

Example 4. Analysis of 3D framework of the structure with base slab on elastic foundation

In this lesson you will learn how to:

- generate design model;
- define elastic foundation;
- deal with design options;
- determine reinforcement for plate elements of the framework;
- select and check steel sections of bars in the framework;
- define static loads and earthquake load.
- define DCF and DCL tables.

Description:

Model of the building structure is presented in Fig.4.1.

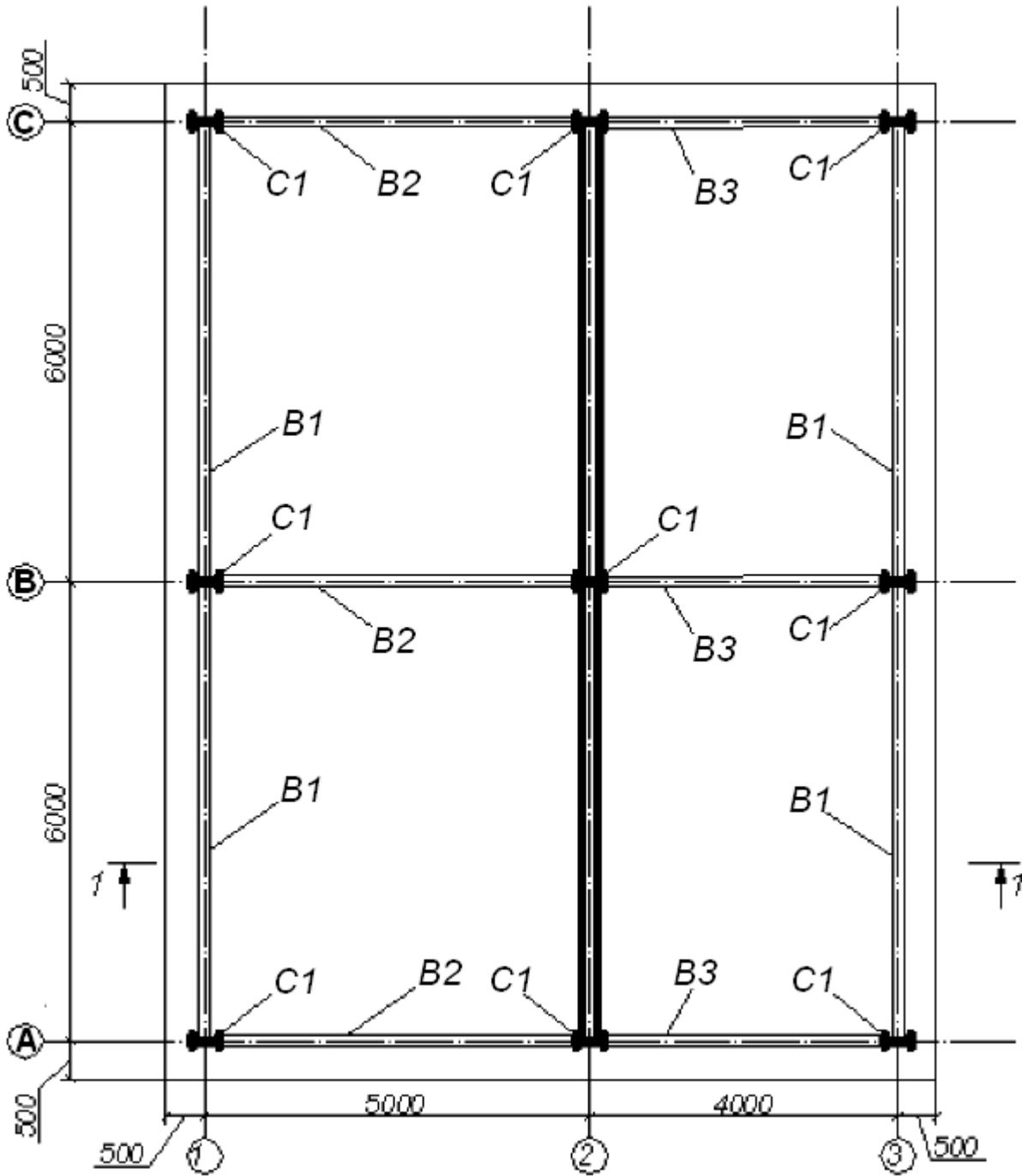
3D structure with base slab on elastic foundation with modulus of subgrade reaction $C1 = 1000 \text{ t/m}^3$.

