

Example 9. Analysis of structure on soil foundation with the use of SOIL system

In this lesson you will learn how to:

- generate design model of the framework using perfectly rigid bodies (PRB) that simulate rigid connections of column and slab elements;
- simulate multi-layer foundation and calculate moduli of subgrade reaction C1 and C2 according to geological engineering survey;
- apply loads including additional loads from neighbouring buildings for calculating subgrade moduli;
- introduce the procedure of iteration refinement of subgrade moduli.

Description:

Reinforced concrete floor slab with dimensions in column axes 4 x 6 m, thickness 150 mm.

Reinforced concrete base slab with dimensions in column axes 4 x 6 m, thickness 500 mm.

Reinforced concrete columns of rectangular section with dimensions 400 x 800 mm.

Height of the framework – 3 m.

Analysis is carried out for floor slab with dimensions of finite elements 0.2 x 0.2 m and base slab with dimensions of FE 0.4 x 0.4 m and 0.4 x 0.2 m (see Fig.9.1).

Loads:

- load case 1 – dead weight;
- load case 2 – dead uniformly distributed load $g_1 = 0.5 \text{ t/m}^2$ applied to floor slab; dead uniformly distributed load $g_2 = 1 \text{ t/m}^2$ and dead concentrated vertical load $P = 100 \text{ t}$ applied to base slab (see Fig.9.2, load case 2);
- load case 3 – concentrated horizontal load $F = 2 \text{ t}$ applied to floor slab (see Fig.9.2, load case 3).

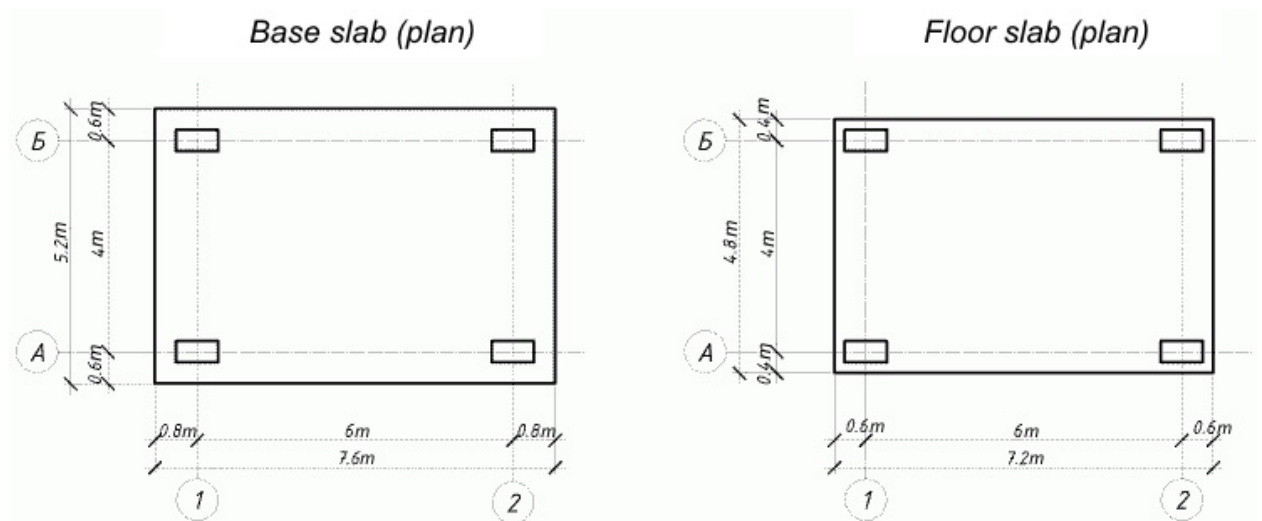


Figure 9.1. Model of 3D framework

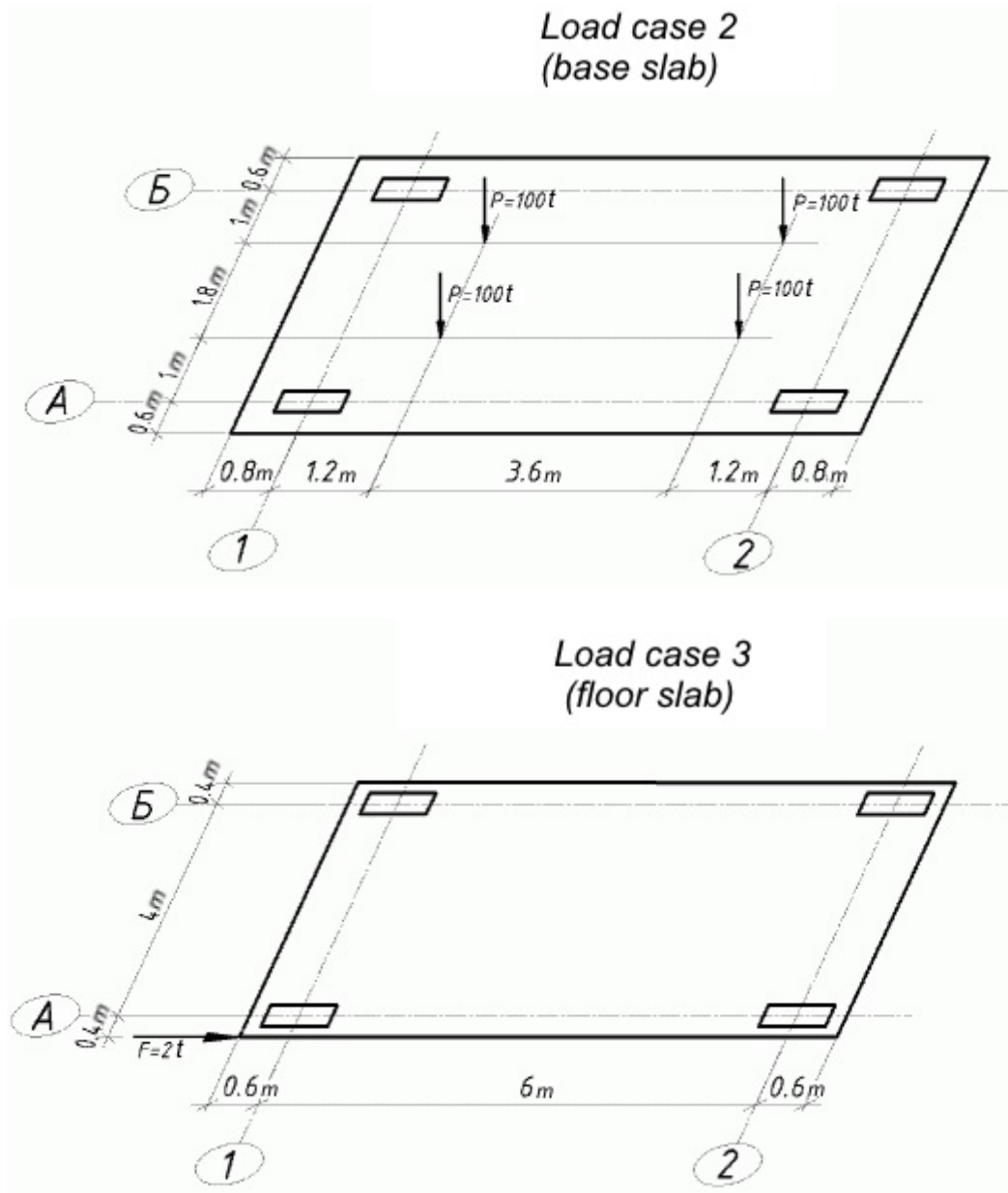


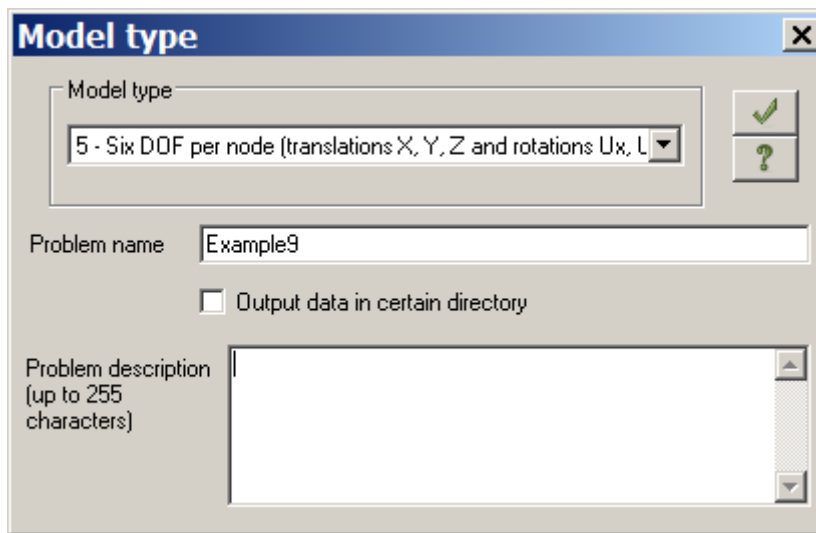




Figure 9.2. Load cases for slabs in framework

- ⇒ On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2015** and then click **LIRA-SAPR 2015**.

Step 1. Creating new problem

- ⇒ On the FILE menu, click **New** (button  on the toolbar).
- ⇒ In the **Model type** dialog box (see Fig.9.3) specify the following data:
- problem name – **Example9**;
 - model type – **5 – Six degrees of freedom per node** (translations X, Y, Z and rotations Ux, Uy, Uz).
- ⇒ Click **OK** .

Figure 9.3 **Model type** dialog box


It is also possible to open the **Model type** dialog box with a pre-defined type of model. To do this, on the **LIRA-SAPR menu** (Application menu), point to **New** and click **Model type 5 (Six DOF per node)** command . One more way to do the same: on the Quick Access Toolbar, click **New** and in the drop-down menu select **Model type 5 (Six DOF per node)** command . Then you should define only problem name.




To save all output data files for the problem in certain directory, select appropriate check box. The directory name will coincide with the name of the problem. This directory will appear in the directory for files with analysis results. This is helpful if you have to find output data files for certain problem, then transfer these files or review and evaluate them with the help of Windows Explorer or other file managers.

Step 2. Generating model geometry

To generate base slab:

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** list and click the **Create slab**  command.
- ⇒ In the **Create plane fragments and grids** dialog box specify the following data:
 - spacing along the first axis: spacing along the second axis:

L(i)	N	L(i)	N
0.4	19	0.4	1
		0.2	2
		0.4	9
		0.2	2
		0.4	1
 - other parameters remain by default (see Fig.9.4).
- ⇒ Click **Apply** .

