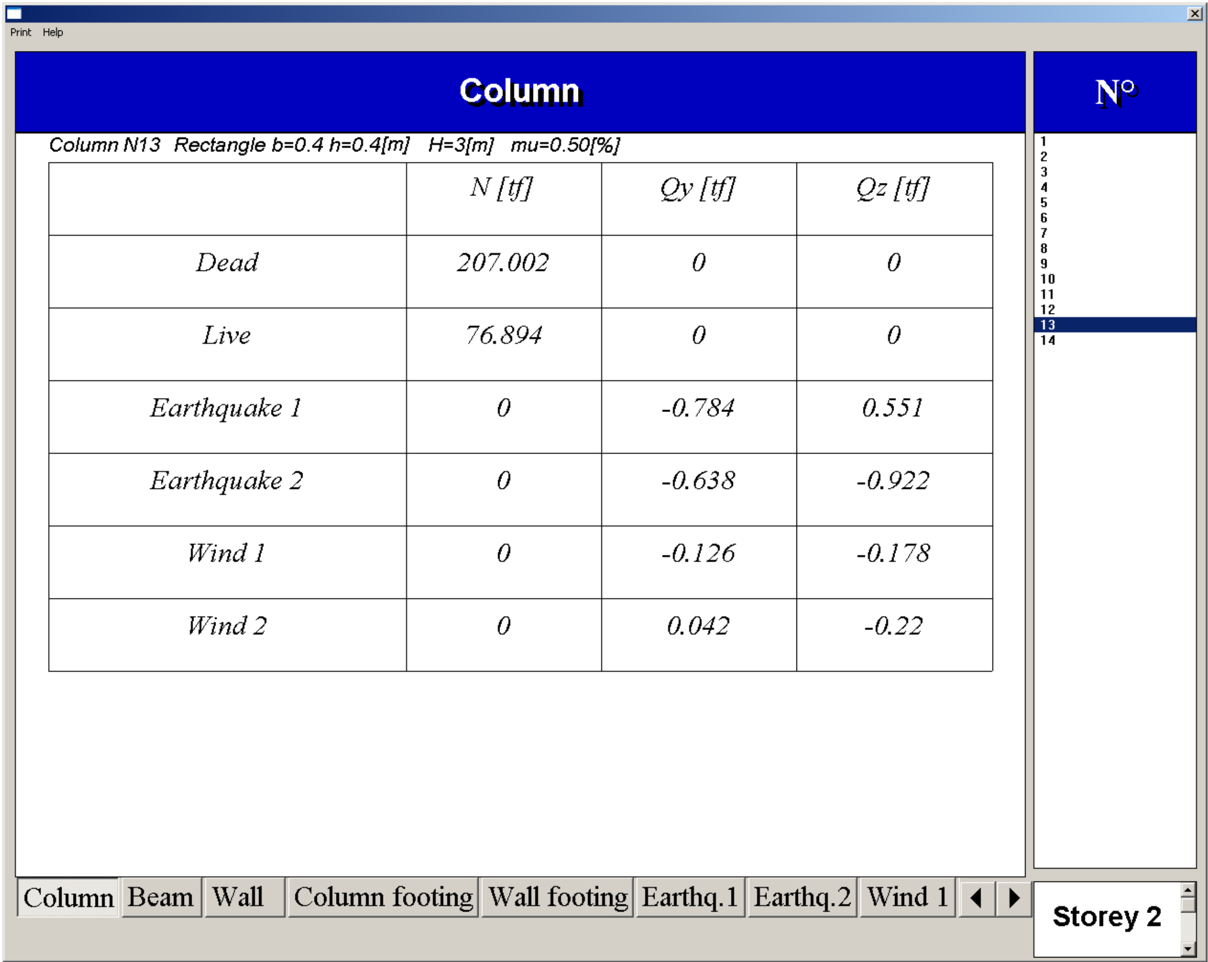
Step 19. Review and generation of the report

To review table of loads:

➪ After analysis of the whole structure, to review table of loads, on the RESULTS menu, point to **Analysis results** and click **Table of loads** (preliminary analysis).

➪ In the dialog box with tables of loads (see Fig.1.19.1), do the following:

- select **Storey No.2** in the lower-right corner of the window;
- select **Column No.13** in the list;



	N [tf]	Qy [tf]	Qz [tf]
Dead	207.002	0	0
Live	76.894	0	0
Earthquake 1	0	-0.784	0.551
Earthquake 2	0	-0.638	-0.922
Wind 1	0	-0.126	-0.178
Wind 2	0	0.042	-0.22

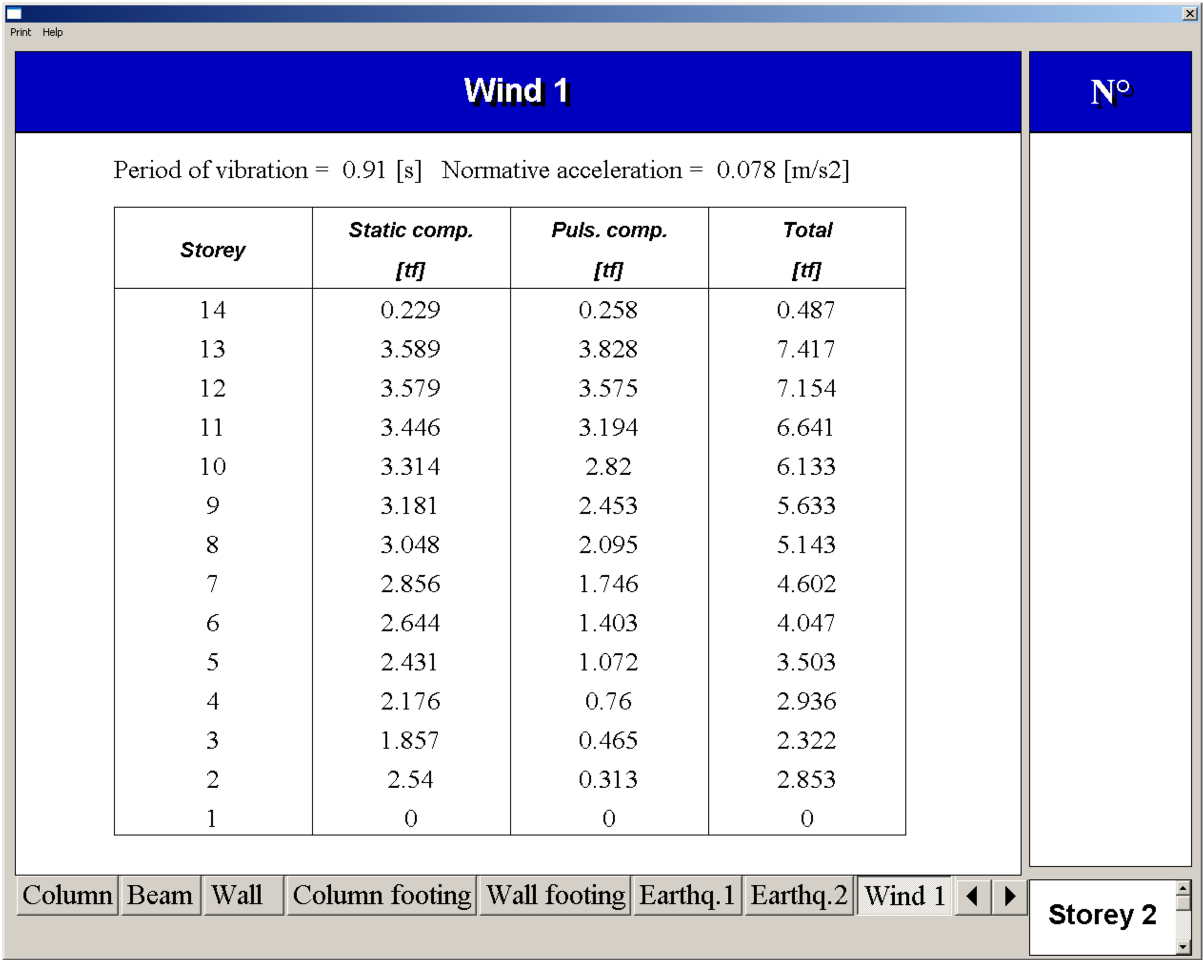
Figure 1.19.1 Table of loads (**Column** tab)

On the **Column** tab of the window, there will be summary table of loads on column No.13 of the second storey.

*Loads are presented in the tables without account of coefficients defined in the **Coefficients** dialog box (see LOADS menu).*

- select the **Wall 1** tab (see Fig.1.19.2);

On the **Wall 1** tab of the window, there is summary table of loads of the 1st wind load case.



Storey	Static comp. [tf]	Puls. comp. [tf]	Total [tf]
14	0.229	0.258	0.487
13	3.589	3.828	7.417
12	3.579	3.575	7.154
11	3.446	3.194	6.641
10	3.314	2.82	6.133
9	3.181	2.453	5.633
8	3.048	2.095	5.143
7	2.856	1.746	4.602
6	2.644	1.403	4.047
5	2.431	1.072	3.503
4	2.176	0.76	2.936
3	1.857	0.465	2.322
2	2.54	0.313	2.853
1	0	0	0

Figure 1.19.2 Table of loads (**Wind 1** tab)

Loads are presented at the floor slab levels of every storey. For wind load, there are its components – static and pulsation.

- To close the window with tables of loads, click the **Close** button .

To review bills of materials:

- After analysis of the whole structure, to review bill of materials, on the RESULTS menu, point to **Analysis results** and click **Bill of materials** (button on the toolbar).
- In the **Prices** dialog box (see Fig.1.19.3), define the following parameters:
  - price for 1m^3 of concrete - 1 (price is defined conventionally);
  - price for 1kg of reinforcement - 1;
  - price for 1m^2 of formwork - 1;
  - click **OK**.
- In the bill of materials table, see the data on the **Total** tab.

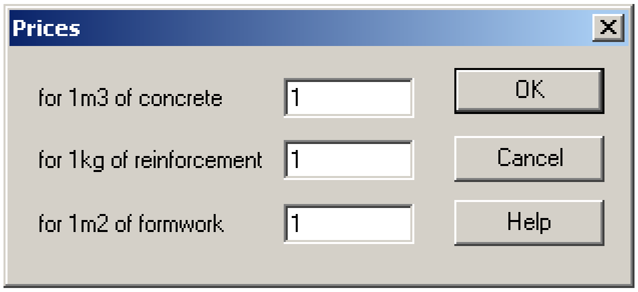


Figure 1.19.3 **Prices** dialog box

Reinforcement is calculated for every element based on obtained design percentage of reinforcement. Percentage of reinforcement is calculated according to approximate formulas for analysis of RC elements on accumulated loads, in this case - according to results of preliminary analysis of the whole structure.

- To close the window with bill of materials, click the **Close** button .

To generate and save the report:

- To generate the report, on the RESULTS menu, point to **Analysis results** and click **Report in RTF format** (preliminary analysis). With this command the report will be generated according to results of preliminary analysis of the whole structure.

- In the **Report for preliminary analysis** dialog box (see Fig.1.19.5), do the following:
  - under **Input data**, select the **Load coefficients** check box;
  - under **Analysis results**, in the **Columns** box, delete the value 1-14(1-14);
  - for the **Columns** option, click **+Chord** button;
  - in the **Define vertical chord of columns** dialog box (see Fig.1.19.4), define the following parameters:
    - storey No. 2;
    - column No. 13.
    - click OK;

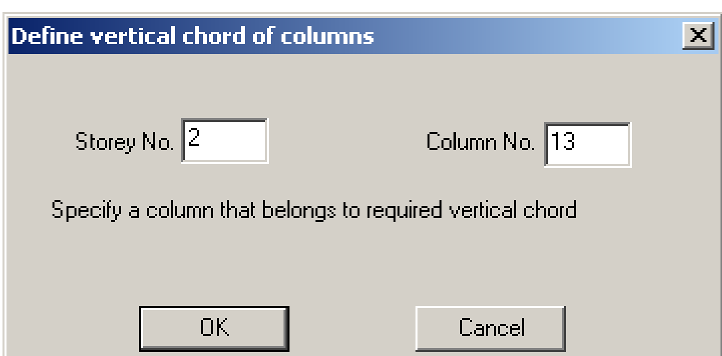


Figure 1.19.4 **Define vertical chord of columns** dialog box

It is possible to include in report the data about certain elements of the model. In this case, we selected chord of columns that passes along whole height of the structure through column No.13 of the second storey, beams No.1 and No.2 of the second storey and wall No.2 of the second storey.

In the **Columns** box of the **Report** dialog box there will be numbers of columns that belong to the same chord.

- for the **Beams** option, define 2(1,2);
- for the **Walls** option, define 2(2);

➪ Click **OK**.

➪ In the **Save as** dialog box, indicate the file *Model1.rtf* in **Notes** directory in the base MONOMAKH-SAPR directory and click **Save**.

<u>To review the report:</u>

➪ On the taskbar, click the **Start** button and point to **Programs** / MS Word.

➪ To open the report file in MS Word, on the **FILE** menu, click **Open**.

➪ In the standard **Open** dialog box, do the following:

- in the **Files of type list**, select *Rich text format* (**.rtf*);
- open the **Notes** directory in the base MONOMAKH-SAPR directory;
- open file *Model1.rtf*.

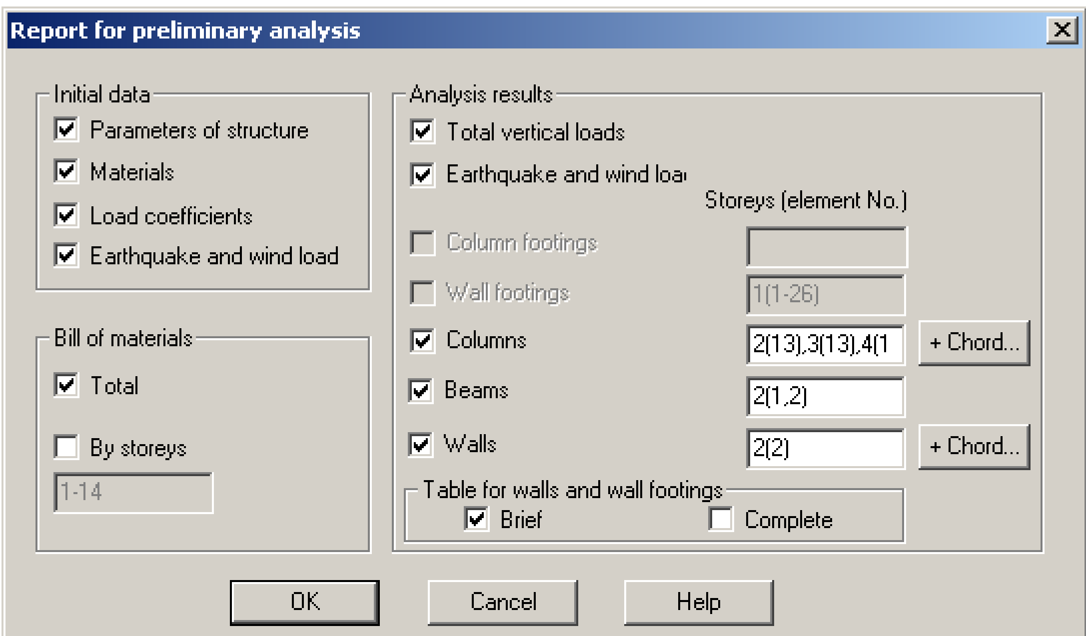


Figure 1.19.5 **Report for preliminary analysis** dialog box

Report file contains several tables that may be reviewed and printed. The file contains input data and results for preliminary analysis of the whole structure.

➪ Close the report file in MS Word.

Step 20. Finite Element Analysis

There are two types of analysis in BUILDING module – preliminary (simplified) analysis and Finite Element Analysis (main). FEA is allowed only when preliminary analysis of the whole structure is completed. FEA is a necessary one and FEA results should be considered as final results.