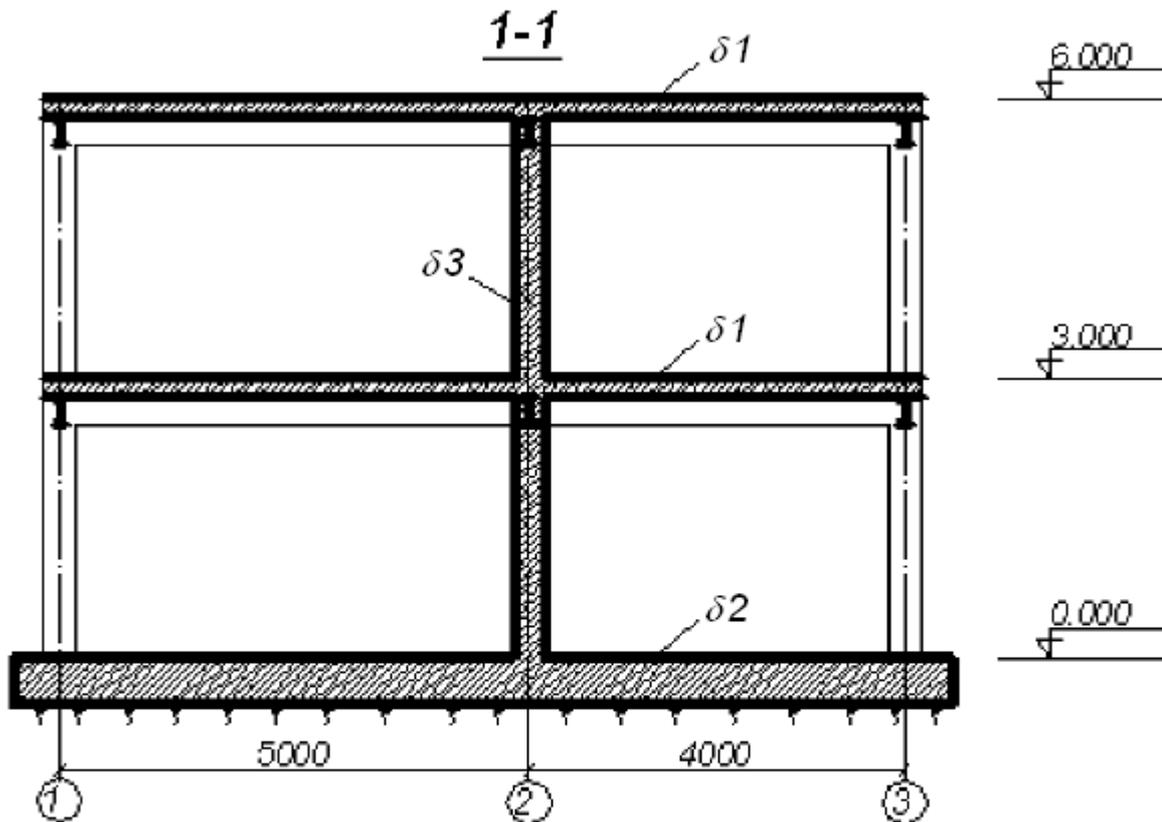
Material for frame – steel, material for slabs and diaphragm - reinforced concrete B30.

Analysis is performed for FE mesh 18×24 .

Loads:

- load case 1 – dead weight;
- load case 2 – uniformly distributed load $g_1 = 1.5 \text{ t/m}^2$ applied to the floor slabs of the 1st and 2nd storeys; uniformly distributed load $g_2 = 2.0 \text{ t/m}^2$ applied to foundation;
- load case 3 – snow load $g_3 = 0.08 \text{ t/m}^2$.
- load case 4 – earthquake load. Seismicity of the site is 7 units of magnitude, soil category 1. Unfavourable direction of earthquake load is along the smaller side of the structure.





C1 - 35 K1

B1, B2, B3 - 30B1

$\delta 1$ - 200mm

$\delta 2$ - 500mm

$\delta 3$ - 300mm

Figure 4.1 Model of the building structure

Sections of elements of the frame:

- beams – I-section with parallel edges of flanges (beam-type), shape **30B1** (**Двутавр с параллельными гранями полок типа Б (балочный)**);
- columns – I-section with parallel edges of flanges, column-type, shape **35K1** (**Двутавр с параллельными гранями полок типа К (колонный)**);
- floor slabs – thickness 200 mm;
- diaphragm – thickness 300 mm;
- foundation – base slab of thickness 500 mm.