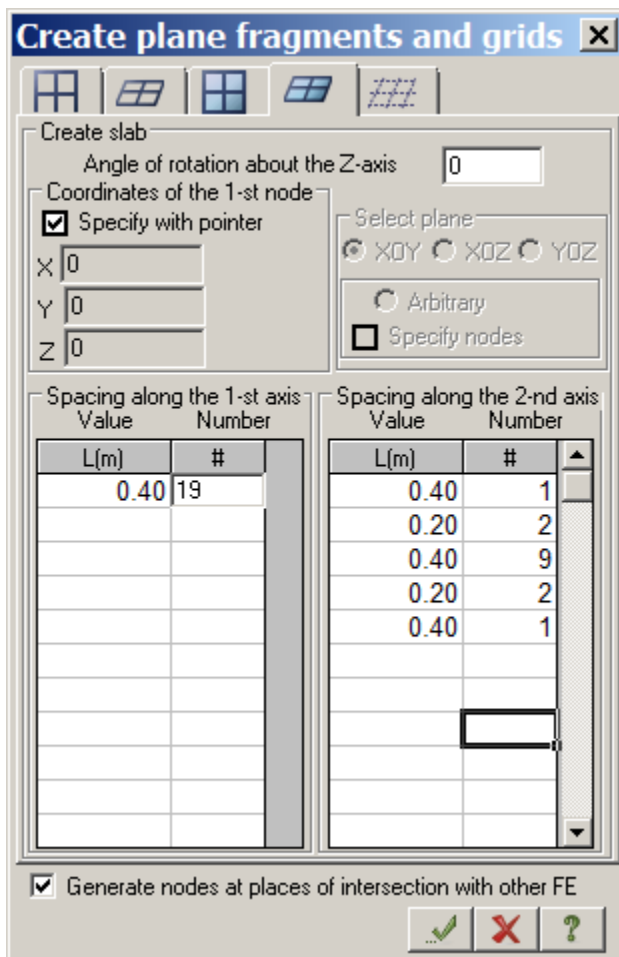





Figure 9.4 Create plane fragments and grids dialog box

To generate the floor slab:

- ⇒ Then in the **Create plane fragments and grids** dialog box, in the **Coordinates of the 1st node** area, clear the **Specify with pointer** check box and define coordinates for the first node of the fragment:
 - X(m) Y(m) Z(m)
0.2 0.2 3.
- ⇒ Define the spacing along the first and the second axes (unnecessary rows should be cleared):
 - spacing along the first axis: spacing along the second axis:
L(m) N L(m) N
0.2 36 0.2 24.
- ⇒ Click **Apply** .

To present numbers of nodes on the screen:

- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **Flags of drawing** button .
- ⇒ In the **Display** dialog box (see Fig.9.5), select the **Node numbers** check box on the **Nodes** tab.
- ⇒ Click **Redraw** .

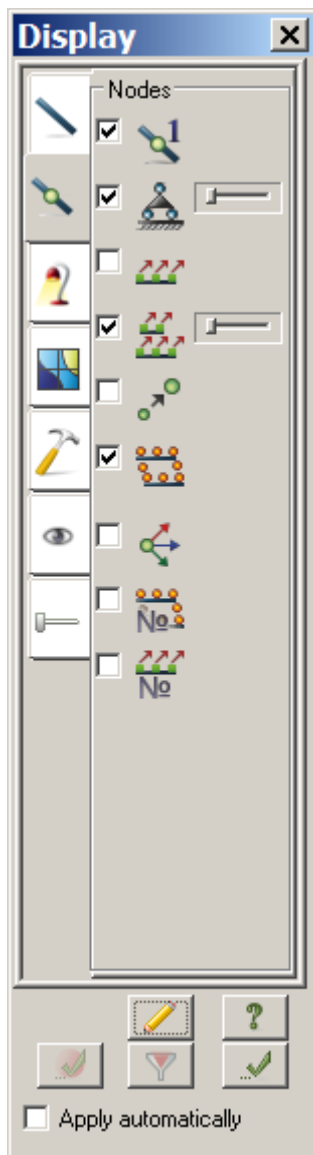




Figure 9.5 Display dialog box

To add columns:

- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **PolyFilter**  .
- ⇒ In the **PolyFilter** dialog box, select the **Filter for nodes** tab (the first tab) (see Fig.9.6).
- ⇒ Select **By numbers of nodes** check box and specify the following node numbers (separated either by comma or by space character): 43, 58, 263, 278, 398, 428, 1138, 1168.
- ⇒ Click **Apply**  .

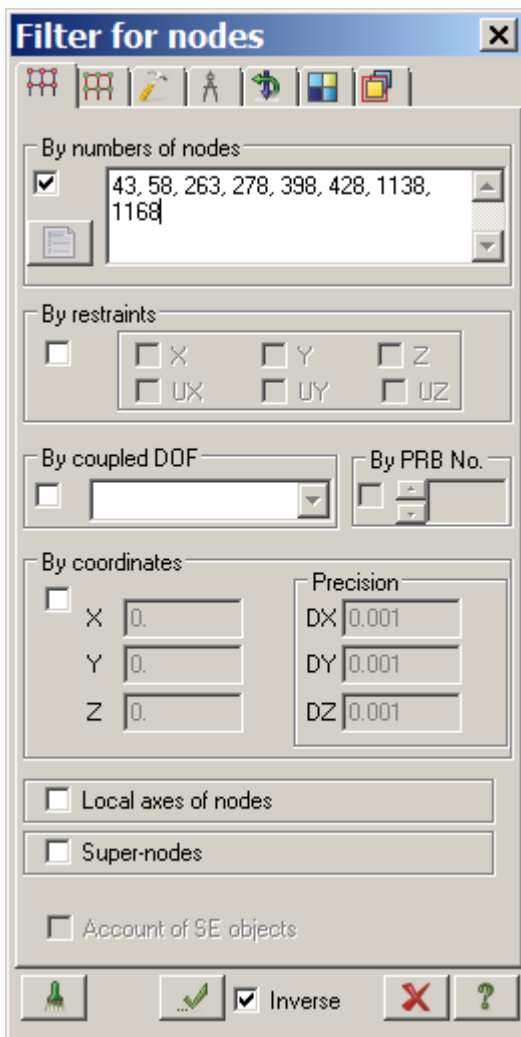




Figure 9.6 Filter for nodes tab

- ⇒ Close the **PolyFilter** dialog box.
- ⇒ To present on the screen only selected nodes of the model, on the **Select** toolbar, click **Fragmentation** .
- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click **Add bar** . The **Add element** dialog box is displayed with the **Add bar** tab open (see Fig.9.7).

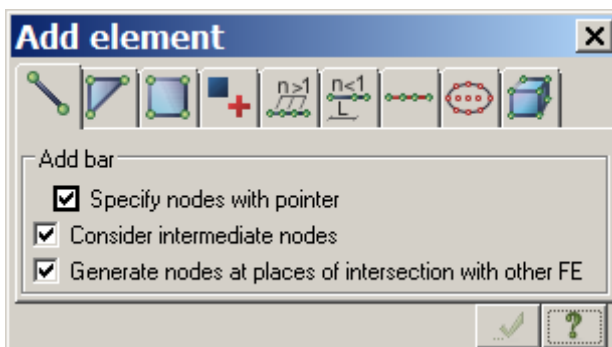




Figure 9.7 Add element dialog box

- ⇒ To add bar elements between nodes, specify with the pointer the following pairs of nodes in sequence: No. 43 and 398, 58 and 428, 263 and 1138, 278 and 1168 (in this case the rubber-band line is automatically stretched between the nodes that you select).





- ⇒ To restore design model in initial view after fragmentation, on the **Select** toolbar, click **Restore model** .

To save data about design model:

- ⇒ On the **LIRA-SAPR menu** (Application menu), click **Save** command .
- ⇒ In the **Save as** dialog box specify the following data:
- file name – **Example9**;
 - location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

Step 3. Generating perfectly rigid bodies (PRB)


To generate perfectly rigid bodies in the base slab:

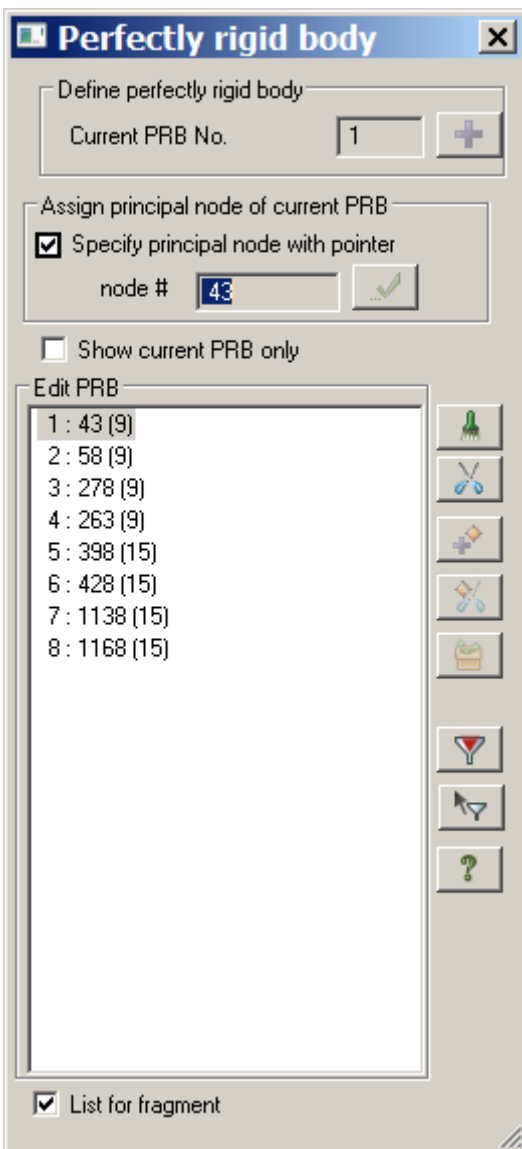
- ⇒ On the **Select** toolbar, click **Select block** button .
- ⇒ Select with the pointer any node or element of the floor slab (nodes and elements of the floor slab will be coloured red).
- ⇒ To present on the screen only unselected nodes and elements, on the **Select** toolbar, click **Inverse fragmentation** button .
- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Perfectly rigid body** button .
- ⇒ On the **Select** toolbar, point to **Select nodes** and click **Select nodes** button .
- ⇒ Select with the pointer nodes where base slab is connected to perfectly rigid body 1 (column body) No. 22-24, 42-44 and 62-64 (nodes will be coloured red).






You can select elements either with a single click or by dragging selection window around appropriate elements.





*If required, it is possible to enlarge the fragment of the model with the **Zoom** command (located on the **Select** toolbar) or with the mouse wheel button.*

- ⇒ In the **Perfectly rigid body** dialog box (see Fig.9.8), in the **Assign principal node of current PRB** area, select the **Specify principal node with pointer** check box.
- ⇒ Then, on the model of base slab, specify with the pointer node No.43 (node will be coloured black).
- ⇒ To generate *the first perfectly rigid body*, in the **Define perfectly rigid body** area, click **Add PRB** button .







Figure 9.8. **Perfectly rigid body** dialog box

*All operations with rigid body are carried out only with their **principal** nodes, e.g. restraints, local (nodal) coordinate system, initial displacement, coupled degrees of freedom.*

- ⇒ Make sure that the **Specify principal node with pointer** check box is selected and specify with the pointer node No.58 on the base slab.
- ⇒ On the **Select** toolbar, point to **Select nodes** and click **Select nodes** button .
- ⇒ Select with the pointer nodes where base slab is connected to perfectly rigid body 2 - nodes No.37-39, 57-59 and 77-79.
- ⇒ To generate *the second perfectly rigid body*, in the **Define perfectly rigid body** area, click **Add PRB** button .
- ⇒ When the **Select nodes**  command is active, select with the pointer nodes where base slab is connected to perfectly rigid body 3 - nodes No.242-244, 262-264 and 282-284.
- ⇒ In the **Perfectly rigid body** dialog box, in the **Assign principal node of current PRB** area, select the **Specify principal node with pointer** check box.
- ⇒ Then, on the model of base slab, specify with the pointer node No.263.