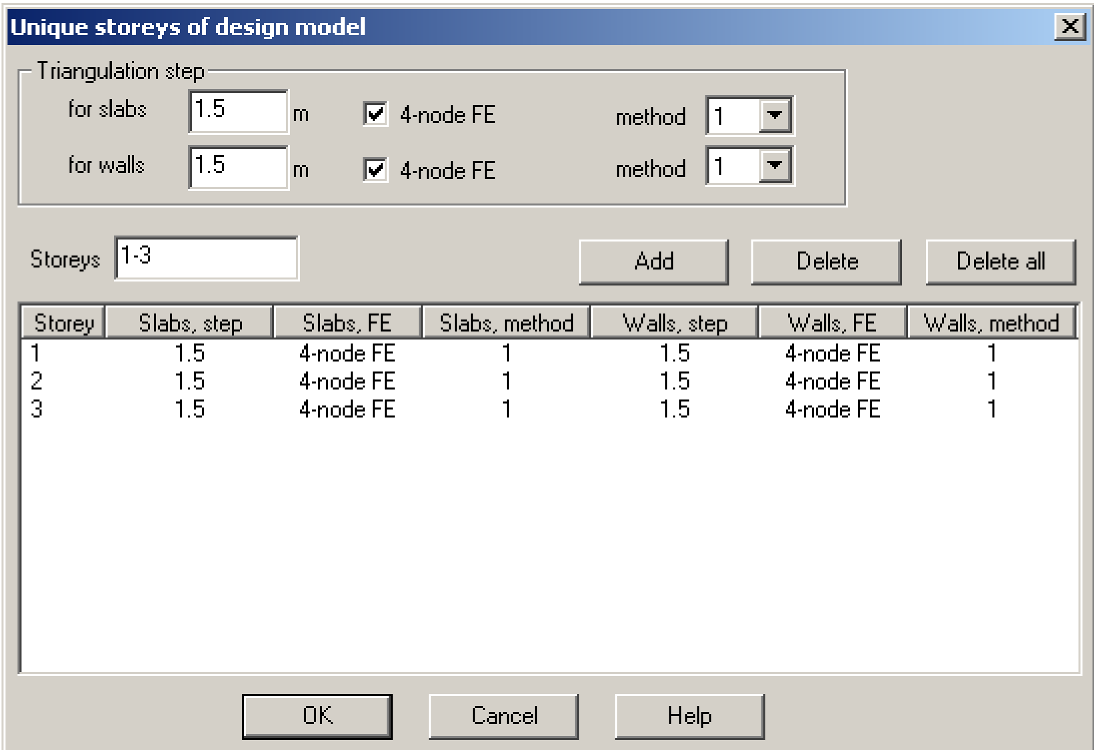


Figure 1.20.1 **Unique storeys of design model** dialog box

For FEA, the finite element model is generated automatically according to the data specified in the ***Finite element analysis*** *dialog box. Analysis (static and dynamic) is preformed by the solver by finite element method (FEM). Output data of analysis contains nodal displacements, stresses and forces in elements. According to FEA results, the program does the following:*
- total loads on walls, columns and footings are computed;
- stresses are determined at design levels defined for the BRICK module;
- sections of RC elements are repeatedly checked on obtained forces;
- design % of reinforcement for RC elements are computed once again;
- data is prepared to export into design modules.

➪ To perform FEA, on the ANALYSIS menu, click **Finite element analysis** (button on the toolbar).

➪ In the **Finite element analysis** dialog box (see Fig.1.20.2), define the following parameters:

- triangulation step for slabs 3 m;
- triangulation step for walls 3 m;
- select the **4-node FE** (slabs) check box;
- select the **4-node FE** (walls) check box;
- select the **4-node FE** (mat foundations) check box;
- click **Specify unique storeys**;

- in the **Unique storeys of design model** dialog box (see Fig.1.20.1), define the following parameters:
  - select the **4-node FE** (slabs) check box;
  - select the **4-node FE** (walls) check box;
  - define numbers of storeys 1-3;
  - other parameters remain by default (triangulation step for slabs and walls is 1 m);
  - click **Add**;

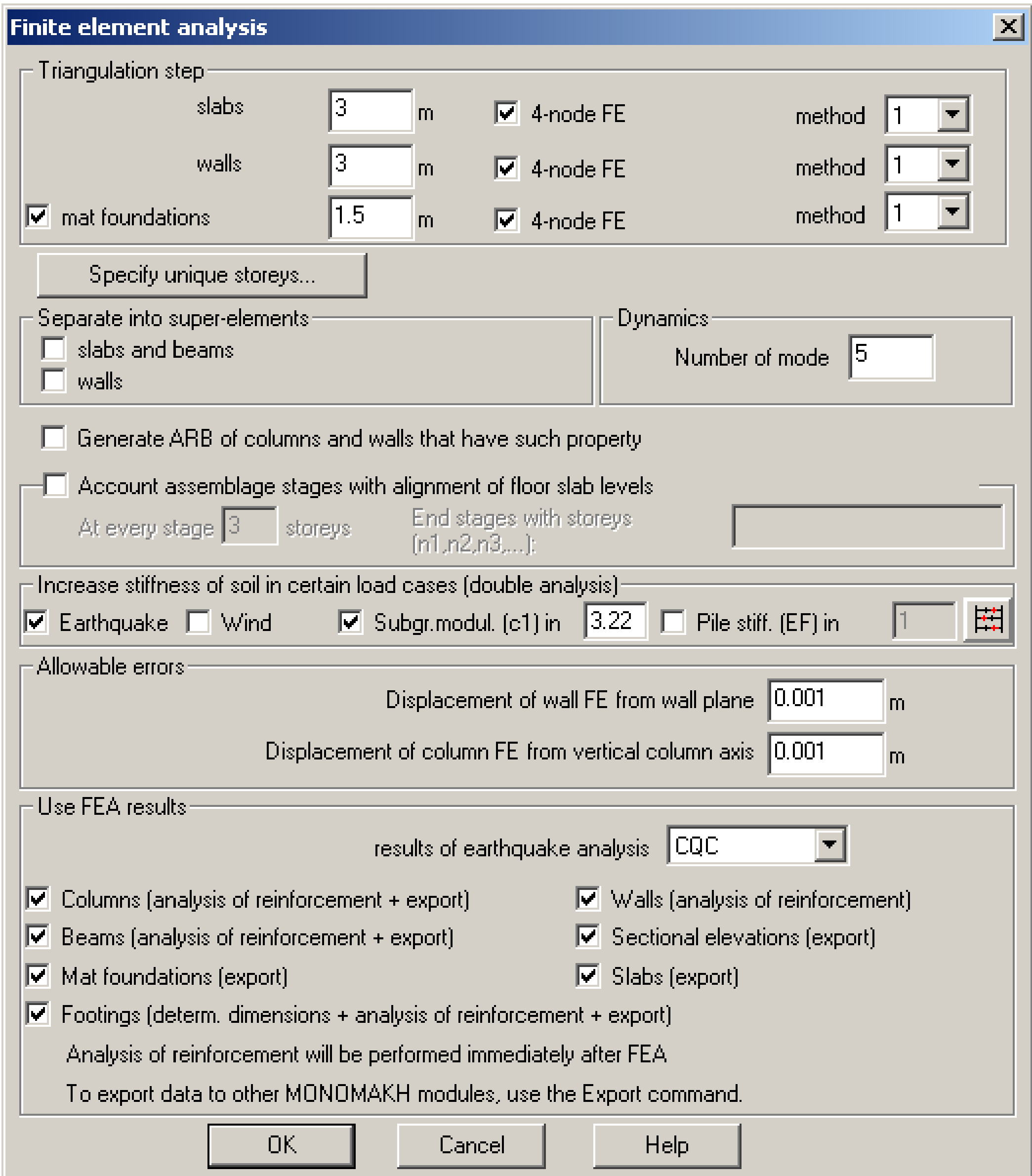
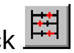


Figure 1.20.2 **Finite element analysis** dialog box

In the **Unique storeys of design model** dialog box, the table will contain lines with description of unique (different from the main one) division into finite elements for storeys No.1-3.

- click **OK**;

- to consider **Increase in soil stiffness for certain load cases** (dual analysis), select the **Earthquake** check box (see Fig.1.20.2);
- select the check box **Subgrade moduli (c1) in** (by default, subgrade modulus for calculation of displacements and forces from earthquake load will be increased by factor of 1);

- to calculate increase factor that will be the most appropriate for this model, click  – the value **Subgrade moduli (c1)** will be calculated automatically;
- other parameters remain by default;

➪ Click **OK**.

In the **Solver** window (see Fig.1.20.3) there will be design model, main parameters of the model and analysis protocol.

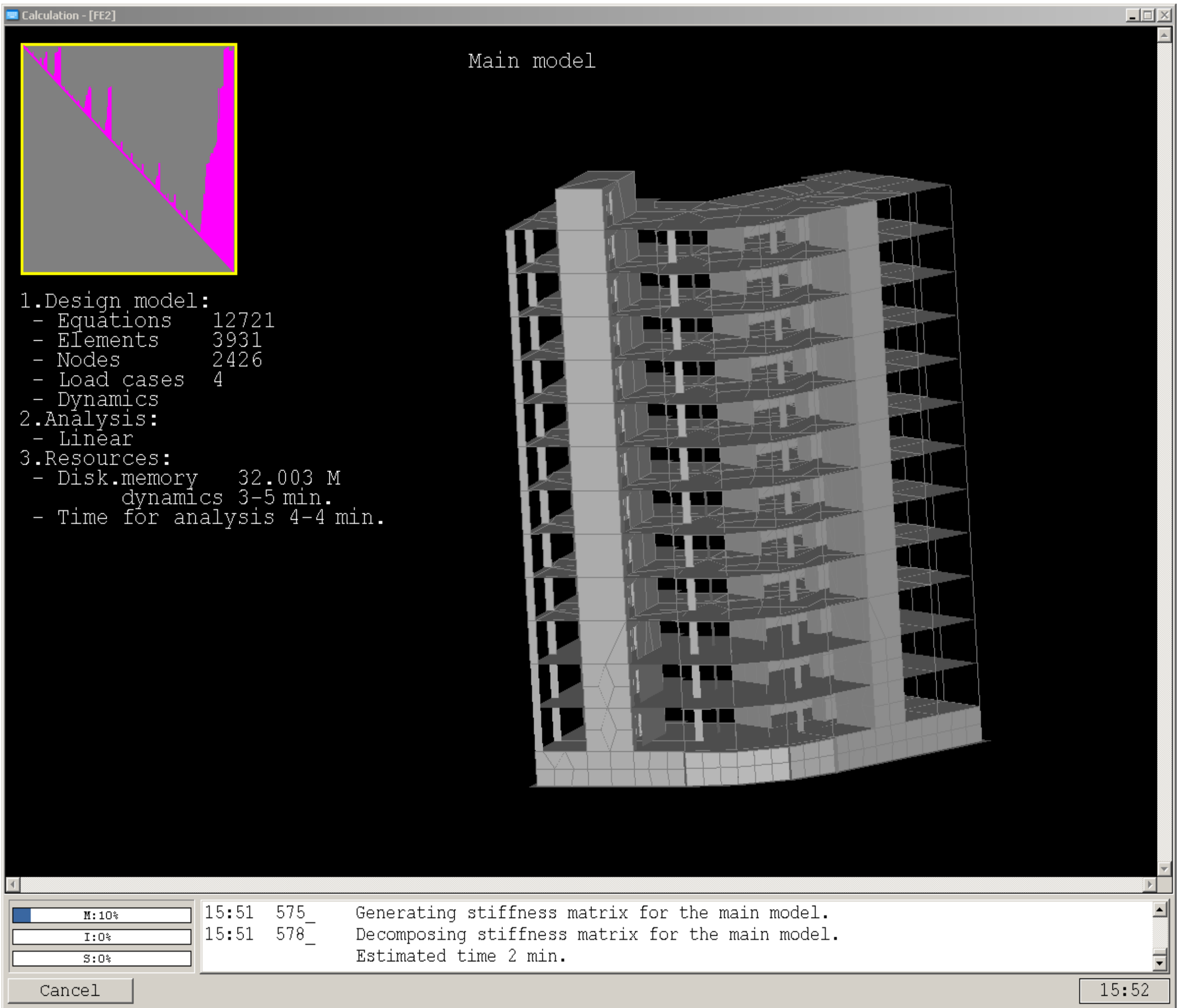


Figure 1.20.3 **Finite element analysis** dialog box

*When dual analysis is defined, the solver is launched two times. To terminate the analysis, click **Cancel**. Report for FEA is saved to the file report_fe1.txt (for dual analysis, the file report_fe2.txt is also generated). After FEA, the program checks sections of RC elements for obtained forces. The errors are displayed in the box.*

Step 21. Evaluation of FEA results

To evaluate results of FEA:

- To evaluate FEA results, on the VIEW menu, click **FEA results** (button on the toolbar).

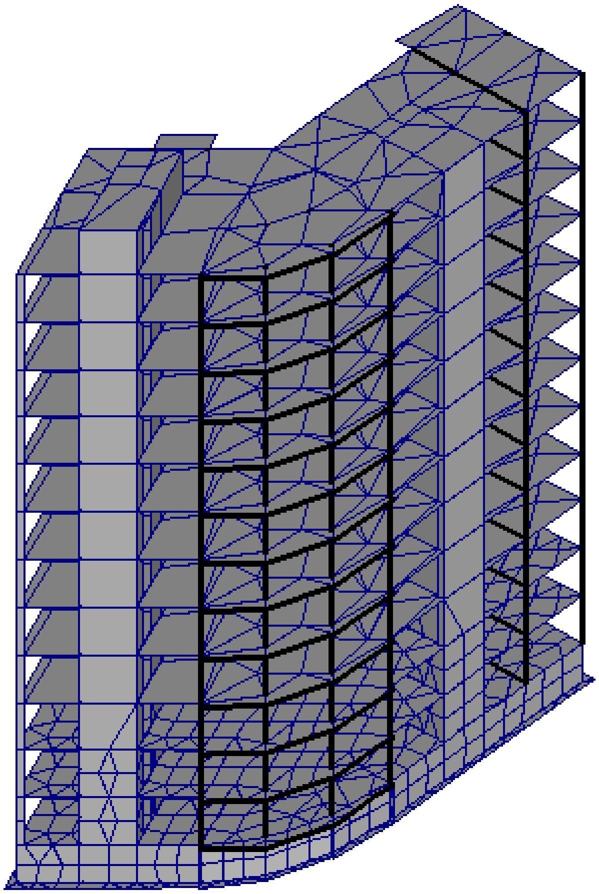


Figure 1.21.1 FEA results

*To print any image from the document window, on the FILE menu, click **Print**. For print preview the document, use the appropriate command in the same menu.*

To display deformed shape and contour plots of displacements:

- Make sure that dead load case is selected as current one (button on the toolbar should be active).
- To display deformed shape, on the RESULTS menu, click **Deformed shape** (button on the toolbar).
- To display contour plots of displacements along the Z-axis, on the RESULTS menu, point to **Displacement contour plots** and click **Z** (button on the toolbar).