- ⇒ On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2014** and then click **LIRA-SAPR 2014**.

Step 1. Creating new problem

- ⇒ On the FILE menu, click **New** (button  on the toolbar).
- ⇒ In the **Model type** dialog box (see Fig.4.2) specify the following data:
- problem name – **Example4** (problem code by default coincides with the problem name)
 - model type – **5 – Six degrees of freedom per node** (translations X, Y, Z and rotations Ux, Uy, Uz).
- ⇒ Click **OK** .

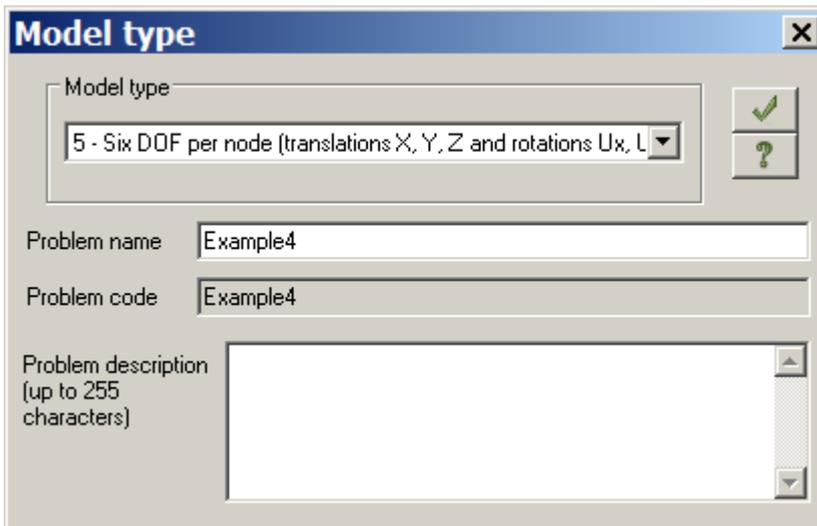


Figure 4.2 **Model type** dialog box



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.

Step 2. Generating model geometry

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** drop-down list and click the **Create frame**  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(m)	N	L(m)	N
5	1	3	2
4	1		
 - other parameters remain by default (see Fig.4.3).
- ⇒ Click **Apply** .

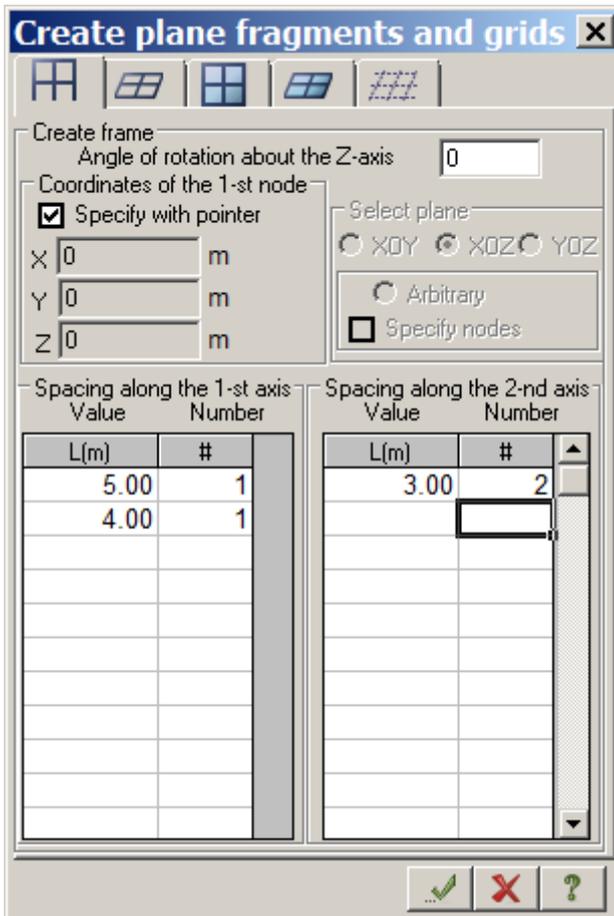


Figure 4.3 Create plane fragments and grids dialog box

- ⇒ Under **Coordinates of the 1st node**, clear the **Specify with pointer** check box and specify coordinates for the first node of the fragment:
 - X(m) Y(m) Z(m)
0 6 0.
- ⇒ Click **Apply** .
- ⇒ Then specify coordinates for the first node of the new fragment:
 - X(m) Y(m) Z(m)
0 12 0.
- ⇒ 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, select the **Node numbers** check box on the **Nodes** tab.
- ⇒ Click **Redraw** .