- ⇒ To generate *the third perfectly rigid body*, in the **Define perfectly rigid body** area, click **Add PRB** button .
- ⇒ When the **Specify principal node with pointer** check box is selected, on the model of base slab, specify with the pointer node No.278.
- ⇒ On the **Select** toolbar, point to **Select nodes** and click **Select nodes** button .
- ⇒ Select with the pointer nodes where base slab is connected to perfectly rigid body 4 - nodes No.257-259, 277-279 and 297-299.
- ⇒ To generate *the fourth perfectly rigid body*, in the **Define perfectly rigid body** area, click **Add PRB** button .
- ⇒ To restore design model in initial view after fragmentation, on the **Select** toolbar, click **Restore model** .

To generate perfectly rigid bodies in the floor slab:

- ⇒ On the **Select** toolbar, click **Select block** button .
- ⇒ Specify with the pointer any node or element of base slab.
- ⇒ To present on the screen only unselected nodes and elements, on the **Select** toolbar, click **Inverse fragmentation** button .
- ⇒ On the **Select** toolbar, point to **Select nodes** and click **Select nodes** button .
- ⇒ Select with the pointer nodes where floor slab is connected to perfectly rigid body 5 - nodes No.359-363, 396-400 and 433-437.
- ⇒ In the **Perfectly rigid body** dialog box, in the **Assign principal node of current PRB** area, select the **Specify principal node with pointer** check box.
- ⇒ Then, on the model of floor slab, specify with the pointer node No.398.
- ⇒ To generate *the fifth perfectly rigid body*, in the **Define perfectly rigid body** area, click **Add PRB** button .
- ⇒ In a similar manner to that used above, define the following data:
 - nodes where floor slab is connected to perfectly rigid body 6 - nodes No.389-393, 426-430, 463-467 with principal node No.428;
 - nodes where floor slab is connected to perfectly rigid body 7 - nodes No.1099-1103, 1136-1140, 1173-1177 with principal node No.1138;
 - nodes where floor slab is connected to perfectly rigid body 8 - nodes No.1129-1133, 1166-1170, 1203-1207 with principal node No.1168.
- ⇒ To restore design model in initial view after fragmentation, on the **Select** toolbar, click **Restore model** .
- ⇒ On the **Select** toolbar, point to **Select nodes** and click **Select nodes** button  in order to make this command not active.

Step 4. Defining material properties to elements of the framework

To create material data sets:

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Material properties** button .
- ⇒ In the **Stiffness and materials** dialog box (see Fig.9.9a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.9.9b).

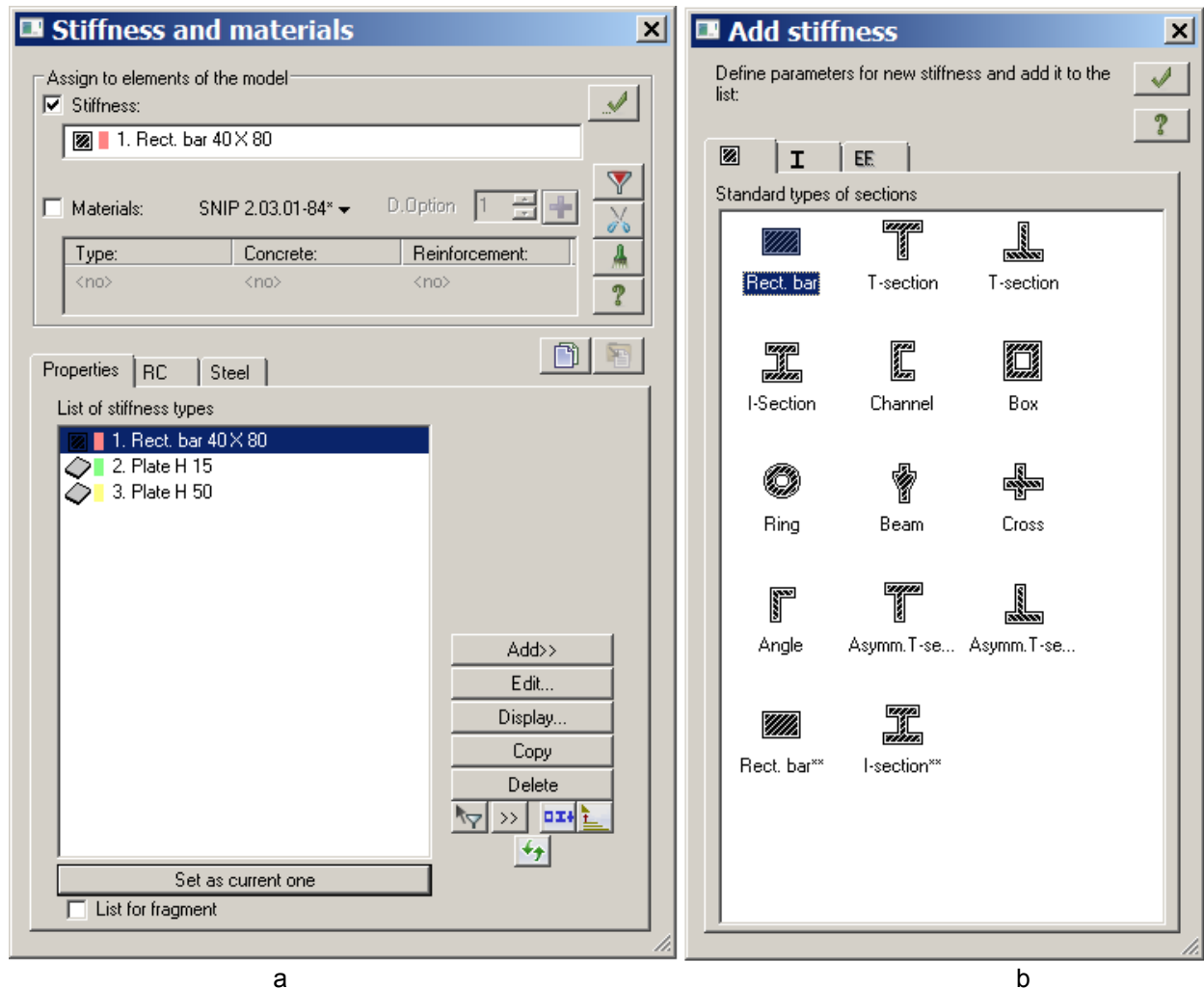



Figure 9.9 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ Double-click the **Rectangular bar** icon in the list. The **Define standard section** dialog box opens. In this dialog box you can define material properties for selected type of the section.
- ⇒ In the **Define standard section** dialog box specify the following parameters for **Rectangular bar** (see Fig.9.10):
 - modulus of elasticity – $E = 3e6 \text{ t/m}^2$ (for the U.S. keyboard layout);
 - geometric properties – $B = 40 \text{ cm}$; $H = 80 \text{ cm}$;
 - unit weight of material – $Ro = 2.75 \text{ t/m}^3$.
- ⇒ To preview schematic presentation, click **Draw**.
- ⇒ To confirm the specified data, click **OK** .

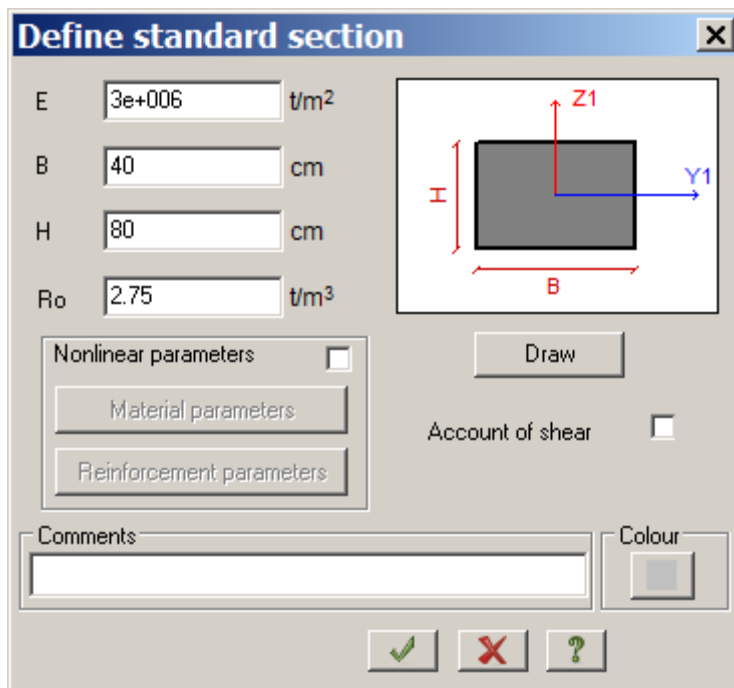

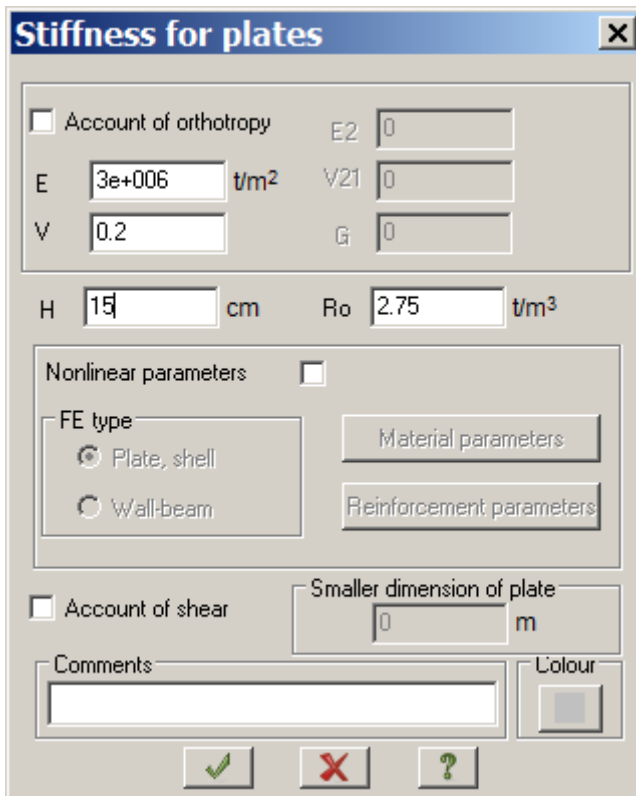


Figure 9.10 Define standard section dialog box


- ⇒ In the **Stiffness and materials** dialog box, select the third tab **Plates, solids, numerical** and double-click the **Plates** icon in the list.
- ⇒ In the **Stiffness for plates** dialog box (see Fig.9.11), specify the following parameters for **Plate** (floor slab):
 - modulus of elasticity – $E = 3e6 \text{ t/m}^2$ (for the U.S. keyboard layout);
 - Poisson's ratio – $\nu = 0.2$;
 - thickness – $H = 15 \text{ cm}$;
 - unit weight of material – $R_o = 2.75 \text{ t/m}^3$.
- ⇒ To confirm the specified data, click **OK** .