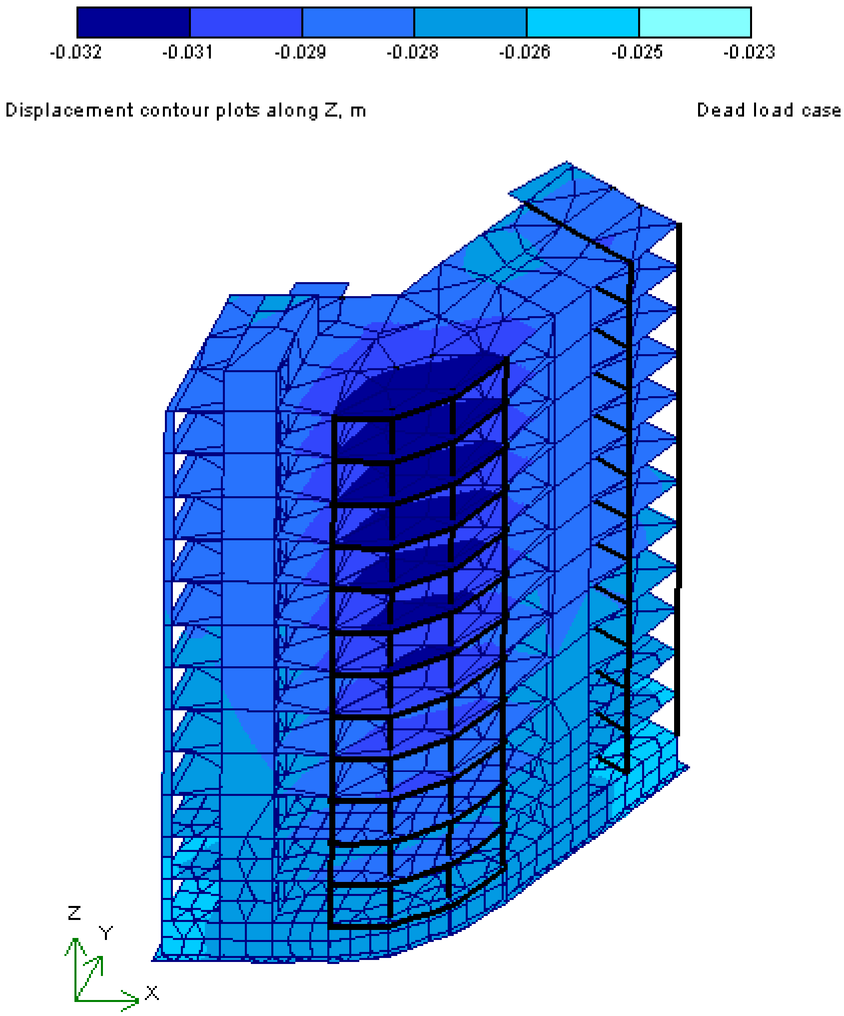


Figure 1.21.2 Displacements contour plots on deformed shape

- To assign the wind load case along the 2nd direction (135 degrees to the X-axis of the plan) as the current load case, on the LOAD CASE menu, click **2nd Wind** (button on the toolbar).
- To present contour plots of displacements along the Y-axis, on the RESULTS menu, point to **Displacement contour plots** and click **Y** (buton on the toolbar).
- To present projection of the right side view, on the VIEW menu, point to **Projection** and click **YOZ Right side view** (buton on the toolbar).
- To return the model to initial projection (axonometry), on the VIEW menu, point to **Projection** and click **Axonometric** (button on the toolbar).
- To display initial model, on the RESULTS menu, click **Initial model** (button on the toolbar).
- To hide the contour pots of displacements along the Y-axis, on the RESULTS menu, point to **Displacement contour plots** and click **Y** once again - button on the toolbar should become inactive.

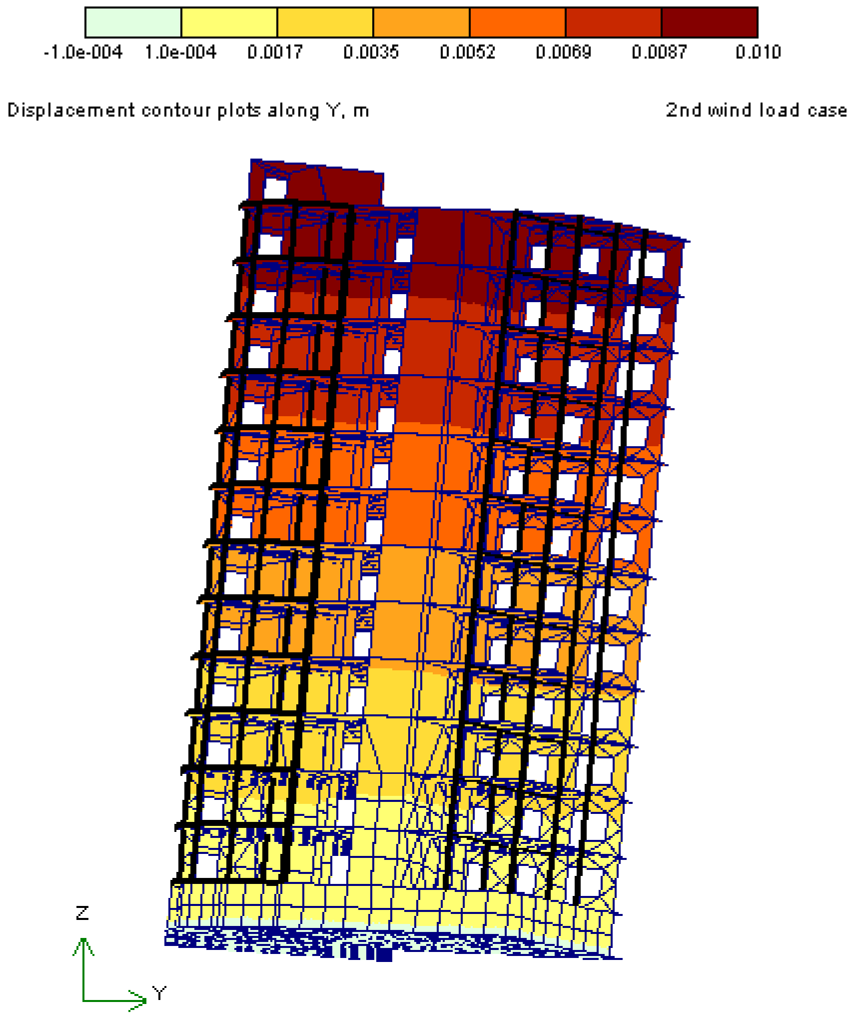


Figure 1.21.3 Displacements contour plots on deformed shape (projection of the right view)

<u>To display contour plots of forces in mat foundation:</u>

- To assign the dead load case as the current one, on the LOAD CASE menu, click **Dead** (button on the toolbar).
- To select the mat foundation on the model, on the SELECT menu, point to **Structural elements (STE)** and click **Mat foundation** (button on the toolbar). Selected finite elements of mat foundation will be coloured red on the model.
- To display on the model only selected elements, on the VIEW menu, click **Fragmentation** (button on the toolbar).
- To display contour plots Mx, on the RESULTS menu, point to **Contour plots of stresses and forces** and click **Mx** (button on the toolbar).
- To display projection of the top view, on the VIEW menu, point to **Projection** and click **XOY Top view** (button on the toolbar).
- To return the model to initial projection (axonometry), on the VIEW menu, point to **Projection** and click **Axonometric** (button on the toolbar).
- To restore initial model, on the VIEW menu, click **Restore model** (button on the toolbar).

- To hide the contour pots Mx, on the RESULTS menu, point to **Contour plots of stresses and forces** and click **Mx** once again - button on the toolbar should become inactive.

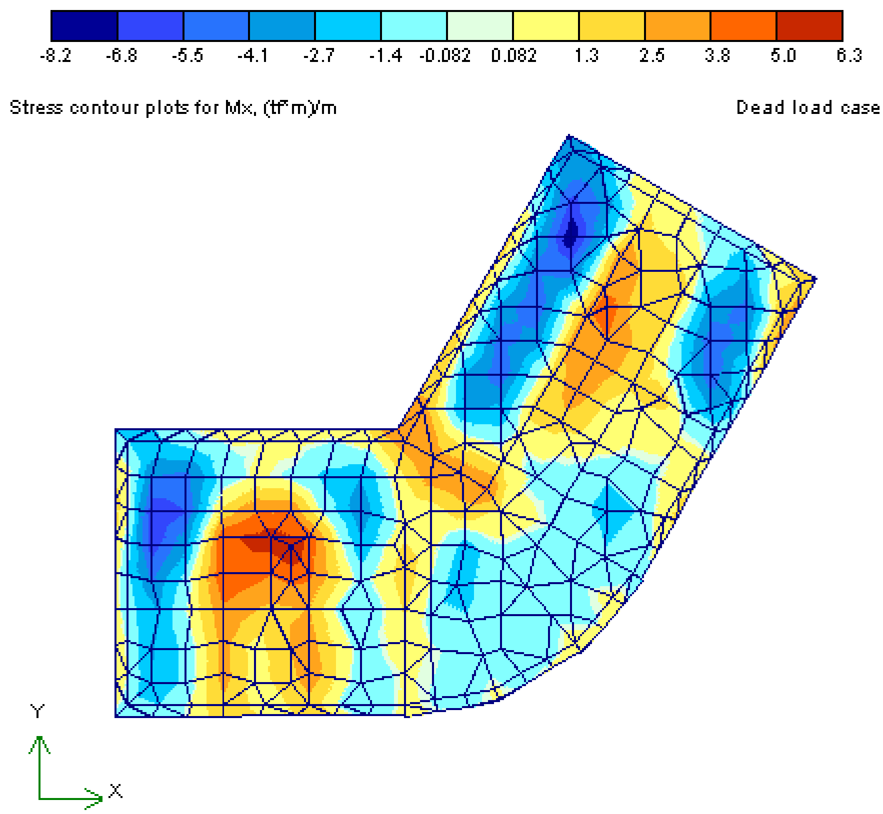
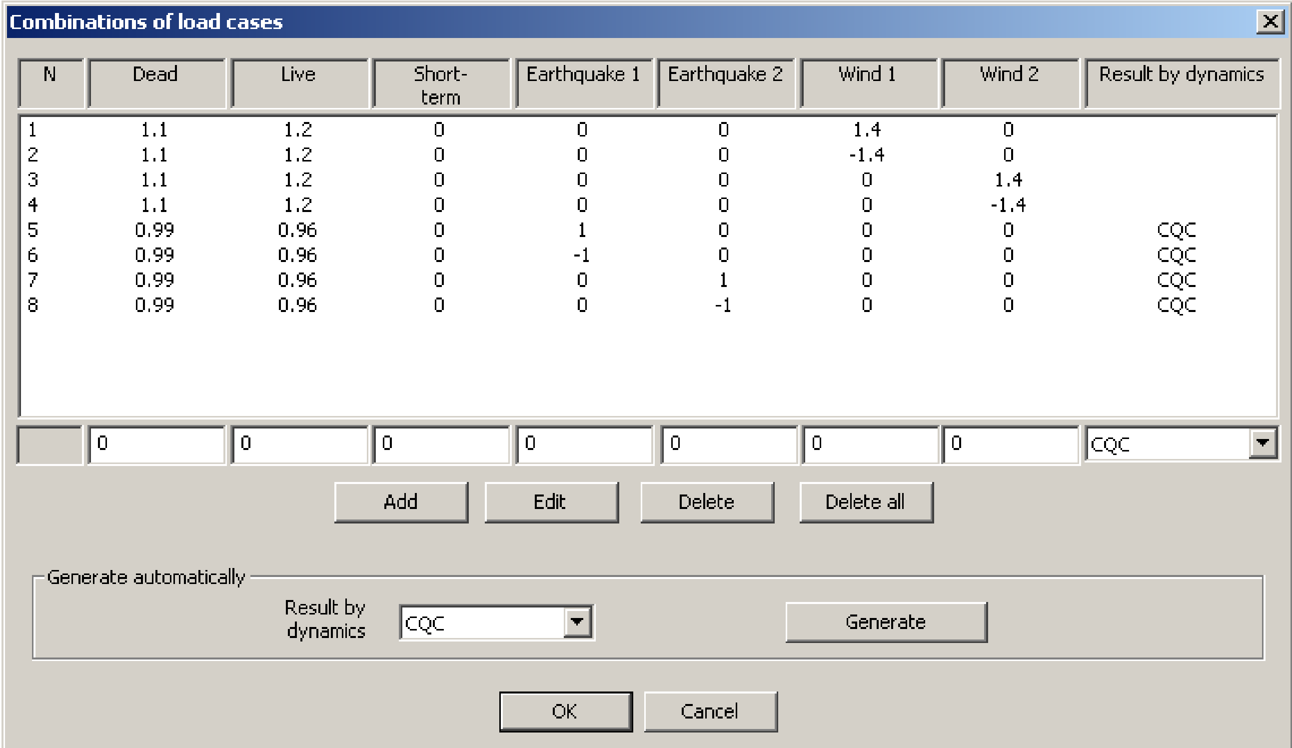


Figure 1.21.4 Contour plots of forces on deformed shape (projection of the right view)

<u>To create design combinations of load cases according to FEA results:</u>

- On the LOAD CASE menu, click **Create combination of load cases** (button on the toolbar).
- In the **Combinations of load cases** dialog box (see Fig.1.21.5), define the following parameters:
  - make sure that under **Generate automatically** – option **CQC** is selected for **Results by dynamics**;
  - to create a set of combinations, click **Generate.**

Combinations of load cases No.1...No.8 will be defined in the table of the dialog box.



N	Dead	Live	Short-term	Earthquake 1	Earthquake 2	Wind 1	Wind 2	Result by dynamics
1	1.1	1.2	0	0	0	1.4	0	
2	1.1	1.2	0	0	0	-1.4	0	
3	1.1	1.2	0	0	0	0	1.4	
4	1.1	1.2	0	0	0	0	-1.4	
5	0.99	0.96	0	1	0	0	0	CQC
6	0.99	0.96	0	-1	0	0	0	CQC
7	0.99	0.96	0	0	1	0	0	CQC
8	0.99	0.96	0	0	-1	0	0	CQC
	0	0	0	0	0	0	0	CQC

Figure 1.21.5 **Create combinations of load cases** dialog box

A set of combinations generated automatically is organized according to the specified building code, load cases, load factors, coefficients for load cases and safety factor for purpose of structure. In this case, sign variability of earthquake and wind load cases are considered. Pairs of differently directed earthquake and wind load cases could not act simultaneously.

The following variants may be generated as total results for earthquake load cases:
— SRSS ;
— CQC ;
*— Main shape for load case * coefficient(SRSS);*
*— Main shape for load case * coefficient(CQC).*
Mode shape with the greatest modal mass for the load case is considered as the main shape for the earthquake load case.

➪ To create one more (arbitrary) combination of load cases, in the edit boxes under the table specify coefficients to load cases (see Fig.1.21.6):

- for dead load case 1.1;
- for live load case 1.2;
- for earthquake load case along the 2nd direction 0.9;
- in the **Result by dynamics** lit box, select **Main shape*k(CQC)**;
- click **Add**.

Combination of load cases No.9 will be added to the table.

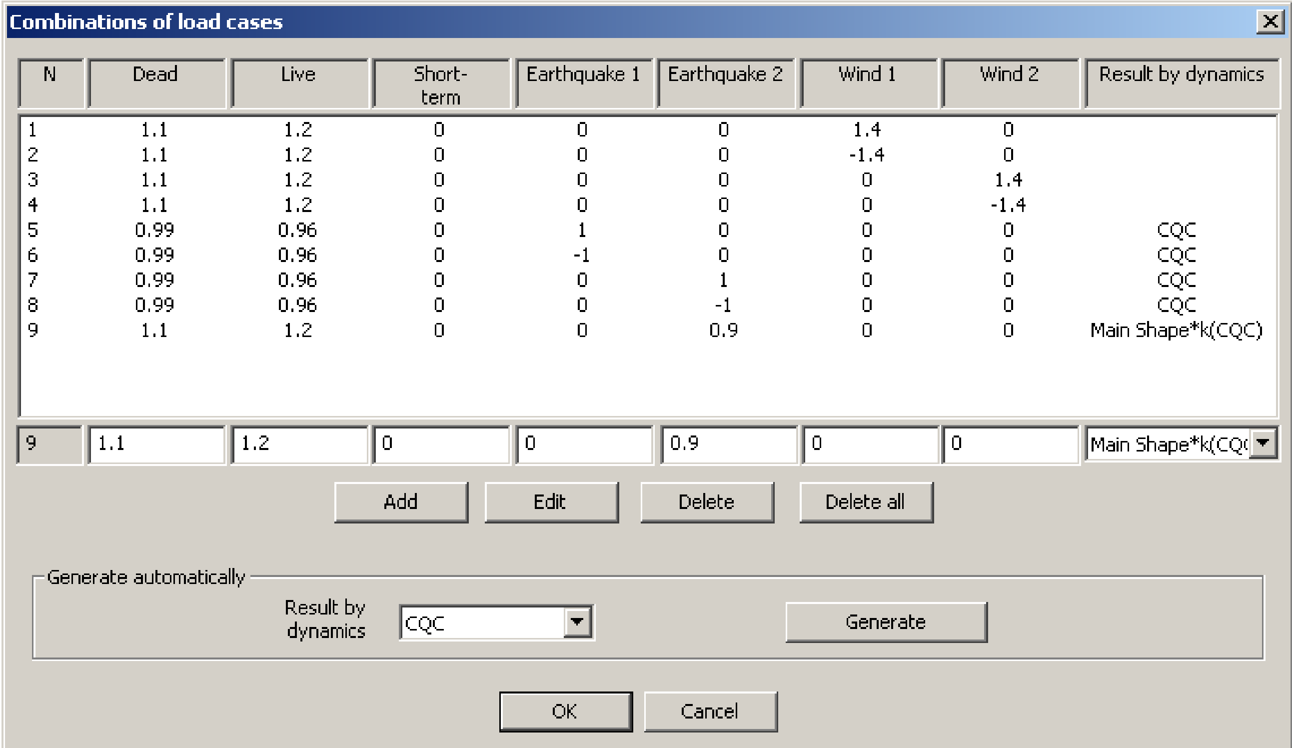


Figure 1.21.6 **Create combinations of load cases** dialog box

➪ Click **OK**.

<u>To display deformed shape and displacements contour plots for the specified combination of load cases:</u>

➪ On the LOAD CASE menu, click **Select combination of load cases** (button on the toolbar).

➪ In the **Select combinations of load cases** dialog box (see Fig.1.21.7), select combination of load cases No.9 from the list.

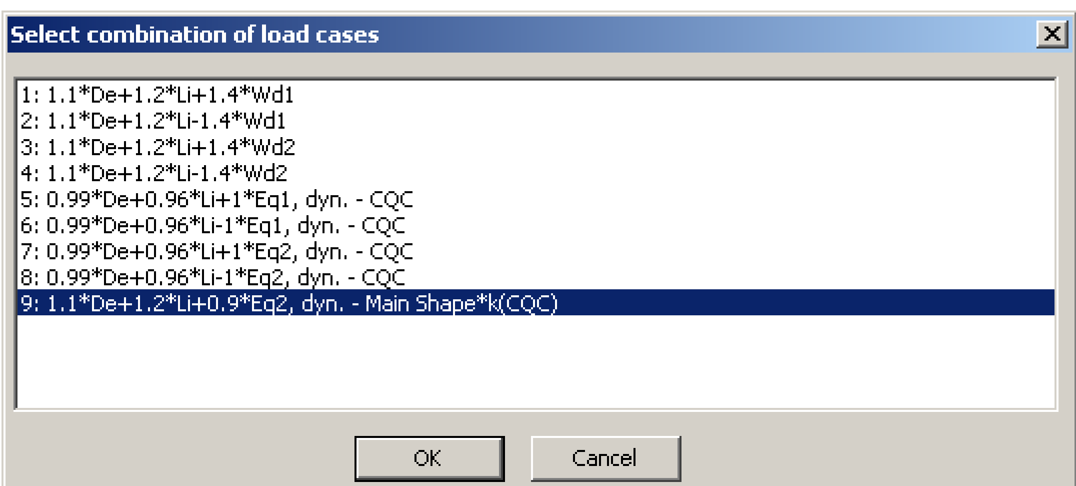


Figure 1.21.7 **Select combinations of load cases** dialog box

➪ Click **OK**.

When you change the combination of load cases, it might be slight delay in presentation (redrawing) of the image because it is necessary to recalculate data. Name of combination of load cases is presented as short formula above the model at the right.

- To display deformed shape, on the RESULTS menu, click **Deformed shape** (button on the toolbar).
- To display contour plots of displacements along the Z-axis, on the RESULTS menu, point to **Displacement contour plots** and click **Z** (button on the toolbar).
- Compare range of values for displacements and the deformed shape for the specified combination of load cases and for the dead load case (see Fig.1.21.2).
- To display initial model, on the RESULTS menu, click **Initial model** (button on the toolbar).
- To hide the contour pots of displacements along the Z-axis, on the RESULTS menu, point to **Displacement contour plots** and click **Z** once again - button on the toolbar should become inactive.

It is possible to select combinations of load cases to display analysis results on the model, to review information about certain nodes and elements, to generate report file.

<u>To display diagrams of forces in columns and beams:</u>

- To select on the model elements of storey No.2, on the SELECT menu, click **Storeys** (button on the toolbar).
- In the **Storeys** dialog box (see Fig.1.21.8), define the following parameters:
  - from storey No.2;
  - to storey No.2.
  - click **OK**.

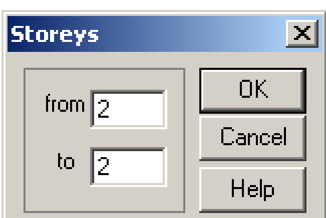


Figure 1.21.8 **Storeys** dialog box

Selected finite elements of storey No.2 will be coloured red on the model.

- To display on the model only selected elements, on the VIEW menu, click **Fragmentation** (button on the toolbar).
- To select the slab on the model, on the SELECT menu, point to **Structural elements (STE)** and click **Slab** (button on the toolbar). Selected finite elements of slab will be coloured red on the model.
- To display on the model only elements that remain unselected, on the VIEW menu, click **Inverse Fragmentation** (button on the toolbar).
- To assign the dead load case as the current one, on the LOAD CASE menu, click **Dead** (button on the toolbar).

- To display diagrams My, on the RESULTS menu, point to **Force diagrams** and click **My** (button on the toolbar).
- To enlarge the image, on the VIEW menu, point to **Image** and click **Zoom to selection** (button on the toolbar) or just rotate the wheel button (on the mouse pointer) forward.
- To display diagram values, on the RESULTS menu, point to **Force diagrams** and click **Diagram ordinates** (button on the toolbar).
- To hide the diagram values, simply click the **Diagram ordinates** button once again – the button should become inactive.
- To hide the force diagrams My, on the RESULTS menu, point to **Force diagrams** and click **My** once again - button on the toolbar should become inactive.
- To restore normal size of the model, on the VIEW menu, point to **Image** and click **Fit in window** (button) or just rotate the wheel button (on the mouse pointer) backward.
- To restore initial model, on the VIEW menu, click **Restore model** (button on the toolbar).

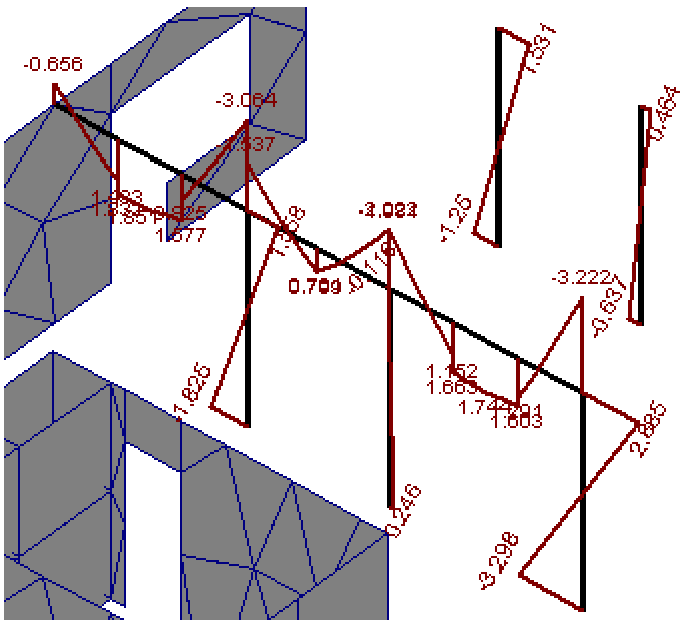
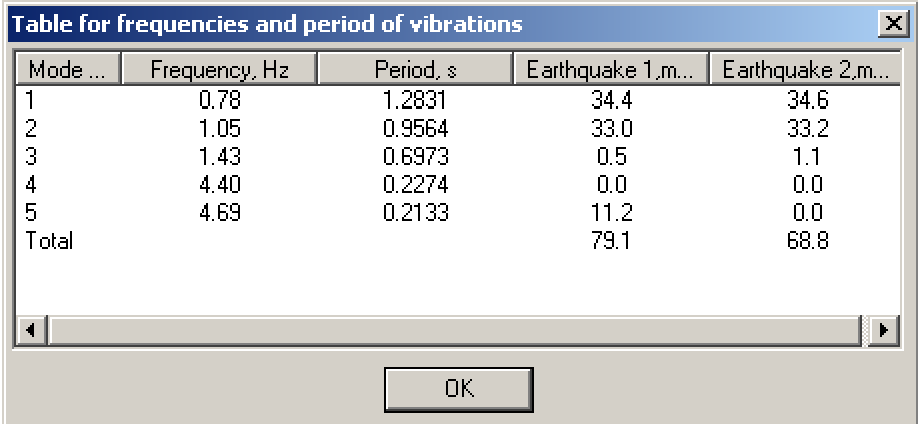


Figure 1.21.9 Force diagrams on deformed shape (fragment)

<u>To display results of modal analysis:</u>

- To display results of modal analysis, on the RESULTS menu, point to **Modal analysis** and click **Frequencies and periods of vibration** (button on the toolbar).
- In the **Table for frequencies and period of vibrations** dialog box (see Fig.1.21.10), review these tables.
- Click **OK**.



Mode ...	Frequency, Hz	Period, s	Earthquake 1,m...	Earthquake 2,m...
1	0.78	1.2831	34.4	34.6
2	1.05	0.9564	33.0	33.2
3	1.43	0.6973	0.5	1.1
4	4.40	0.2274	0.0	0.0
5	4.69	0.2133	11.2	0.0
Total			79.1	68.8

Figure 1.21.10 **Table for frequencies and period of vibrations** dialog box

- To display animation of vibrations, on the RESULTS menu, point to **Modal analysis** and click **Animate mode shapes** (button on the toolbar).
- In the **Animation** dialog box (see Fig.1.21.11), do the following:
  - select mode 2 from the list;
  - click **Start**;

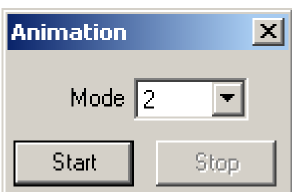


Figure 1.21.11 **Animation** dialog box

You will see animation of natural vibrations of design model by mode shape No.2.

  - click **Stop**;
- To close the **Animation** dialog box, click the **Close** button .

When you assign one of the earthquake load cases as the current one (with the LOAD CASE menu), it is necessary to define the mode shape. Modal contribution of every mode shape may be determined by values of masses indicated in percentage for every mode shape in the ***Table for frequencies and periods of vibrations*** *dialog box.*

<u>To display information about elements and nodes:</u>

- To display information about elements and nodes, on the RESULTS menu, click **Information about element or node** (button on the toolbar).
- In the dialog box (see Fig.1.21.12), clear the **Elements** check box.

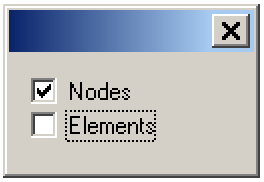


Figure 1.21.12 Dialog box to define selection option

- Specify node of slab of the upper storey at the intersection between lines I and A according to the plan of the structure.
- In the **Node No.818** dialog box (see Fig.1.21.13), do the following:
  - from the **Load case** list, select **Earthquake 2**;
  - from the **Mode** list, select **CQC**;

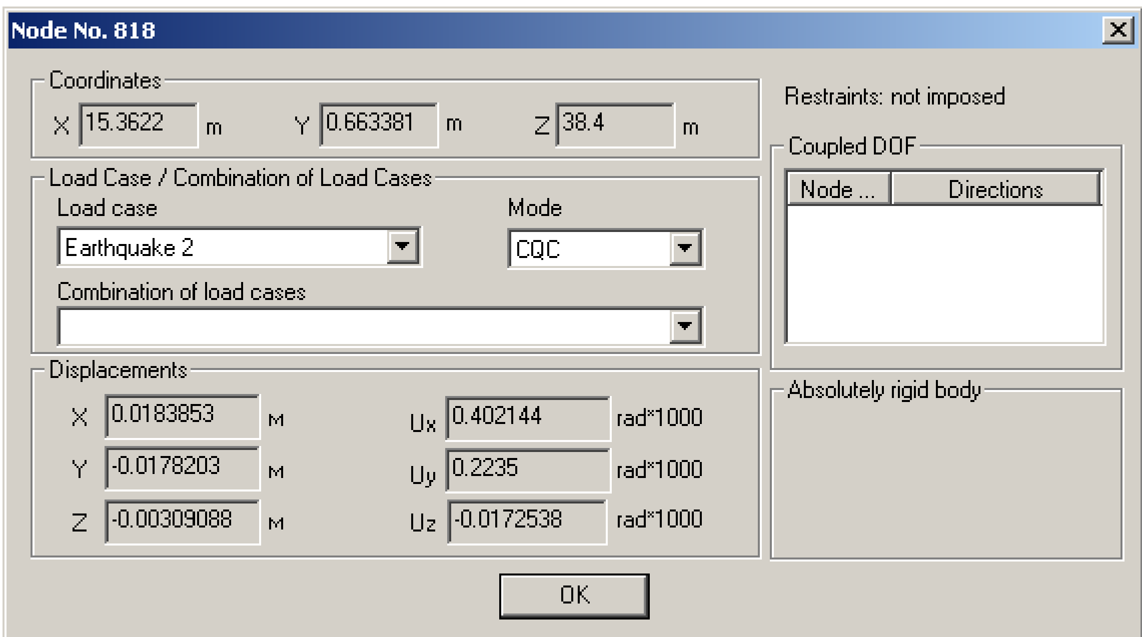


Figure 1.21.13 **Node No.818** dialog box

In the dialog box there are coordinates and component displacements of this node. When you select an earthquake load case, it is possible to select the combination of mode shapes. For example, CQC - total quadratic combination of displacements that correspond to mode shapes (for dependent mode shapes).

- Click **OK**.

- In the still open dialog box ((see Fig.1.21.12), do the following:
  - clear the **Nodes** check box;
  - select the **Elements** check box.

- Specify wall of storey No.2 at the intersection between lines 1 and A according to the plan of the structure.
- In the **Elements** dialog box, clarify selection – select **Wall No.2_1** from the list.
- In the **Wall No.2_1** dialog box (see Fig.1.21.14) review total loads calculated at the wall bottom level.

➪ Click **OK**.

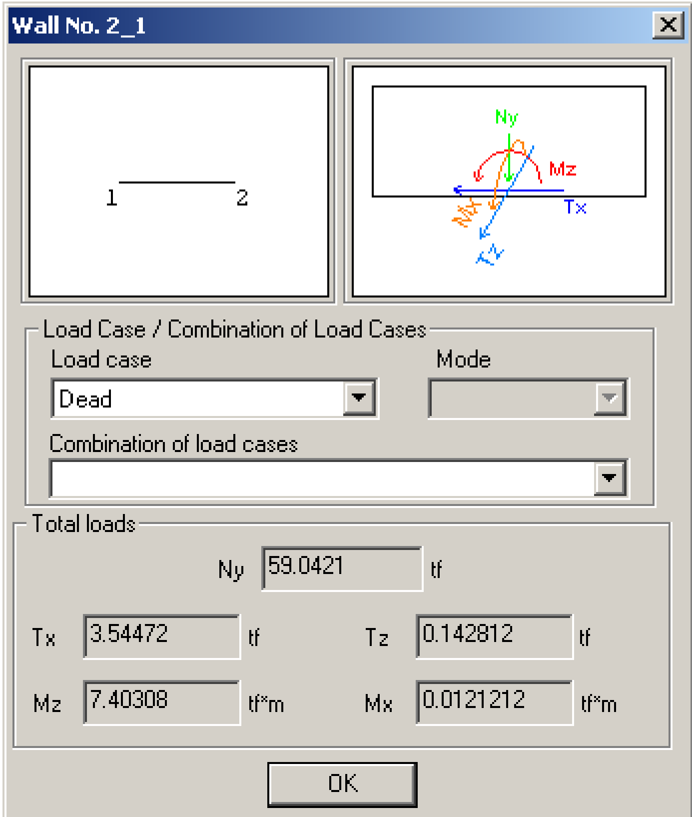


Figure 1.21.14 **Wall No.2_1** dialog box

In the dialog box with information about element, you will see normative values of displacements, forces and loads with the load factor equal to 1 (Yf=1). To review values of displacements, forces and loads with account of specified load factors (Yf>1) and combination coefficients, select one of previously specified load cases.