The dialog box titled "Stiffness for plates" contains the following fields and controls:

- ☐ Account of orthotropy
- E2:
- E: t/m²
- V21:
- V:
- G:
- H: cm
- Ro: t/m³
- Nonlinear parameters: ☐
- FE type:
 - ☒ Plate, shell
 - ☐ Wall-beam
- Material parameters:
- Reinforcement parameters:
- ☐ Account of shear
- Smaller dimension of plate: m
- Comments:
- Colour:
- Buttons at the bottom:

Figure 9.11 Stiffness for plates dialog box


- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select '2.Plate H 15'.
- ⇒ Click **Copy**.
- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select '3.Plate H 15'.
- ⇒ Click **Edit**.
- ⇒ In another **Specify stiffness for plates** dialog box specify parameter for base slab:
 - thickness - H = 50 cm.
- ⇒ Click **OK** .
- ⇒ To hide the library of stiffness parameters, click **Add** in the **Stiffness and materials** dialog box.



To assign stiffness to elements of the framework:



- ⇒ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the stiffness type **1. Rect. bar 40x80**.
- ⇒ Click **Set as current type** (in this case selected type will be displayed in the **Stiffness** box in the **Assign to elements of the model** area). You can also specify the current type by double-clicking the necessary type in the **List of stiffness types**.)
- ⇒ On the **Select** toolbar, click **Select vertical bars** button .
- ⇒ Select all vertical elements of the model with the pointer. The elements will be coloured red.




*You can select elements either with a single click or by dragging selection window around appropriate elements. When the **Select vertical bars** command is active, you can drag selection window around the whole model and only vertical bars will be selected.*

- ⇒ In the **Stiffness and materials** dialog box, click **Apply** . The elements become unselected. It indicates that the current combination of stiffness type and material is assigned to selected elements.

- ⇒ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the stiffness type **2. Plate H15**.
- ⇒ Click **Set as current type**.
- ⇒ On the **Select** toolbar, click **Select block** button .
- ⇒ Specify with the pointer any node or element of slab.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .

- ⇒ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the stiffness type **3. Plate H50**.
- ⇒ Click **Set as current type**.
- ⇒ On the **Select** toolbar, click **Select block** button .
- ⇒ Specify with the pointer any node or element of base slab.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .


- ⇒ To unselect nodes, on the **Select** toolbar, click **Unselect all** button .

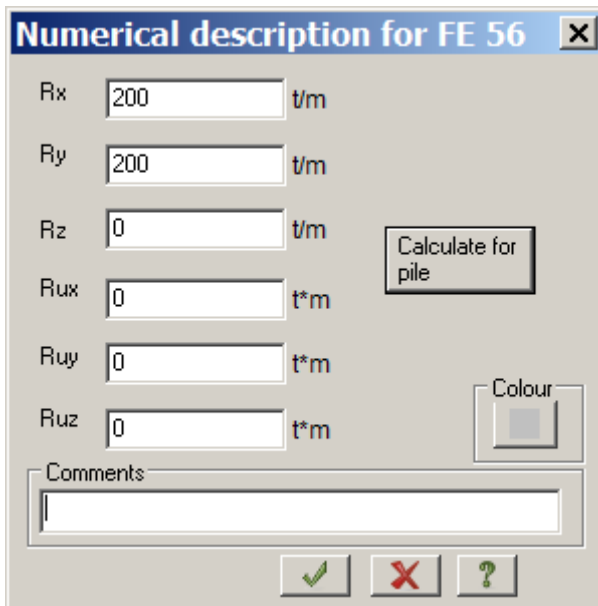
Step 5. Defining boundary conditions



To avoid geometric instability in the XOY-plane, additional boundary conditions are defined for base slab (with the help of one-node elements FE 56). These elements are defined at nodes of the base slab. The stiffness for all FE 56 will be defined as equal to the following value: 70% of stiffness of elastic foundation C1z multiplied to load area associated with one FE 56. As the stiffness of elastic foundation is unknown, we accept initial values of stiffness for FE 56 as equal to 200 t/m. When moduli of subgrade reaction are calculated, the stiffness value for FE 56 may be clarified.

To define stiffness for FE 56:

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Material properties** button .
- ⇒ In the **Stiffness and materials** dialog box, click **Add**. The dialog box expands to display the library of stiffness parameters. In the **Add stiffness** dialog box, select the **Plates, solids, numerical** tab (the third tab).
- ⇒ Double-click the **Numerical for FE 56** icon in the list. The **Numerical description for FE 56** dialog box opens.
- ⇒ In the **Numerical description for FE 56** dialog box (see Fig.9.12), specify the following parameters for the section:
 - stiffness of FE per unit length in tension-compression along the global X-axis – $R_x = 200 \text{ t/m}$;
 - stiffness of FE per unit length in tension-compression along the global Y-axis – $R_y = 200 \text{ t/m}$.
- ⇒ To confirm the specified data, click **OK**.



Numerical description for FE 56

Rx: 200 t/m
 Ry: 200 t/m
 Rz: 0 t/m
 Rux: 0 t*m
 Ruy: 0 t*m
 Ruz: 0 t*m

Calculate for pile




Colour

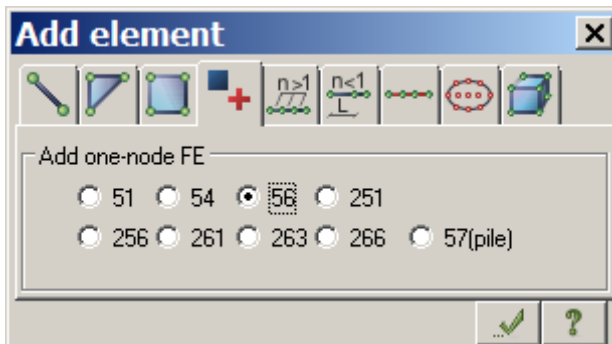
Comments

✓ ✗ ?










Figure 9.12 Numerical description for FE 56 dialog box

To add FE 56:

- ⇒ When the **Select block** button  is active, select with the pointer any node or element of the base slab.
- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click **Add 1-node FE** .
- ⇒ The **Add element** dialog box is presented with the **Add one-node FE** tab open (see Fig.9.13).
- ⇒ In this dialog box, specify with the pointer FE '56' option.
- ⇒ Click **Apply** .



Add element

Add one-node FE

☐ 51 ☐ 54 ☒ 56 ☐ 251
☐ 256 ☐ 261 ☐ 263 ☐ 266 ☐ 57(pile)

✓ ?

Figure 9.13 Add element dialog box





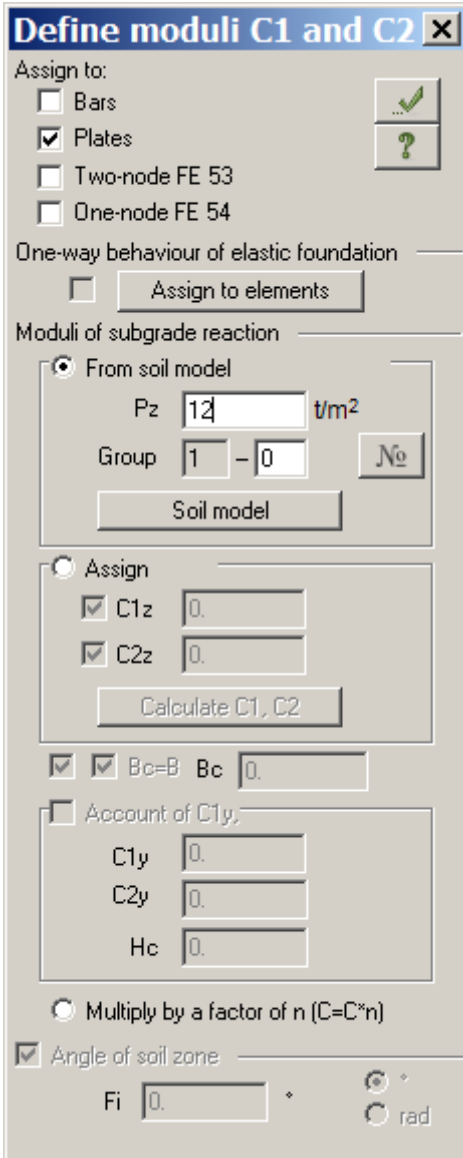
When FE 56 are added to design model, stiffness for these FE was assigned as the current one in the **Stiffness and materials** dialog box. This stiffness was automatically assigned to these added FE.

- ⇒ To unselect elements, on the **Select** toolbar, click **Unselect all** button .

Step 6. Defining parameters of elastic foundation

- ⇒ On the **Select** toolbar, click **Select block** button .

- ⇒ Select with the pointer any node or element of the base slab.
- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Moduli of subgrade reaction** button .
- ⇒ In the **Define moduli C1 and C2** dialog box (see Fig.9.14), make sure that the **Plates** check box is selected.
- ⇒ To define moduli of subgrade reaction, select the **From soil model** option and define:
 - uniformly distributed load on soil $P_z = 12 \text{ t/m}^2$.
- ⇒ Click **Apply** .


Figure 9.14 **Define moduli C1 and C2** dialog box

- ⇒ To unselect all nodes, on the **Select** toolbar, click **Unselect all** button .


To activate **SOIL** system:

- ⇒ To activate the SOIL system, in the **Define moduli C1, C2** dialog box, click **Soil model** button.
- ⇒ The **Soil model** dialog box (see Fig.9.15) appears on the screen. By default, in this dialog box, **Method 3** is defined in the **Calculation method (C1, C2)** list and SNIP 2.02.01-83 is defined in the **Building code** area.

- ⇒ On the calculate C1, C2 tab, define the following parameters that will be applied for calculation of moduli of subgrade reaction:
 - in the **Data for calculation** area, define **coefficient for depth of compressible stratum** as equal to 0.5;
 - in the **Merge loads** area, clear the **Replace adjacent loads or loads of neighbouring values with one load equal to average value** check box.
- ⇒ Then in the **Soil model** dialog box, click **Attach soil model** button.

Figure 9.15 Soil model dialog box



You could also open **Sol model** dialog box by clicking the **Soil model** button  (**Advanced edit options** ribbon tab, **SOIL** panel).

- ⇒ In the **Open file with soil model** dialog box (see Fig.9.16), when the file name **Example9** is mentioned, click **Open**.