To generate and save the report:

➪ To generate the report, on the RESULTS menu, point to **Analysis results** and click **Reportin RTF format** (button  on the toolbar).

In this case the report is generated according to FEA results.

➪ In the **Report for FEA results** dialog box (see Fig.1.21.15), do the following:

- under **Input data**, select the **Load coefficients** check box;
- under **Analysis results**, select the **Drifts of storeys** check box;
- for the **Columns** option, define value 2(1-14);
- for the **Beams** option, define value 2(1-4);
- for the **Walls** option, define value 2(1-26);

➪ Click **OK**.

➪ In the **Save as** dialog box, indicate the file *Model1_fea.rtf* in **Notes** directory in the base MONOMAKH-SAPR directory and click **Save**.

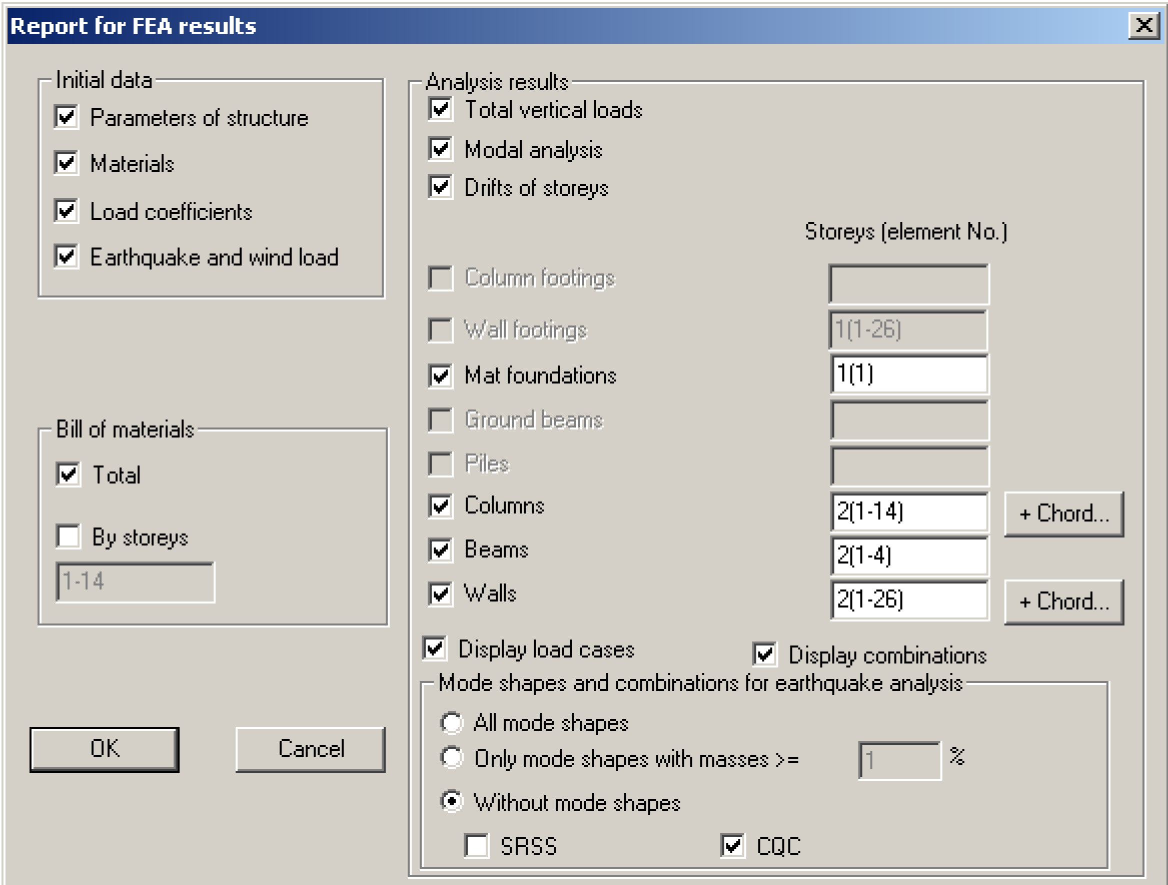


Figure 1.21.15 **Report for FEA results** dialog box

<u>To review the report:</u>

➪ On the taskbar, click the **Start** button and point to **Programs** / MS Word.

➪ To open the report file in MS Word, on the FILE menu, click **Open**.

➪ In the standard **Open** dialog box, do the following:

- in the **Files of type list**, select *Rich text format* (*.rtf);
- open the **Notes** directory in the base MONOMAKH-SAPR directory;
- open file *Model1_fea.rtf*.

Report file contains several tables that may be reviewed and printed. The file contains input data and FEA results.

➪ Close the report file in MS Word.

➪ To return to the main view, on the VIEW menu, click **Main view** (button on the toolbar).

Step 22. Export to design modules of MONOMAKH-SAPR

- To export results of FEA to design modules of MONOMAKH-SAPR software, on the RESULTS menu, click **Export data to MONOMAKH-SAPR modules** (button on the toolbar).
- In the **Export data to MONOMAKH-SAPR modules** dialog box (see Fig.1.22.1), define the following parameters:
  - as columns export walls of length <= 1 m (by default, the value <= 3 m is defined);
  - other parameters remain by default;
  - click **OK**.

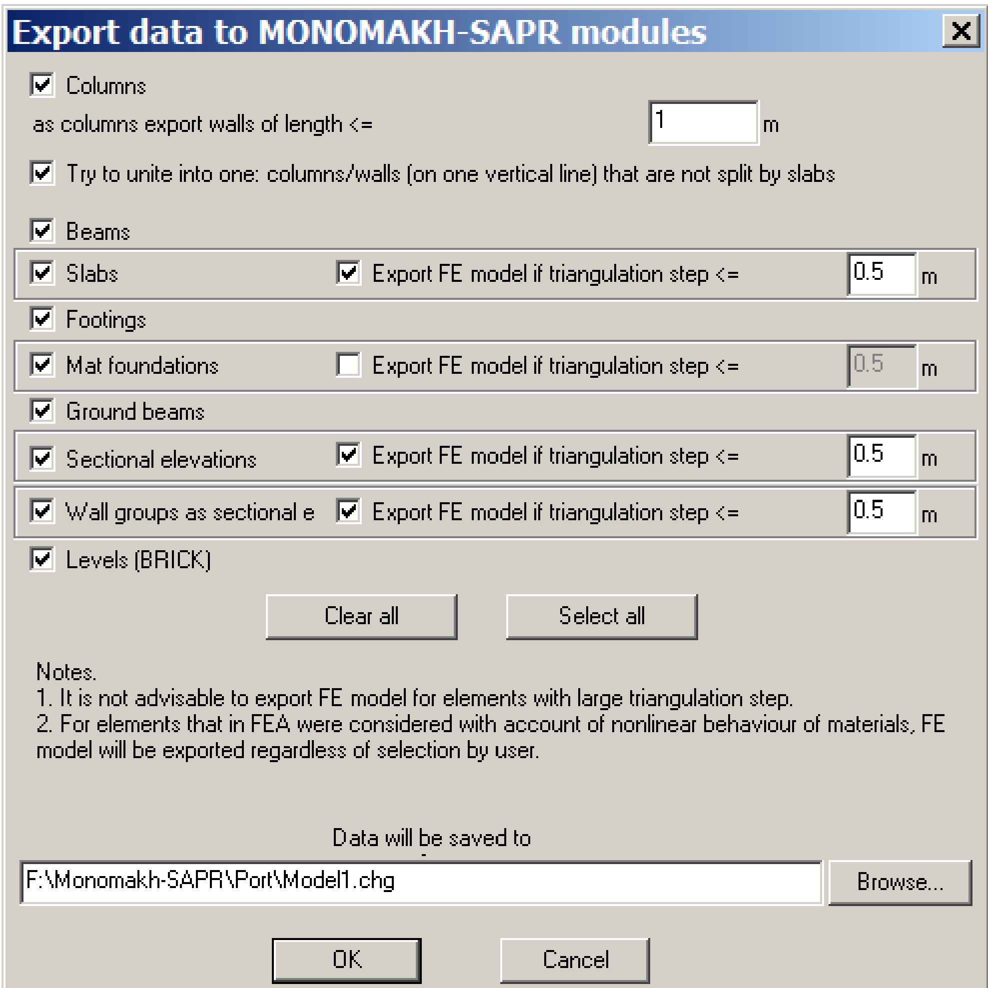


Figure 1.22.1 **Export data to MONOMAKH-SAPR modules** dialog box

Directory named as the problem *Model1.chg* will be created in the **Port** directory of base MONOMAKH-SAPR directory. This directory will contain files with data about structural elements.

*Limitation on triangulation step is defined for elements that will be exported to design modules as **FE model** because in this case the model is exported to design modules with ready-to-use solution (division into finite elements accepted in BUILDING module; displacements, stresses and forces computed in BUILDING module).*

Step 23. Export to LIRA-SAPR software

To export data to LIRA-SAPR software:

During the export procedure, model of the structure is automatically transformed to finite element model.

- To carry out wind analysis according to SNIP 2.01.07-85 and to create separate dynamic load case in LIRA-SAPR, change option for wind load. On the MODEL menu, point to **Add** and click **Earthquake and wind loads** (button on the toolbar).
- In the **Earthquake and wind loads** dialog box, do the following:
  - define parameters for default building code (SNIP 2.01.07-85), click **Parameters**;
  - in the **SNIP 2.01.07-85** dialog box (see Fig.1.23.1), define the following parameters:
    - under **Account of pulsation component**, click **Separate dynamic load case**;
    - other parameters remain by default;
    - click **OK**;
- Click **OK**.

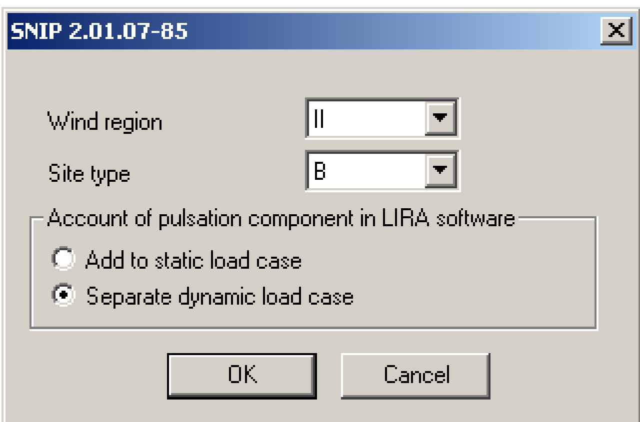


Figure 1.23.1 **SNIP 2.01.07-85** dialog box

When you edit input data, results of preliminary analysis and FEA are cancelled. To export to LIRA-SAPR program, it is necessary to perform preliminary analysis of the whole structure.

- To make preliminary analysis of the whole structure, on the ANALYSIS menu, click **Analysis of whole structure** (button on the toolbar).
- On the RESULTS menu, click **Export to LIRA-SAPR** (button on the toolbar).
- In the **Export to LIRA-SAPR software** dialog box (see Fig.1.23.2), define the following parameters:
  - select the 4-node FE (slabs) check box;
  - select the 4-node FE (walls) check box;
  - select the 4-node FE (mat foundations) check box;
  - triangulation step 0.5m for slabs, walls and mat foundations;
  - other parameters remain by default.
- Click **OK**.

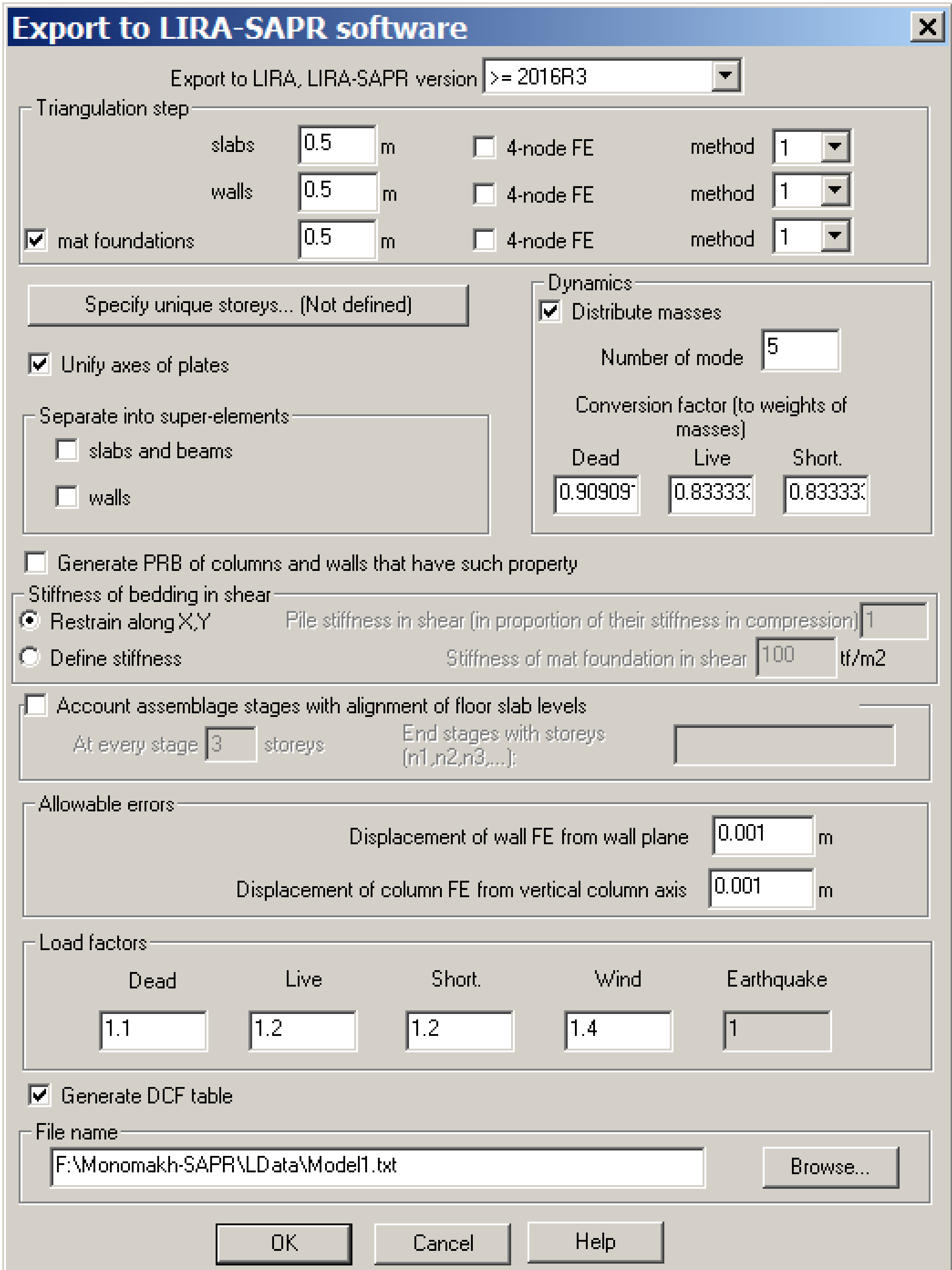


Figure 1.23.2 **Export to LIRA-SAPR software** dialog box

File named as the problem *Model1.txt* will be created in the **LData** directory of base MONOMAKH-SAPR directory.

To open import file in LIRA-SAPR program:

- On the taskbar, click the **Start** button, and then point to **Programs/LIRA SAPR/LIRA-SAPR 2017** and then click **LIRA-SAPR 2017**.
- On the FILE menu, click **Import problem**.

- In the **Import** dialog box, do the following:
  - in the **Files of type list**, select *Text files* (*.txt);
  - open the **LData** directory in the base MONOMAKH-SAPR directory;
  - open file *Model1.txt*.

Now it is possible to analyse the model.

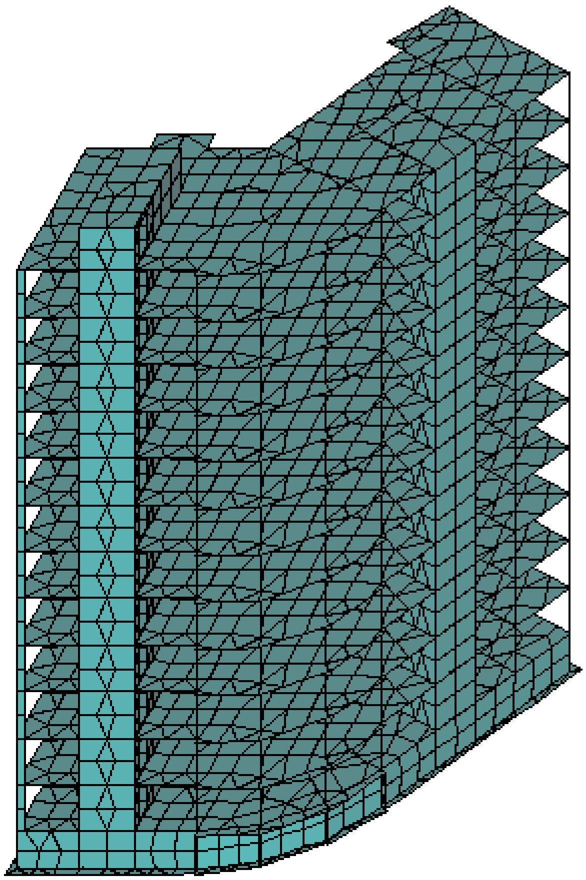


Figure 1.23.3 Design model in LIRA-SAPR software

- To make the model look as it is presented in Figure 1.23.3, on the OPTIONS menu, click **Flags of drawing**.
- In the **Display** dialog box, do the following:
  - click the **General** tab ;
  - select the **Shading** check box ;
  - clear the **Loads** check box;
  - click **Redraw** button .

Save the file as *Model1.lir*.

Step 24. Modifying the type of mat foundation

<u>To save the problem under new name:</u>

- To save the problem under new name, on the FILE menu, click **Save as**.
- In the **Save as** dialog box, define:
  - file name *Model2.chg*;
- Click **Save**.

The problem file *Model2.chg* will be created on the disk in the base MONOMAKH-SAPR directory,

*To rename the new object, on the FILE menu, click **Rename**.*

<u>To modify the type of mat foundation:</u>

- To define storey No.1 as current one, click the button on the STOREYS toolbar.
- On the MODEL menu, point to **Select** and click **Select elements by criteria** (button on the toolbar).
- In the **Select elements by criteria** dialog box, do the following:
  - on the **Mat foundations** tab, all parameters remain by default;
  - click **Apply** .
- Close the **Select elements by criteria** dialog box.

Selected mat foundation will be coloured red.

- On the MODEL menu, point to **Edit** and click **Element properties** (button on the toolbar).
- In the **Mat foundation No.1_1** dialog box (see Fig.1.24.1), do the following:
  - under the **Bedding**, select **Pile foundation**;
  - other parameters remain by default;
  - click **Apply** .

The type of mat foundation will be modified.

- On the MODEL menu, point to **Select** and click **Unselect all** (button on the toolbar).

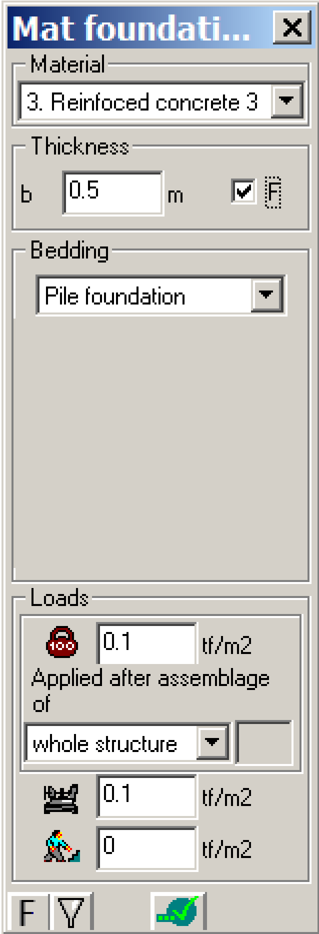
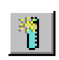


Figure 1.24.1 **Mat foundation No.1_1** dialog box (Element properties)

Step 25. Defining the pile group

<u>To define pile group:</u>

Define parameters and location of piles:

- On the MODEL menu, point to **Add** and click **Pile** (button  on the toolbar).
- In the **Add pile** dialog box (see Fig.1.25.1), define the following data:
  - stiffness - **Geometry**;
  - section shape - **Rectangle**;
  - section width b = 0.4 m;
  - section height h = 0.4 m;
  - pile length l = 5.5 m;
  - other parameters remain by default.

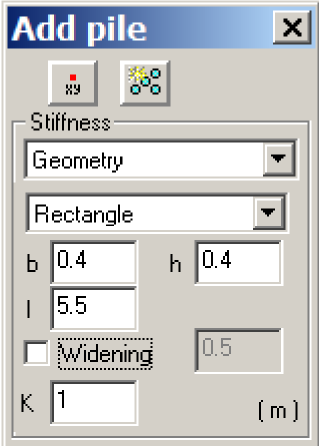
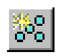


Figure 1.25.1 **Add pile** dialog box

*Reinforced concrete with modulus of elasticity E = 3000000 tf/m2 is considered as material for piles for the stiffness **Geometry**. In this case, stiffness of FE that simulates the pile will be computed according to the data on pile geometry and soil properties defined with the **General parameters of structure** commend (on the MODEL menu). If there is information about bearing capacity of piles, then it is preferable to select the **stiffness EF** or **Bearing capacity**. For example, if load on pile is 160 tf, the pile settlement is 4 cm, in this case pile stiffness is EF=160/0,04=4000 tf per 1 m of pile length.*

- In the **Add pile** dialog box, click the **Add pile group**  .
- In the **Arrangement of pile group** dialog box (see Fig.1.25.2), define the following data:
  - spacing of piles along the X-axis - X = 1.16 m;
  - spacing of piles along the Y-axis Y = 1 м;
  - number of pile spacings along the X-axis - 10;
  - number of pile spacings along the Y-axis - 11;
  - other parameters remain by default (**Regular** type of pile arrangement is selected by default);
  - click **Apply**.

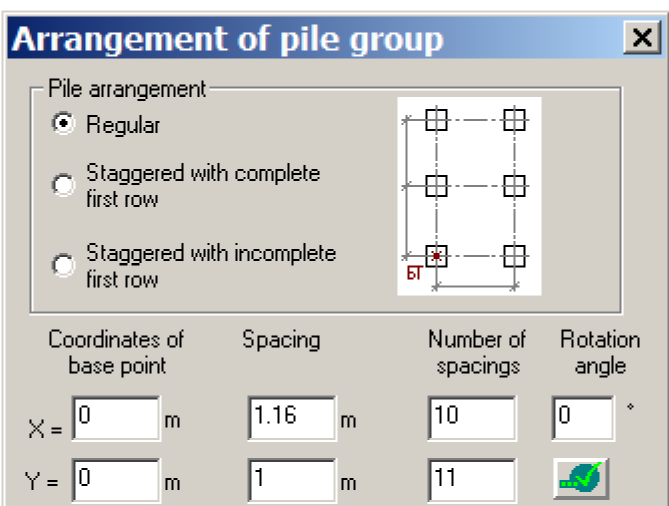


Figure 1.25.2 **Arrangement of pile group** dialog box

The pile group will be defined at the plan in axes 1…4, A…D (see Fig.1.25.3).

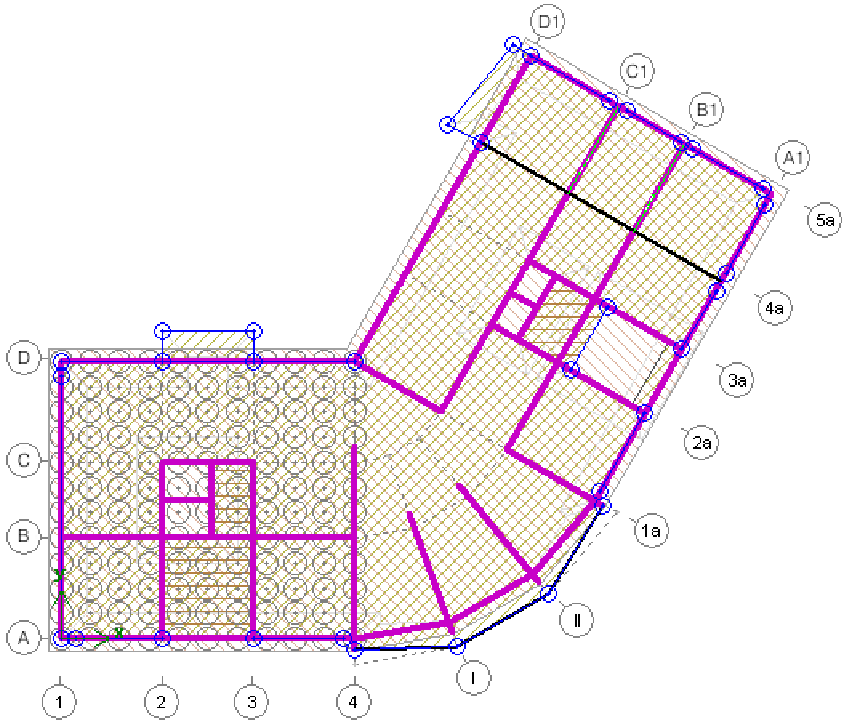


Figure 1.25.3 Pile group on plan

Piles are always presented on the model as circles regardless of section shape defined for the piles.

- On the MODEL menu, point to **Coordinate system** and click **Fix**.
- Select on the model the node where axes 1a and A1 intersect.
- Select on the model the node where axes 1a and D1 intersect.

Y-axis of coordinate system should coincide with the axis 1a.

- In the **Arrangement of pile group** dialog box, define the following data:
  - spacing of piles along the X-axis - X = 1 m;
  - spacing of piles along the Y-axis Y = 1 м;
  - number of pile spacings along the X-axis - 14;
  - number of pile spacings along the Y-axis - 11;
  - other parameters remain by default;
  - click **Apply**.

The pile group will be defined at the plan in axes 1…5a, A1…D1. Note that two piles are defined at the intersection of axes 4 and D. One of them should be deleted.

<u>To delete the pile:</u>

- On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).
- When this mode is active, select the pile on the model.
- In the **Select element** dialog box (see Fig.1.25.4), clarify the selection:

- select from the list – Pile No.1_298.

➪ Click **OK**.

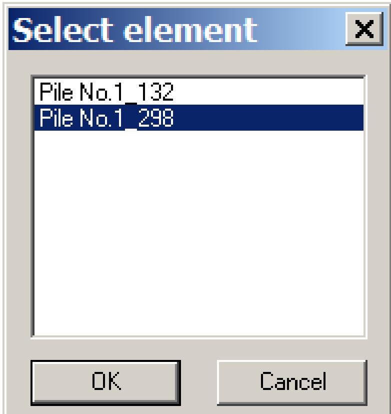


Figure 1.25.4 **Select element** dialog box

Selected pile will be coloured red on the model.

➪ On the MODEL menu, click **Delete elements** (button on the toolbar).

<u>To copy piles:</u>

➪ On the MODEL menu, point to **Coordinate system** and click **Initial location**.

➪ On the MODEL menu, point to **Coordinate system** and click **Move and rotate** (button on the toolbar) and move the origin to the centre of polar grid at the intersection of axes 4 and D.

Select six piles starting from the pile at the intersection of axes 4 and A and then upwards on the axis 4:

➪ On the MODEL menu, point to **Select** and click **Select elements** (button on the toolbar).

➪ When this mode is active, with the single pointer (button on the toolbar) select the piles one by one.

Piles selected on the model will be coloured red.

➪ On the MODEL menu, point to **Edit** and click **Copy and Move** (button on the toolbar).

➪ In the **Copy and move** dialog box, do the following:

- select the **Rotate** tab (see Fig.1.25.5);
- define the rotation angle Fi = 10 degrees;
- N = 5;
- other parameters remain by default;
- click **Apply** .

Select the next three piles on the axis 4 near the C axis.

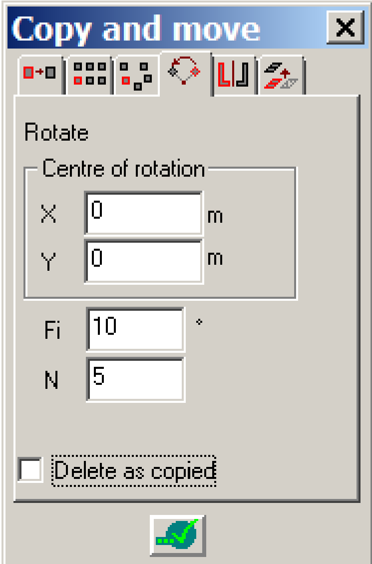


Figure 1.25.5 **Copy and move** dialog box (**Rotate** tab)

➪ In the **Copy and move** dialog box, define the following data:

- rotation angle Fi = 20 degrees;
- N = 2;
- click **Apply** .

Select the pile on the axis 4 between axes C and D:

➪ In the **Copy and move** dialog box, define the following data:

- rotation angle Fi = 30 degrees;
- N = 1;
- click **Apply** .

Select six pairs of piles near the bottom ring axis starting from axis 4 as it is presented in Fig.1.25.6.

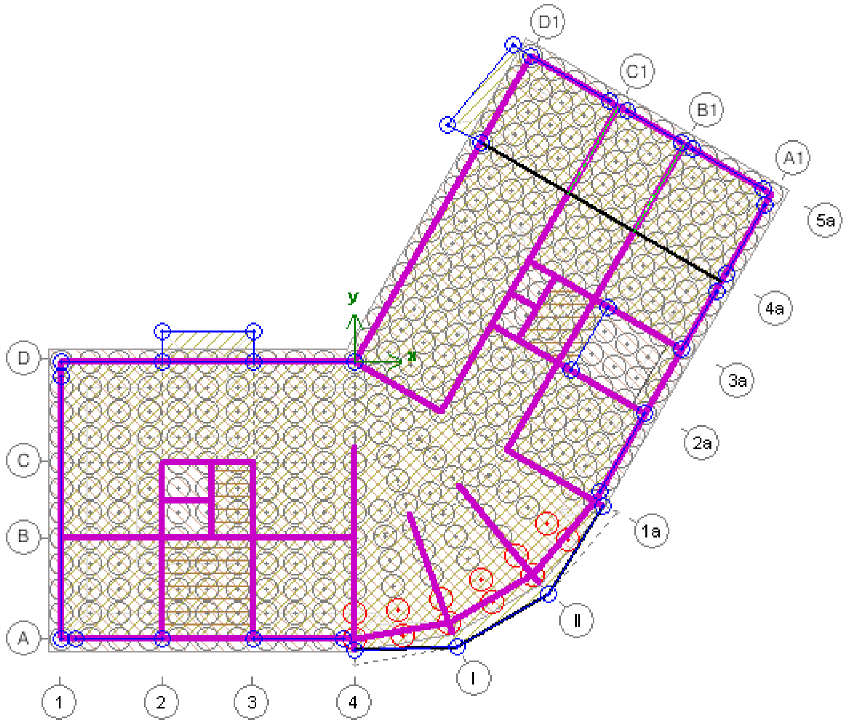
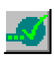
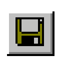


Figure 1.25.6 Selected piles

- In the **Copy and move** dialog box, define the following data:
  - rotation angle Fi = 5 degrees;
  - N = 1;
  - click **Apply**  .