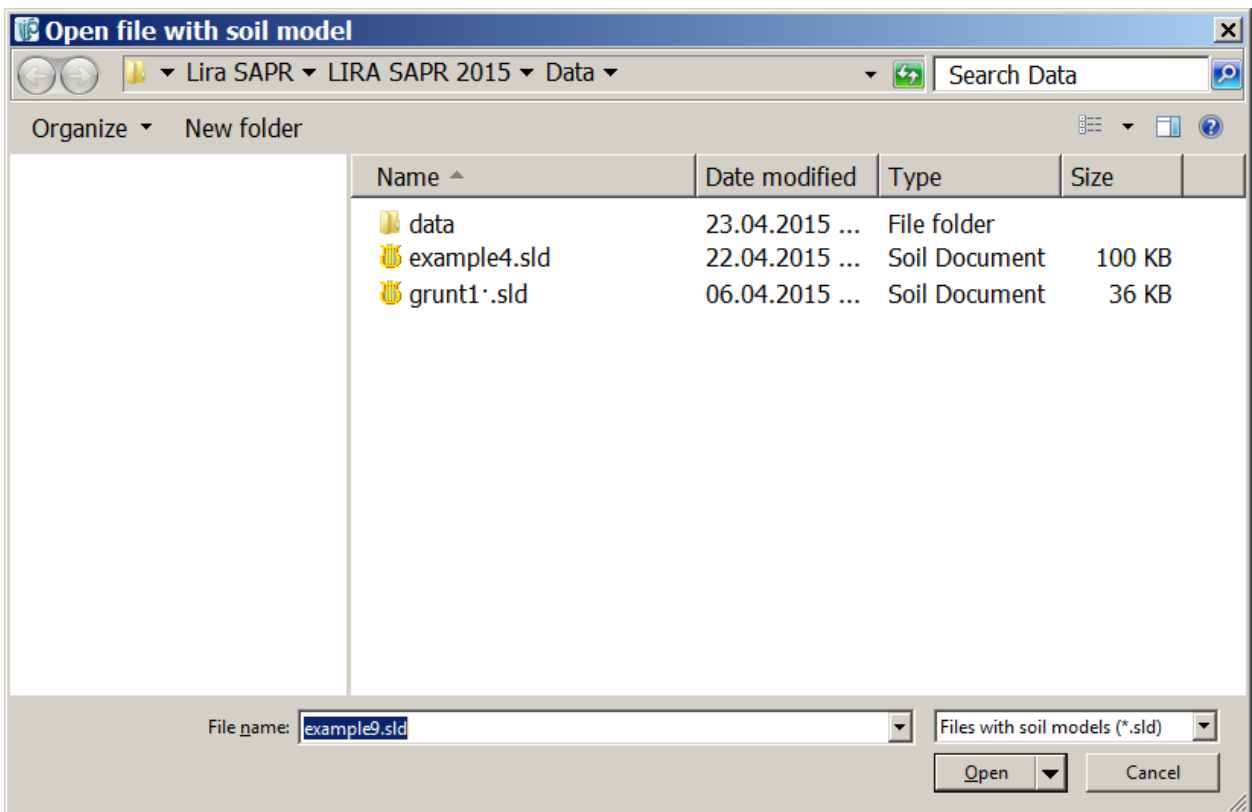


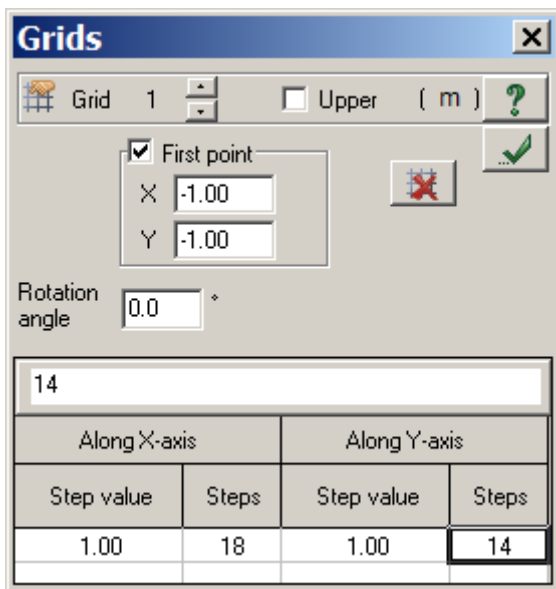


Figure 9.16 Open file with soil model dialog box

To define parameters for grid of nodes:


- ⇒ In the **SOIL** system window, on the MODEL menu, click **Grids** (button  on the toolbar).
- ⇒ In the **Grids** dialog box (see Fig.9.17), select the **First point** check box and define coordinates (m):
 - $X = -1$; $Y = -1$.
- ⇒ Then define the step value for the grid:






Along X-axis		Along Y-axis	
Step value	Steps	Step value	Steps
1	18	1	14.
- ⇒ Click **Apply** .
- ⇒ Close the **Grids** dialog box.

Figure 9.17 **Grids** dialog box

To define parameters for boreholes in geological profile:



To calculate moduli of subgrade reaction we will use soil properties that were defined according to results of geological engineering survey and are located in the **Soil properties** table. To activate this table, on the MODEL menu, click **Soil properties** (button  on the toolbar).

- ⇒ On the MODEL menu, click **Boreholes** (button  on the toolbar).
- ⇒ In the **Boreholes** dialog box (see Fig.9.18), for the mouth elevation specified as equal to 100 (m), define the following data for borehole 1:
 - select **Coordinates** check box;
 - coordinates for the borehole (m): X = 2, Y = 6;
 - in the **Geological element (GE)** list from the **Soil properties** table, define GE 1;
 - **depth of layer** equal to 3 (m);
 - click **Refresh table**  ;
 - in the **Geological element (GE)** list from the **Soil properties** table, define GE 2;
 - **depth of layer** equal to 5 (m);
 - click **Refresh table**  ;
 - in the **Geological element (GE)** list from the **Soil properties** table, define GE 5;
 - **depth of layer** equal to 15 (m);
 - click **Refresh table**  ;
 - other parameters are calculated automatically.
- ⇒ Click **Apply** .

Boreholes

Borehole 1 (m) ?

☒ Coordinates

X 2.00 Y 6.00

Mouth elevation 100.00

Depth 15.00





☒ Table

GE 5




☒ Specify depth of layer

N	Name	Bottom elevation	Thickness of layer	Depth of layer
1	Man-made...	97.00	3.00	3.00
2	Fine...	95.00	2.00	5.00
5	Semihard...	85.00	10.00	15.00

Figure 9.18 Boreholes dialog box

- ⇒ For the borehole 2, define the following parameters:
- coordinates of the borehole (m): X = 9, Y = 4;
 - **depth of layer** for the first layer of soil (GE1) equal to 1 (m);
 - **depth of layer** for the second layer of soil (GE1) equal to 9 (m);
 - **depth of layer** for the third layer of soil (GE1) equal to 15 (m);
 - click **Refresh table** ;
 - other parameters are calculated automatically.
- ⇒ Click **Apply** .
- ⇒ For the borehole 3, define the following parameters:
- coordinates of the borehole (m): X = 4, Y = 1.2;
 - **depth of layer** for the first layer of soil (GE1) equal to 2 (m);
 - **depth of layer** for the second layer of soil (GE1) equal to 6 (m);
 - **depth of layer** for the third layer of soil (GE1) equal to 15 (m);
 - click **Refresh table** ;
 - other parameters are calculated automatically.
- ⇒ Click **Apply** .
- ⇒ Close the **Boreholes** dialog box.

To define loads from the neighbouring building:

- ⇒ On the MODEL menu, click **Loads** (button  on the toolbar).
- ⇒ In the **Loads** dialog box (see Fig.9.19), select the type of load – **Arbitrary load**  (the second tab).
- ⇒ Select the **Elevation** check box and specify the value equal to 97 (m).
- ⇒ For the specified intensity of soil pressure as 20 t/m², click the **Specify load (arbitrary polygon) on plan** button .

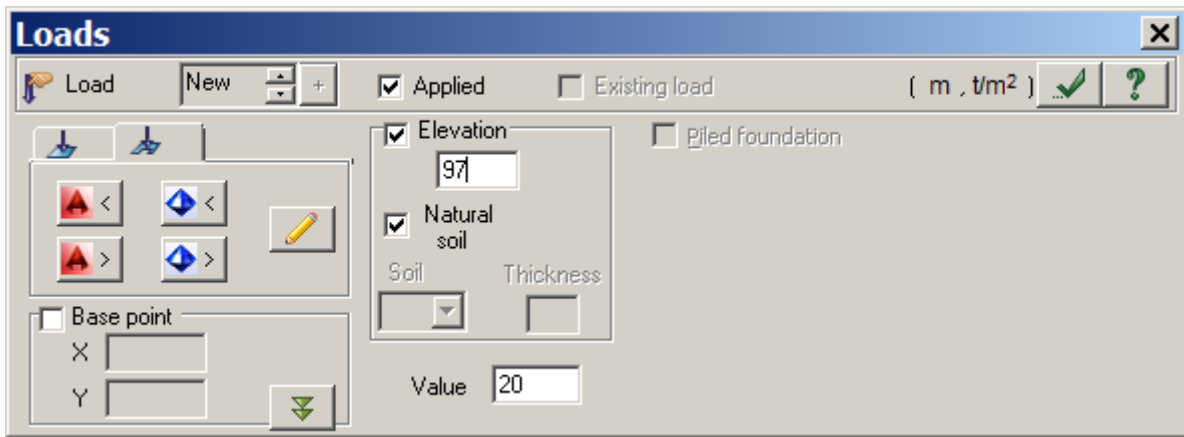


Figure 9.19 Loads dialog box

- ⇒ Then specify (with the pointer) load contours on the grid of soil as presented in Fig.9.20 (beginning from the point with coordinate X = 16, Y = 12 and move the pointer 8m left, then 4m back, then 4m right, 5m back, 4m right and come back to initial point).

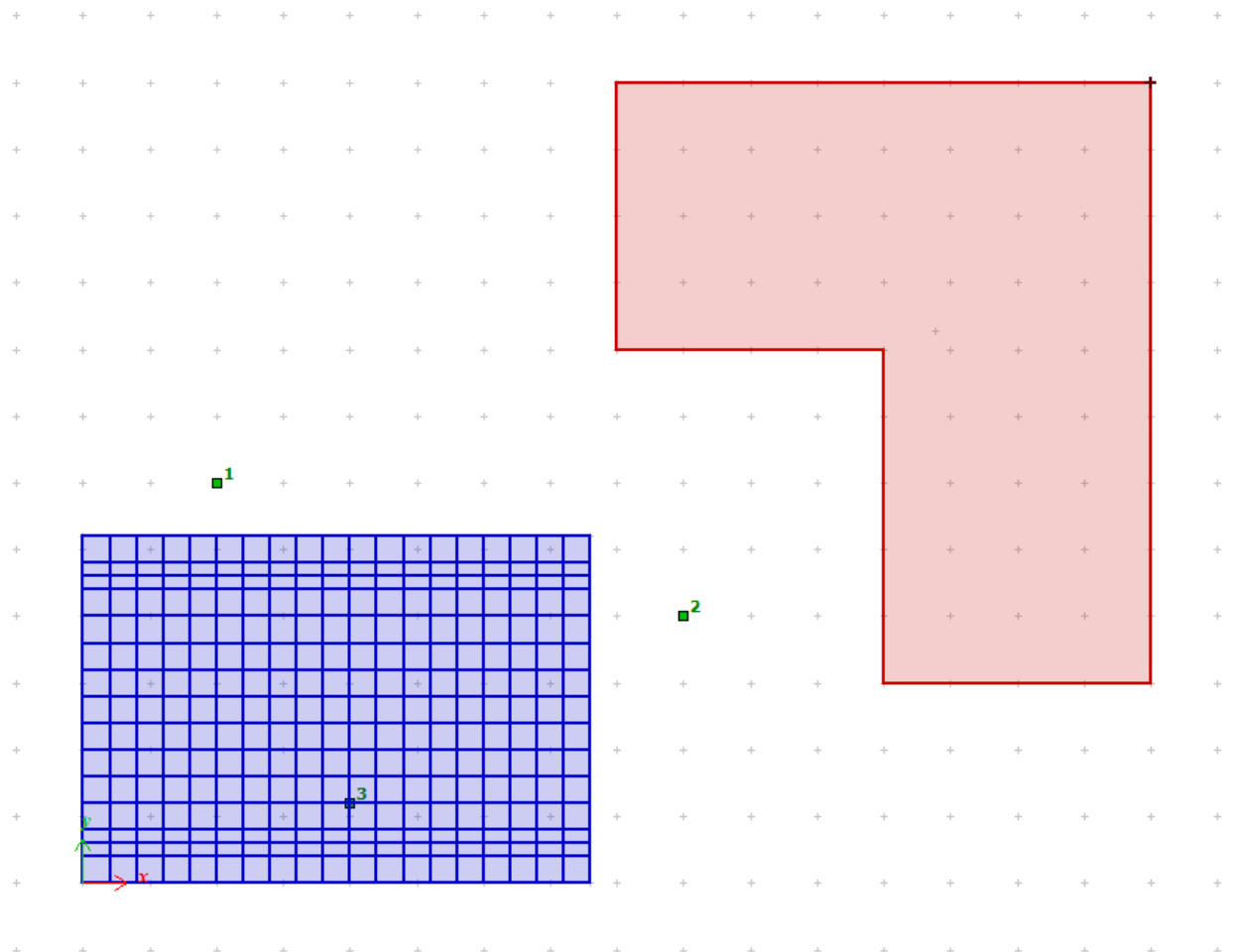


Figure 9.20. Location of load contour on grid of soil

- ⇒ In the **SOIL** dialog box (see Fig.9.21), click **OK**.