Pile group will be defined.

 *In this example, piles were defined as pile groups and by copy procedure. Apart from these methods, piles may be defined by direct selection on the model or with coordinates for the single pile in the dialog box (in the same manner as you define the column).*

- On the FILE menu, click **Save** (button  on the toolbar).

To display notation of piles on the model:

- On the VIEW menu, click **Display options**.
- In the **Display options** dialog box, do the following:
  - select the **Numbers and parameters** tab;
  - select the **Piles: Numbers** check box;
  - click **Apply**  .
- Close the **Display options** dialog box.
- To do the same, you could use the **Piles: Numbers** button on VISUALIZATION toolbar.

Storey No. (always 1) and pile No. are displayed for every pile. To make the image clear, you could temporarily cancel presentation of elements (just click appropriate buttons – ***Columns***, – ***Beams***, – ***Walls***, – ***Partitions***, – ***Slabs***, – ***Loads***, – ***Footings***, – ***Mat foundations*** *on VISUALIZATION toolbar).*

- To hide numbers and parameters of piles, click the **Piles: Numbers** button on the toolbar. This button should become not active.

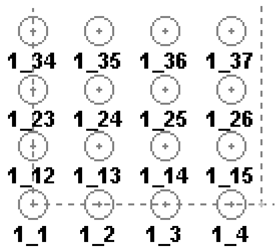


Figure 1.25.7 Notation of piles on the model

<u>To preview the model in 3D-view:</u>

- On the VIEW menu, point to **3D view** and click **Whole structure** (button on the toolbar).
- In the *3D view* mode, on the VIEW menu, point to **Projection** and click **Perspective** (button on the toolbar).
- To move the model image downwards, click the PAGE UP key several times.
- To move the image away, click the DOWN ARROW key several times.
- To rotate the model image about the viewer, click the SHIFT+LEFT ARROW key combination several times.
- To rotate the model image about its own vertical axis, use the CTRL+RIGHT ARROW or CTRL+LEFT ARROW key combinations (buttons and on the toolbar).
- To return to the main view, on the VIEW menu, click **Main view** (button on the toolbar).

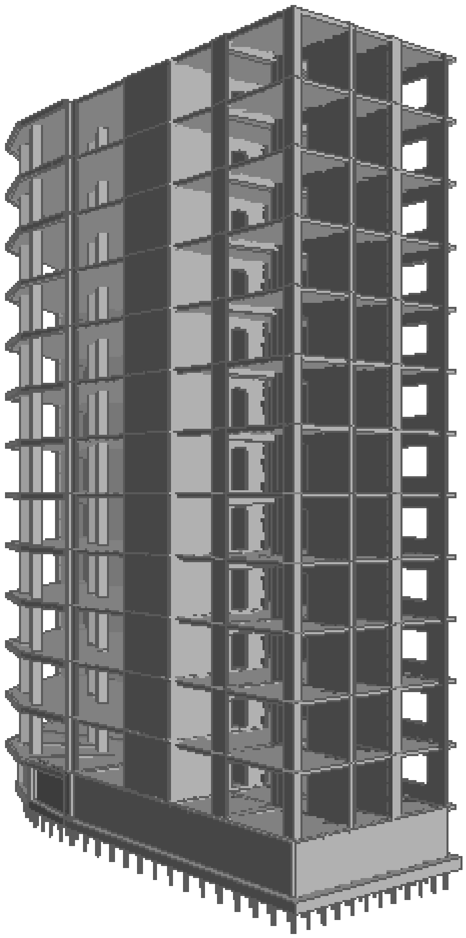


Figure 1.25.8 3D-view of the specified model (perspective view)

Step 26. Analysis of whole structure and FEA

<u>To carry out analysis of the whole structure:</u>

- On the ANALYSIS menu, click **Analysis of whole structure** (button on the toolbar).

At the stage of preliminary analysis, load on piles is not determined.

<u>To carry out FEA (finite element analysis):</u>

- On the ANALYSIS menu, click **Finite element analysis** (button on the toolbar).

When you saved file Model2.chg, parameters for FEA remain as they were defined and saved in the file Model1.chg (see step 20).

- In the **Finite element analysis** dialog box, define the following data:
  - to consider **increase stiffness of soil in certain load cases (dual analysis),** select the **Pile stiffness (EF)** check box (by default, increase factor for stiffness of the pile footing in order to compute displacements and forces from earthquake load is defined as 1 time);

- to compute the increase factor that would be the most appropriate for this model, click the button and **Pile stiffness (EF)** values will be computed automatically;
- other parameters remain by default.

➪ Click **OK**.

In the solver window you will see design model and analysis protocol. Wait for the analysis to be complete.

*To stop analysis procedure, click **Terminate** (button). In case of dual analysis, the solver will start two times. Report of FEA is saved to the file report_fe1.txt (in dual analysis, the program generates additional file report_fe2.txt). The program also checks sections of RC elements for obtained forces. If there are any errors, they will be displayed in the message window. To colour (in red) the certain element on the model, just click appropriate row in the list of warnings.*

<u>To save analysis results:</u>

*On the FILE menu, click **Save** (button on the toolbar). In this case, analysis results are also saved to the *.chg file.*

Step 27. Evaluate FEA results

<u>To present results of FEA:</u>

➪ On the VIEW menu, click **FEA results** (button on the toolbar).

<u>To present mosaic plot of pile stiffness:</u>

➪ In the *FEA results* view, on the SELECT menu, point to **Structural elements (StE)** and click **Mat foundation** (button on the toolbar).

➪ On the SELECT menu, point to **Structural elements (StE)** and click **Pile** (button on the toolbar).

Selected finite elements of mat foundation and finite elements of piles will be coloured red on the model.

➪ On the VIEW menu, click **Fragmentation** (button on the toolbar).

Finite elements that piles are simulated with are denoted on the model with small squares.

➪ On the VIEW menu, point to **Projection** and click **XOY top view** (button on the toolbar).

➪ On the RESULTS menu, click **Mosaic plot EF in piles** (button on the toolbar).

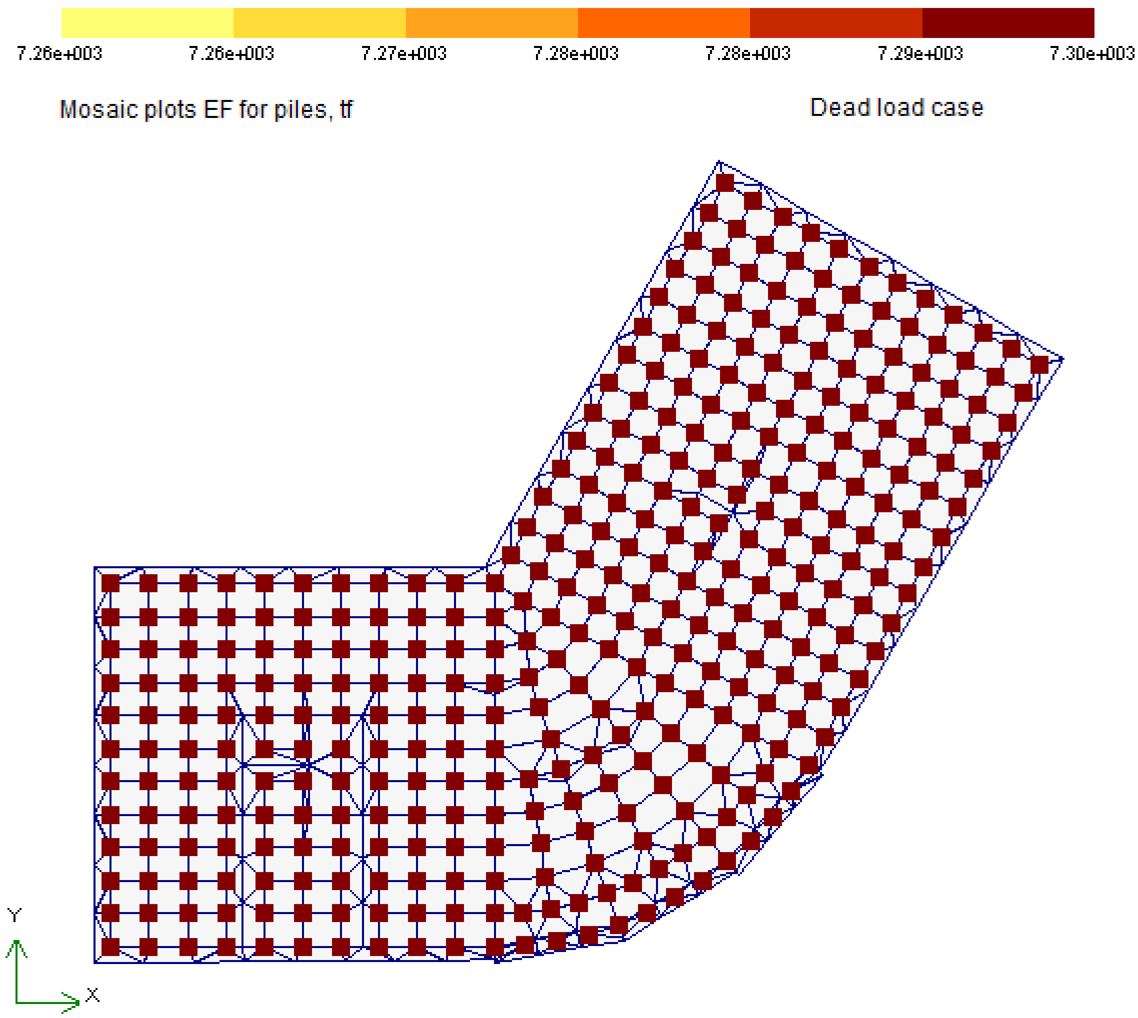


Figure 1.27.1 Mosaic plot EF in piles on the fragmented model (projection - top view)

<u>To present mosaic plot of forces in piles:</u>

- Make sure that dead load case is defined as the current one – button on the toolbar should be active.
- On the RESULTS menu, point to **Forces in piles** and click **Rz** (button on the toolbar).

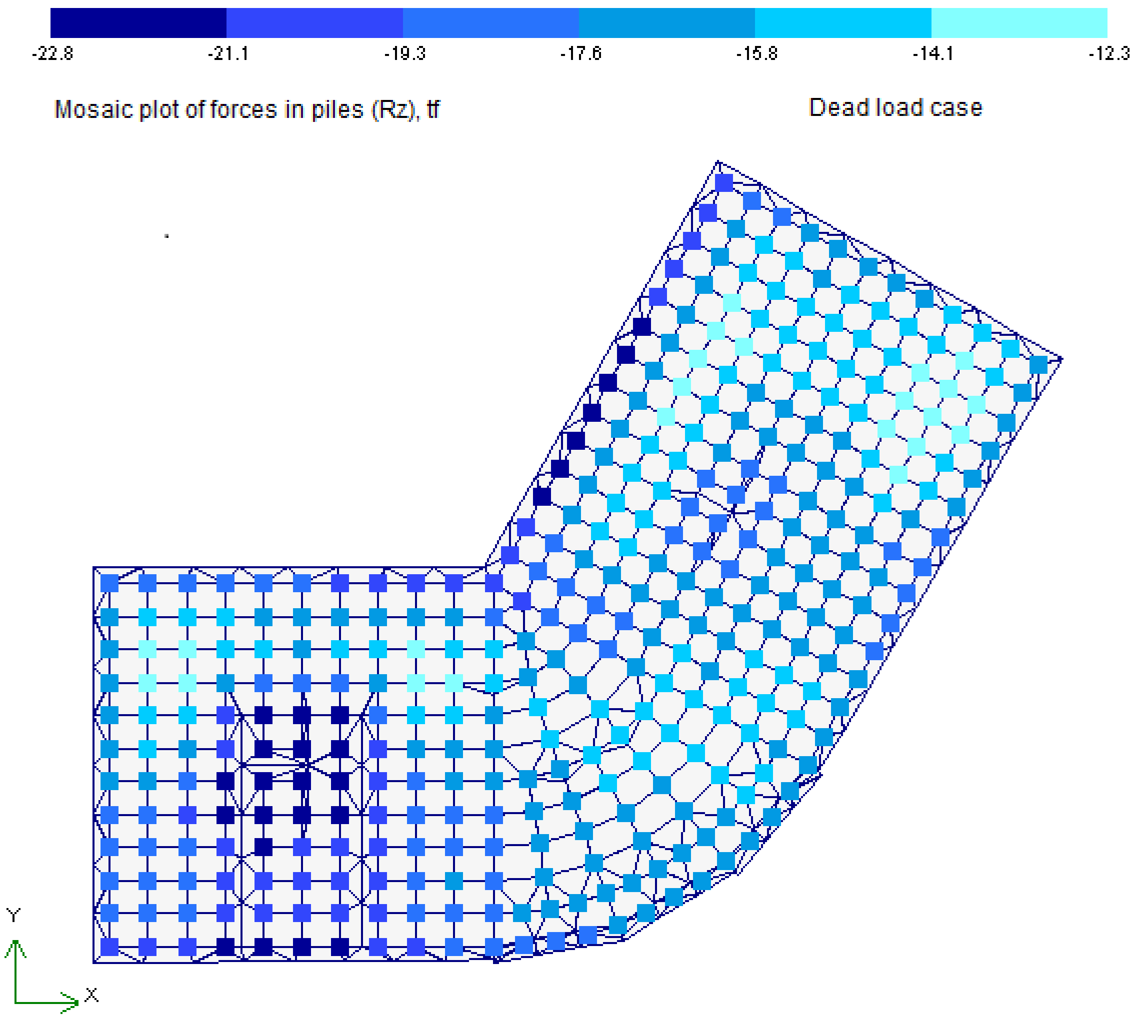


Figure 1.27.2 Mosaic plot of forces in piles on fragmented model (projection - top view)

➪ On the LOADCASE menu, click **2nd wind** (button on the toolbar).