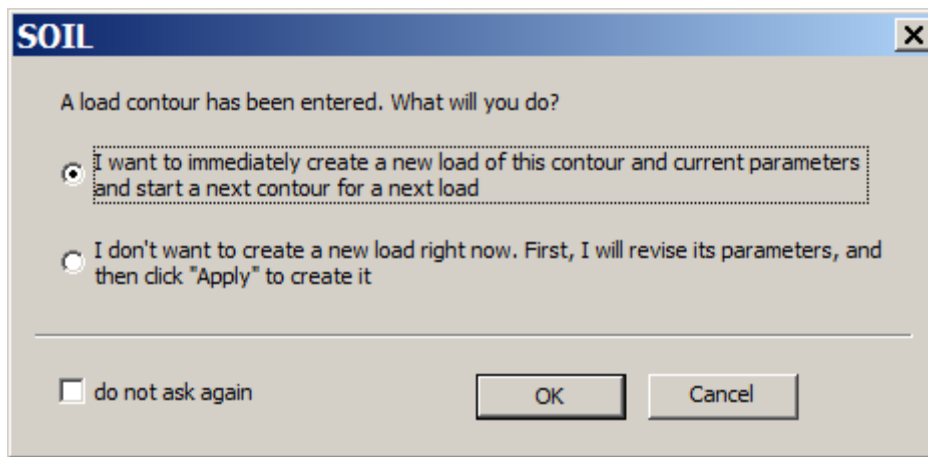




Figure 9.21. SOIL dialog box

⇒ Close the **Loads** dialog box.

To define elevation for the base slab of the framework:

- ⇒ On the MODEL menu, click **Imported loads** (button  on the toolbar).
- ⇒ In the **Imported loads** dialog box (see Fig.9.22), define the elevation value as equal to Z = 96 m.
- ⇒ Click **Apply** .

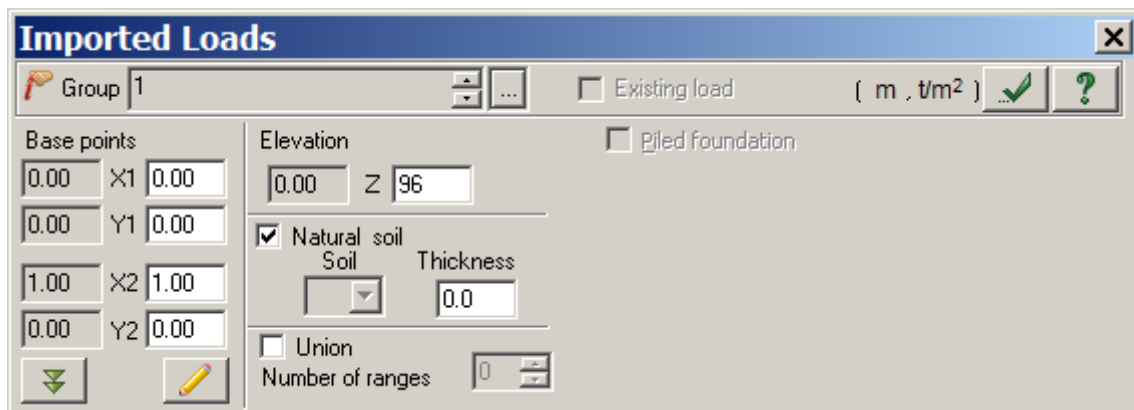





Figure 9.22 Imported loads dialog box

- ⇒ Close the **Imported loads** dialog box.
- ⇒ To save the data about design model, on the FILE menu, click **Save** (button  on the toolbar).
- ⇒ Close the **SOIL** system window and return to the main window of **LIRA-SAPR**.

Step 7. Applying loads

To create load case No.1:

- ⇒ To define load from dead weight of the slab, on the **Create and edit** ribbon tab, select the **Loads** panel and click **Add dead weight** .
- ⇒ In the **Add dead weight** dialog box (see Fig.9.23), click **All elements** and specify **Load factor** as equal to 1. Then click **Apply**  (dead weight of elements is added automatically).

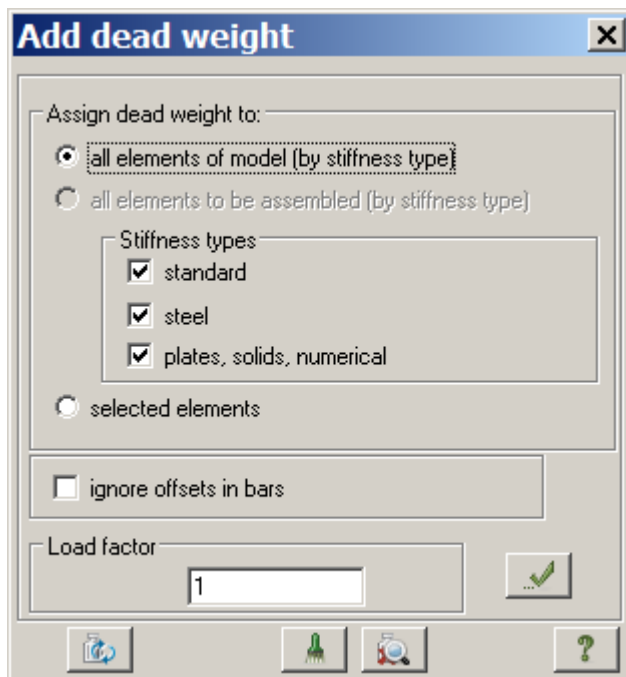





Figure 9.23 Add dead weight dialog box

To create load case No.2:

- ⇒ To change the number of the current load case, click the **Next load case** button  located on the Status bar or on the toolbar.
- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Select nodes No.106, 115, 206 and 215 with the pointer.
- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel, then select **Load on nodes** command  from the **Loads on nodes and elements** drop-down list.
- ⇒ In the **Define loads** dialog box (see Fig.9.24), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

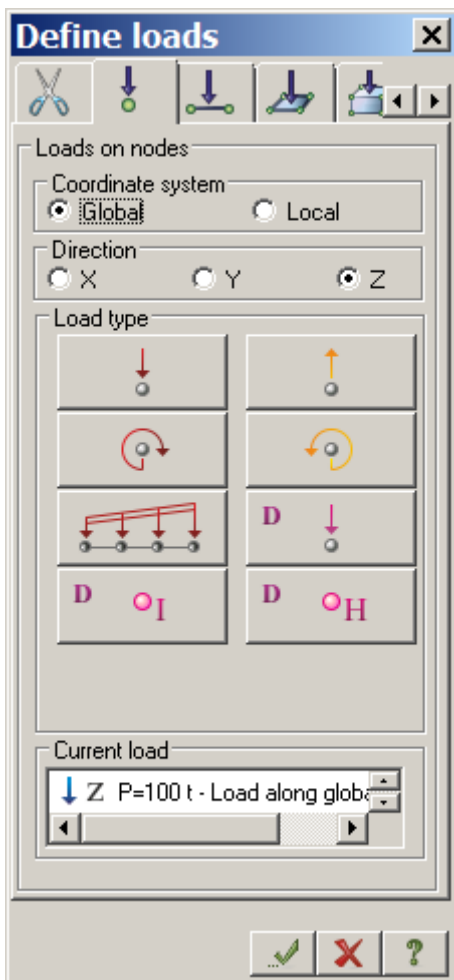




Figure 9.24 Define loads dialog box

- ⇒ In the **Load type** area, click **Concentrated load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 100 \text{ t}$ (see Fig.9.25).
- ⇒ Click **OK** .

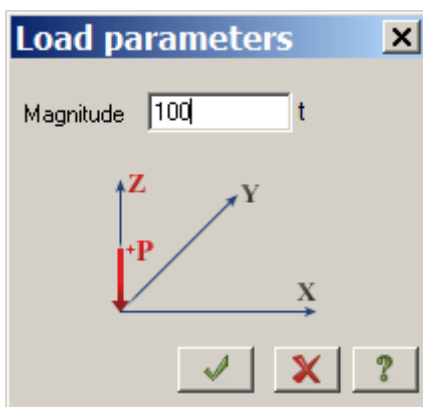

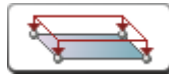


Figure 9.25 Load parameters dialog box

- ⇒ With **Select block** option, select the elements of base slab.
- ⇒ In the **Define loads** dialog box, select the fourth tab **Load on plates** .
- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 1 \text{ t/m}^2$ (see Fig.9.26).

⇒ Click **OK** .

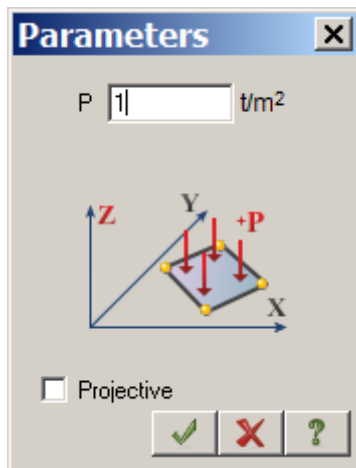
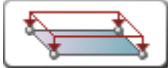



Figure 9.26 Load parameters dialog box

⇒ Select elements of the base slab.


⇒ In the **Define loads** dialog box, click **Uniformly distributed load** button  once again.


⇒ In the **Load parameters** dialog box specify $P = 0.5 \text{ t/m}^2$.


⇒ Click **OK** .

⇒ To unselect nodes, on the **Select** toolbar, click **Unselect all** button .

To create load case No.3:

⇒ To change the number of the current load case, click the **Next load case** button  located on the Status bar or on the toolbar.

⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button . Then select node No.321 (the nearest left node on the floor slab).


⇒ In the **Define loads** dialog box, select the second tab **Load on nodes**  and define direction along the **X**-axis.


⇒ In the **Load type** area, click **Concentrated load** button .


⇒ In the **Load parameters** dialog box specify $P = -2 \text{ t}$.


⇒ Click **OK** .

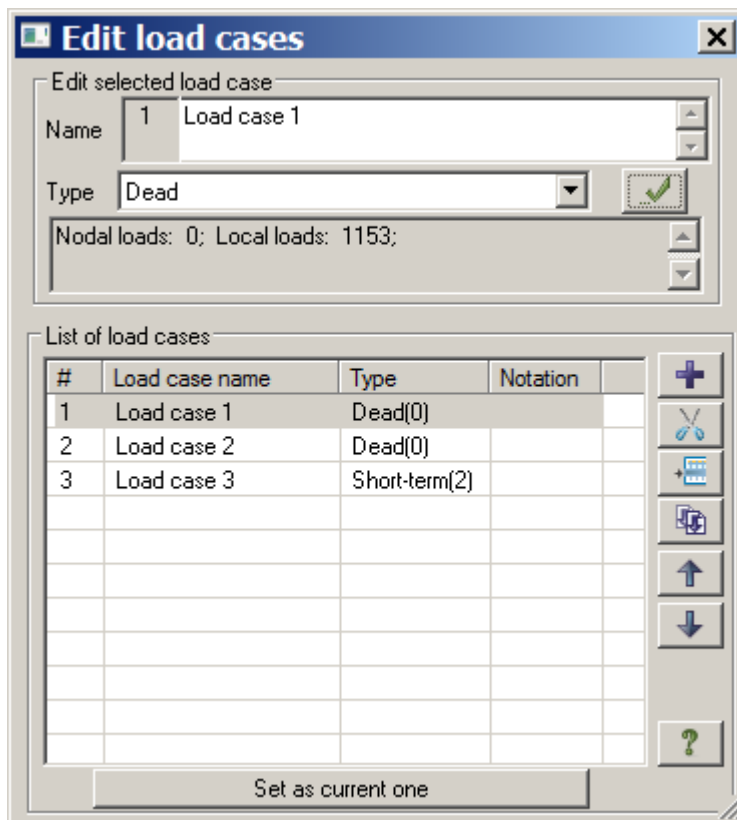
To define detailed information about load cases:

⇒ On the **Create and edit** ribbon tab, on the **Loads** panel, click **Edit load cases** . The **Edit load cases** dialog box is displayed on the screen (see Fig.9.27).

⇒ For load case 1 – in the **Edit selected load case** area, in the **Type** box, select **Dead** and click **Apply** .

⇒ For load case 2 – in the **Edit selected load case** area, in the **Type** box, select **Dead** and click **Apply** .

⇒ For load case 3 – in the **Edit selected load case** area, in the **Type** box, select **Short-term** and click **Apply** .

Figure 9.27. **Edit load cases** dialog box

⇒ Close the **Edit load cases** dialog box.

Step 8. Generating DCL table

⇒ On the **Analysis** ribbon tab, select the **More calculations** panel and click **DCL** button .



*As the type of load cases was defined in the **Edit load cases** dialog box (see Fig.9.27), the DCL table is generated automatically with parameters accepted by default for every load case. Now you have to modify parameters for the third load case and define combinations.*

⇒ In the **Design combinations of loads** dialog box (see Fig.9.28), select building code **SNIP 2.01.07-85*** and for load case 3, in the **Load factor** cell, define factor as equal to 1.4.

⇒ To define combinations, do the following steps:

- in the list of combinations, select the first row (**1 main**) and click **Add**.
- then in the list of combinations, select the second row (**2 main**) and click **Add** (columns with coefficient values according to SNIP 2.01.07-85 will appear in the table).

⇒ To save defined data, click **Save data** .

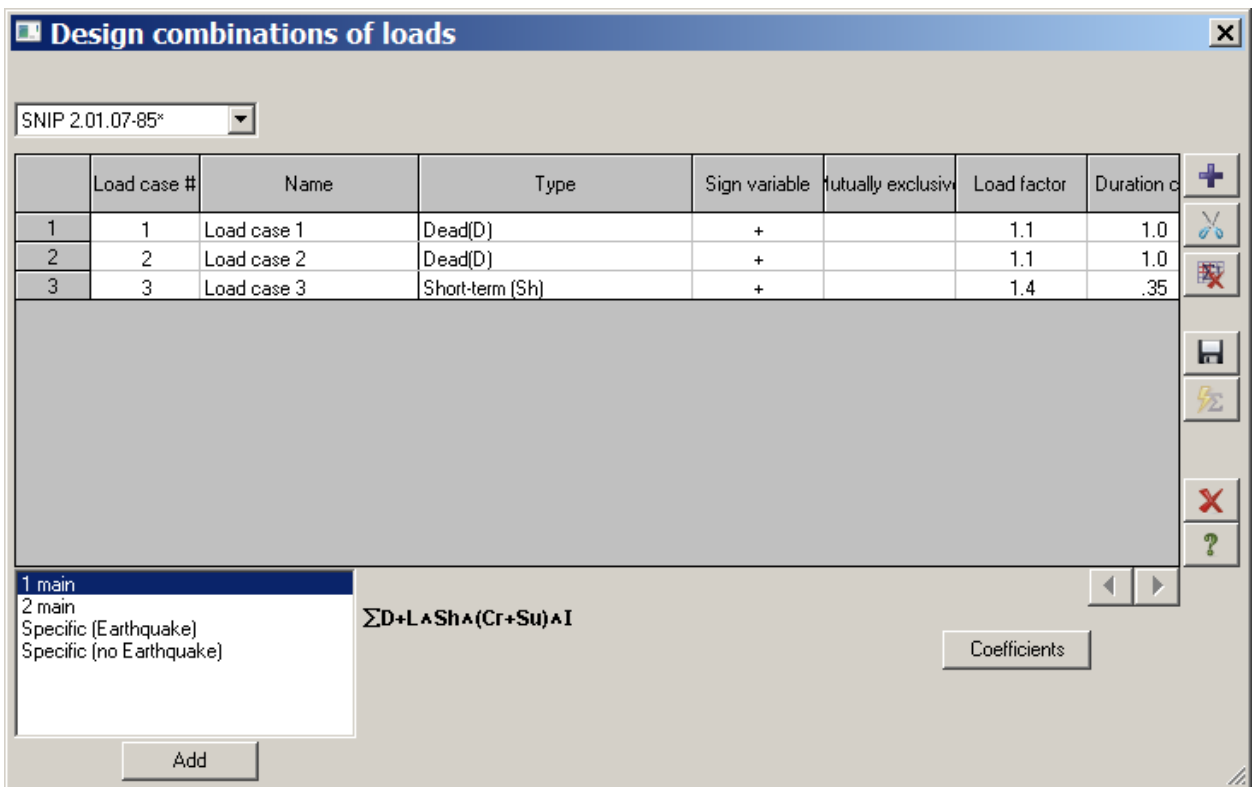



Figure 9.28 Design combination of loads dialog box

⇒ Close the **Design combinations of loads** dialog box.

Step 9. Static analysis of framework

- ⇒ To carry out static analysis, on the **Analysis** ribbon tab, select the **Analysis** panel and in the **Analyse** drop-down list, click **Complete analysis** .
- ⇒ In the **Warning** box (see Fig.9.29) that appears on the screen, make sure that **Recalculate moduli of subgrade reaction C1 and C2 for elastic foundation by soil model** check box is selected and click **OK**.

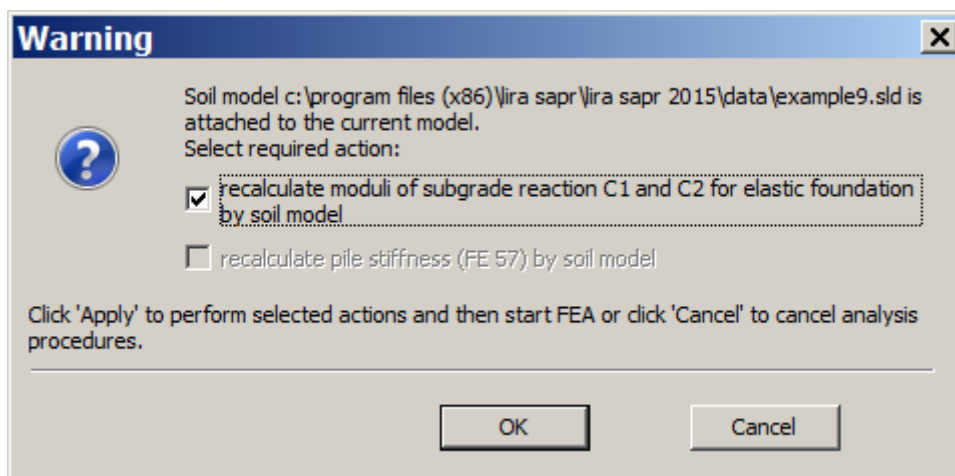


Figure 9.29. Warning box

Step 10. Review and evaluation of static analysis results



When analysis procedure is complete, to review and evaluate analysis results, select the **Results** ribbon tab.

To hide presentation of loads on design model:

- ⇒ On the **Select** toolbar, click **Flags of drawing** button . In the **Display** dialog box, clear the **Loads** check box on the **General** tab.
- ⇒ Click **Redraw** .
- ⇒ In the mode of analysis results visualization, by default design model is presented with account of nodal displacements (see Fig.9.30).

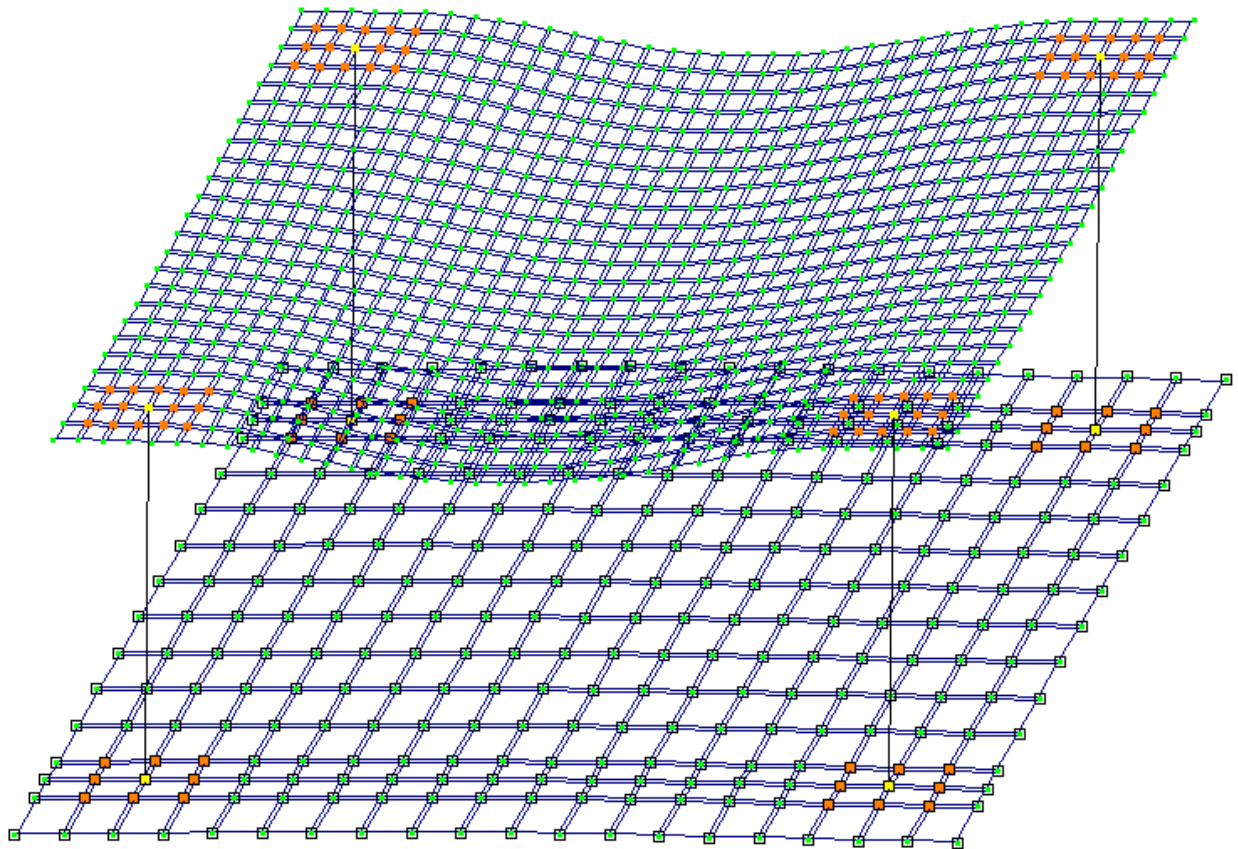





Figure 9.30. Design model with account of nodal displacements

To present displacement contour plots:






- ⇒ To present contour plot of displacements along the Z-axis, on the **Results** ribbon tab, on the **Deformations** panel, select the **Displacement mosaic/contour plot in global coordinate system** command in the **Displacement mosaic/contour plot** drop-down list.
- ⇒ Then click **Displacements along Z** button on the same panel.

To present diagrams of internal forces:

- ⇒ To display diagram M_y , on the **Results** tab, select **Forces in bars** panel and click **Moment diagrams (M_y)** button .

- ⇒ To display diagram Q_z , on the **Results** tab, select **Forces in bars** panel and click **Shear force diagrams (Q_z)** button .
- ⇒ To display diagram N , on the **Results** tab, select **Forces in bars** panel and click **Axial force diagrams (N)** button .
- ⇒ To display mosaic plots N , on the **Results** tab, select **Forces in bars** panel and click **Mosaic plot of forces in bars** command  in the **Force diagrams/Mosaic plots** drop-down list.




To present stress mosaic plots:

- ⇒ To present stress mosaic plot for M_x , on the **Results** ribbon tab, on the **Stress in plates and solids** panel, select the **Stress mosaic plot** command  in the **Stress mosaic/contour plots** drop-down list.
- ⇒ Then click **Stress M_x** button  on the same panel.
- ⇒ To present stress mosaic plot for N_x , click **Stress N_x** button  on the same panel.
- ⇒ To present stress mosaic plot for R_z (soil pressure), click **Stress R_z** button  on the same panel.
- ⇒ To present the full picture of stress mosaic plots for R_z in base slab, select the slab with the **Select block** command and perform fragmentation.
- ⇒ To restore design model in initial view, on the **Select** toolbar, click **Restore model** .




*Elements located inside the PRB are not considered in analysis. That's why, forces in such elements are absent with the exception of soil pressure.
Elements of floor slab that are located inside the PRB may be deleted from design model because they have no influence on analysis of the framework.*



To present mosaic plots for subgrade moduli:

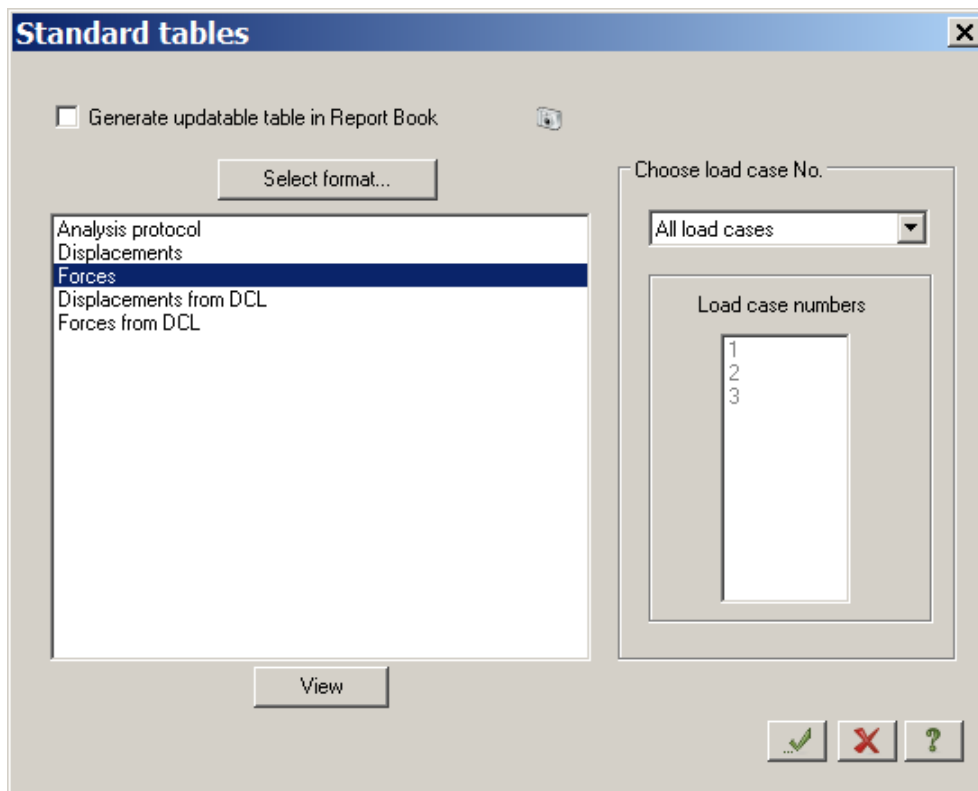
- ⇒ To present mosaic plot for modulus of subgrade reaction C_1 , on the **Advanced results** ribbon tab, on the **Subgrade moduli** panel, in the **Mosaic/Contour plot** drop-down list, click **Mosaic plot C_1 , C_2 , P_z** button . Then, on the same panel, click **Mosaic plot C_1z** .
- ⇒ To present mosaic plot for uniformly distributed load on soil P_z , on the **Subgrade moduli** panel, click **Mosaic plot P_z** .

To change number of active load case:

- ⇒ On the status bar (displayed at the bottom of the screen), in the **Load case No.** list, select No. 2 and click **Apply** .





To generate and review tables of analysis results:

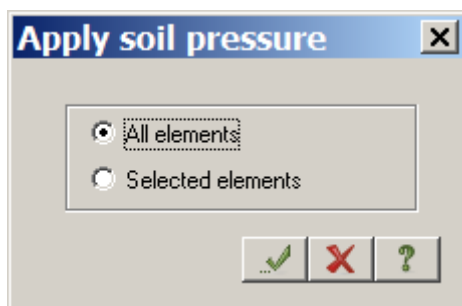
- ⇒ To present table with force values in elements of the model, on the **Results** ribbon tab, select **Tables** panel and click **Standard tables**  in the **Documents** drop-down list.
- ⇒ In the **Standard tables** dialog box (see Fig.9.31), select **Forces** in the list.
- ⇒ Make sure that **All load cases** option is defined in the **Choose load case No.** list and click **Apply** .

Figure 9.31 **Standard tables** dialog box

- ⇒ To close the table, on the FILE menu, click **Close**.
- ⇒ Close the **Standard tables** dialog box.

Step 11. Iterative refinement for moduli of subgrade reaction

- ⇒ To visualize analysis results by DCL, click **Results by DCL** button  on the Status bar.
- ⇒ To present mosaic plot R_z (soil pressure), on the **Results** ribbon tab, on the **Stress in plates and solids** panel, click **Stress mosaic plot R_z** .
- ⇒ Then evaluate stress mosaic plot R_z from every DCL. To do this, switch to the next combination (in a manner similar to changing the load case No.). Then find out the worst combination (in this case it will be combination 2).
- ⇒ Make sure that **Stress mosaic plot R_z** button  is active for the second DCL. Then on the **Results** ribbon tab, on the **Tools** panel, click **Apply soil pressure** button .
- ⇒ In the **Apply soil pressure** dialog box (see Fig.9.32), when **All elements** option is selected, click **OK**.

Figure 9.32. **Apply soil pressure** dialog box

⇒ In the warning box (see Fig.9.33), click **OK**.

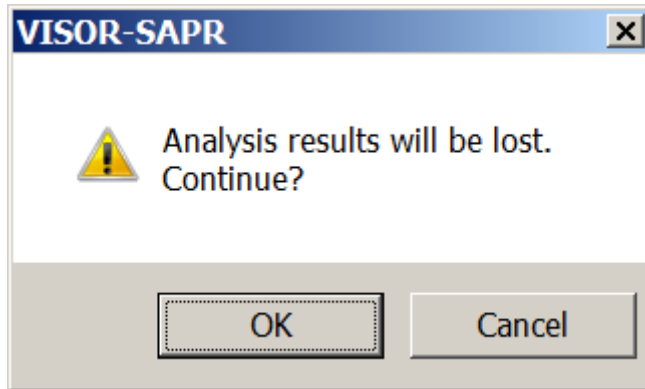







Figure 9.33. Warning box

- ⇒ The system automatically goes back to the mode of creating design model and analysis results become unavailable.
- ⇒ To carry out analysis, on the **Analysis** ribbon tab, select the **Analysis** panel and in the **Analyse** drop-down list, click **Complete analysis** .
- ⇒ In the **Warning** box (see Fig.9.29) that appears on the screen, make sure that **Recalculate moduli of subgrade reaction C1 and C2 for elastic foundation by soil model** is selected and click **OK**.
- ⇒ When analysis is complete, to present stress mosaic plot R_z (soil pressure), on the **Results** ribbon tab, on the **Stress in plates and solids** panel, click **Stress mosaic plot R_z** . Stress mosaic plot should be presented for the second DCL.
- ⇒ To compare obtained values of soil pressure R_z with the load on soil P_z , on the **WINDOW** menu, click **New window** and then on the **WINDOW** menu, click **Arrange icons**.
- ⇒ In the new window with design model, to present mosaic plot of uniformly distributed load on soil P_z , on the **Advanced results** ribbon tab, on the **Subgrade moduli** panel, click **Mosaic plot P_z** .
- ⇒ For the next refinement of subgrade moduli, make sure that **Stress mosaic plot R_z**  option is active for the second DCL and then on the **Results** ribbon tab, on the **Tools** panel, click **Apply soil pressure** button .
- ⇒ In the **Apply soil pressure** dialog box (see Fig.9.32), when **All elements** option is selected, click **OK**.
- ⇒ In the new warning box (see Fig.9.33), click **OK**.
- ⇒ Recalculation of the problem and comparison of obtained values of soil pressure R_z to the load on soil P_z should be made in a manner identical to the one described above.



The structural engineer decides in each case whether refinement of subgrade moduli is required. It is recommended that iterations for refinement of subgrade moduli are made up to the state when difference between values of soil pressure R_z and load on soil P_z does not exceed 5% (but not more than 5 iterations). One iteration is taken to mean applying soil pressure and recalculation of the problem with new value of load on soil.