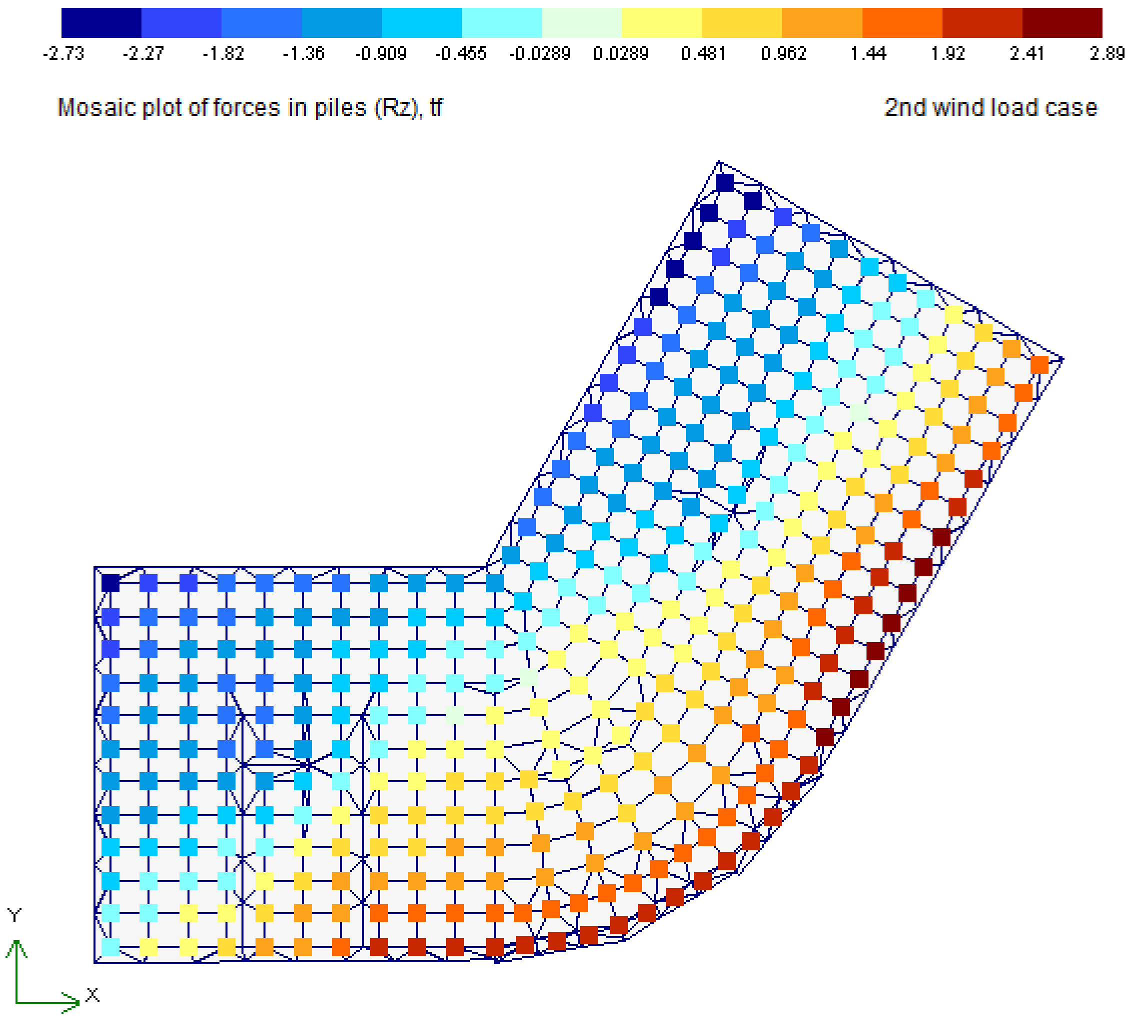


Figure 1.27.3. Mosaic plot of forces in piles on fragmented model (projection - top view)

- On the LOADCASE menu, click **Dead** (button on the toolbar).
- To hide mosaic plots of forces in piles, on the RESULTS menu, point to **Forces in piles** and click **Rz** once again (button on the toolbar) – as result, this command should be inactive.

To present information about piles:

- On the RESULTS menu, click **Information about element or node** (button on the toolbar).
- In the dialog box (see Fig.1.27.4), define selection option:
  - clear the **Nodes** check box.

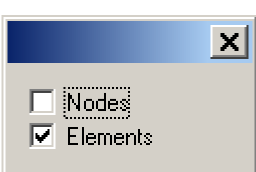


Figure 1.27.4 Dialog box to define selection option

- Define pile at the intersection of axes 1 and A according to the plan of the structure.
- In the **Elements** dialog box, clarify selection:
  - select the element **Pile No.1_1** from the list.
  - click **OK**.
- In the **Pile No.1_1** dialog box (see Fig.1.27.5), review the stiffness EF of the pile and forces in pile from different load cases.

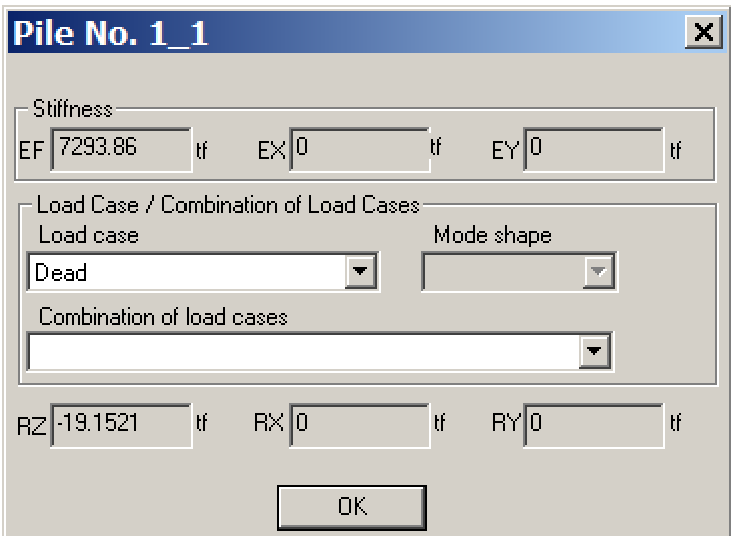


Figure 1.27.5 **Pile No.1_1** dialog box

If you select the earthquake load case, you could choose the mode shape and combination of mode shapes as well. For example, CQC – total quadratic combination of displacements that correspond to mode shapes (for dependent mode shapes).

- Click **OK**.

- In the dialog box (see Fig.1.27.4), define selection option:
  - select the **Nodes** check box;
  - clear the **Elements** check box.
- Define pile at the intersection of axes 1 and A according to the plan of the structure.
- In the **Node No.283** dialog box (see Fig.1.27.6), review nodal displacements from different load cases.
- Click **OK**.

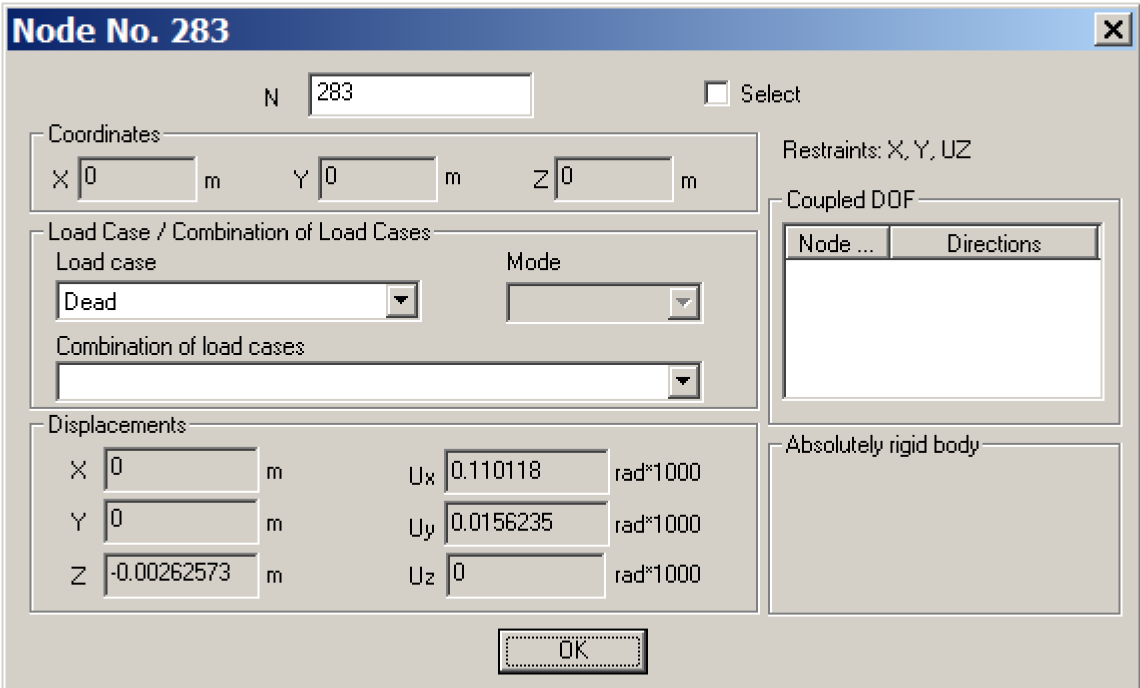


Figure 1.27.6 **Node No.283** dialog box

According to analysis results, you will obtain normative values of displacements, stresses, forces and loads with load factors equal to 1 (Yf=1). To display values of displacements, stresses, forces and loads with account of defined load factors (Yf>1) and combination coefficients, select one of previously defined load combinations.

- On the VIEW menu, click **Restore model** (button on the toolbar).

To generate and save the report file:

- On the RESULTS menu, click **Report (RTF file)** (button on the toolbar).

With this command, report will be generated according to results of FEA.

- In the **Report for FEA results** dialog box (see Fig.1.27.7), do the following:
  - select the **Load coefficients** check box;
  - select the **Drifts of storeys** check box;
  - for the **Piles** check box, define value 1(1-10);
  - clear the **Columns** check box;
  - clear the **Beams** check box;
  - clear the **Walls** check box.
- Click **OK**.

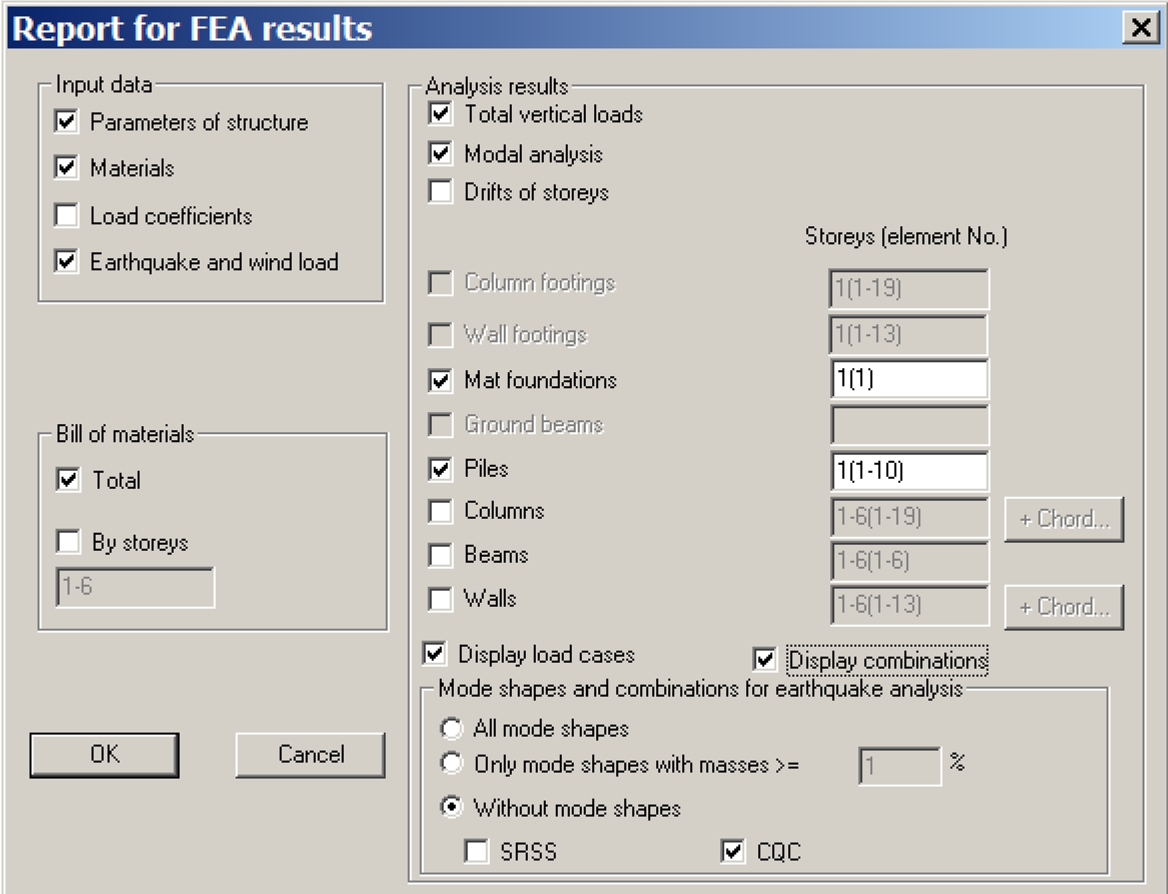


Figure 1.27.7 **Report for FEA results** dialog box

➪ In the **Save as** dialog box, rename the *Model2.rtf* file as *Model2_fea.rtf* and save the file in the **Notes** subdirectory of the base MONOMAKH-SAPR directory.

<u>To review the report file:</u>
Open the report file in MS Word.

➪ on the taskbar, click the **Start** button, point to **All programs** and click **MS Word**.

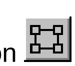
➪ Open the report file with the **Open** command (on the FILE menu).

➪ In the **Open** dialog box, do the following:

- in the **File type** list, select **Rich Text Format (*.rtf)**;
- open the the **Notes** subdirectory of the base MONOMAKH-SAPR directory;
- open the *Model2_fea.rtf* file.

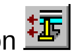
*Report file contains several tables; it is possible to review and print this file. Here you will find input data and results of FEA. You could also open the report file in *.rtf format in the WordPad.*

➪ Close the file in MS Word.

➪ To return to the main view, on the VIEW menu, click **Main view** (button  on the toolbar).

Step 28. Export to design modules of MONOMAKH-SAPR

To export data to design module of MONOMAKH-SAPR:

- On the RESULTS menu, click **Export data to MONOMAKH-SAPR modules** (button  on the toolbar).
- In the **Export data to MONOMAKH-SAPR modules** dialog box (see Fig.1.28.1), do the following:
  - clear the **Columns** check box;
  - clear the **Beams** check box;
  - clear the **Slabs** check box;
  - clear the **Footings** check box;
  - clear the **Ground beams** check box;
  - clear the **Sectional elevations** check box;
  - clear the **Wall groups as sectional elevations** check box;
  - clear the **Elevations (BRICK)** check box;
  - other parameters remain by default (**Mat foundations** check box should be remain selected).
- Click **OK**.

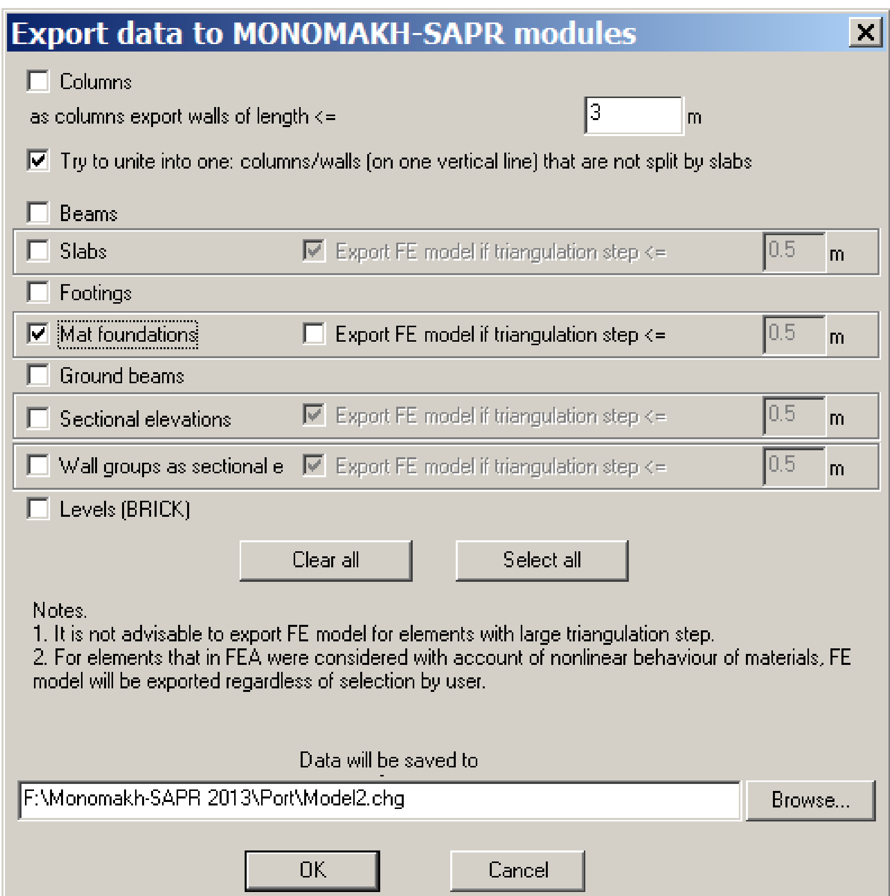


Figure 1.28.1 **Export data to MONOMAKH-SAPR modules** dialog box

In the **Port** subdirectory in the base MONOMAKH-SAPR directory the program generates new directory named as the problem name *Model2.chg*. File with data about mat foundation on pile footing will be placed in this directory.