To define floor slab of the 1st storey:

- ⇒ In the **Create plane fragments and grids** dialog box, select **Create slab** tab.
- ⇒ Under **Coordinates of the 1st node**, select the **Specify with pointer** check box and specify node No.4 with the pointer (the node will be coloured pink and its coordinates will be displayed in the dialog box) (see Fig.4.4).
- ⇒ In the same dialog box, define the following parameters of the floor slab:

- spacing along the first axis: spacing along the second axis:

L(m)	N	L(m)	N
0.5	18	0.5	24

⇒ Click **Apply** .

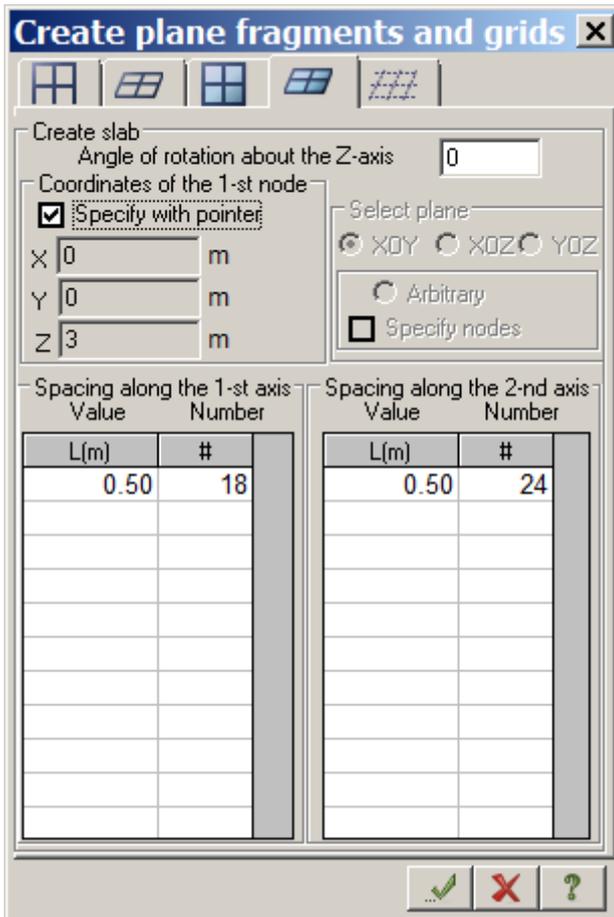


Figure 4.4 Create plane fragments and grids dialog box

To edit the model:

- ⇒ To select horizontal bar elements of the greater length, 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 FE geometry** tab (the fourth tab) (see Fig.4.5) and specify the following data:
 - in the **Criterion** box, select **Length of bar**;
 - under **Range of values**, click **Discretely** and specify the value **5**.
- ⇒ Click **Apply**.

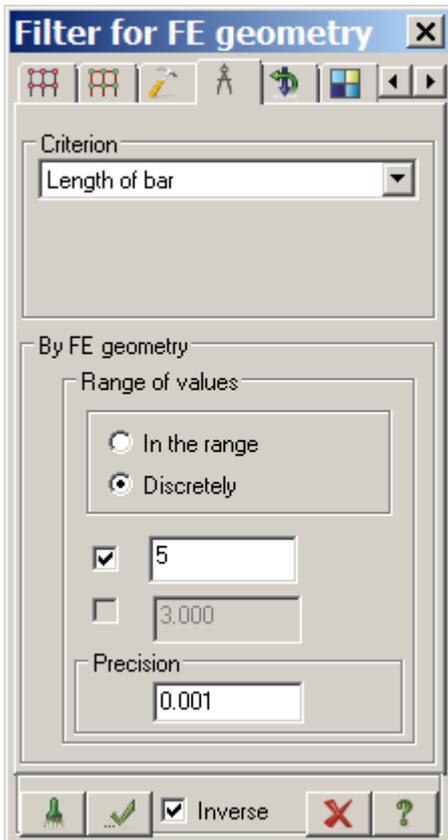


Figure 4.5 PolyFilter dialog box (Filter for FE geometry)

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click **Add bar** .
- ⇒ In the **Add element** dialog box, select the fifth tab **Divide into N equal parts** (see Fig.4.6) and specify the value:
 - N = 10.
- ⇒ Click **Apply**.

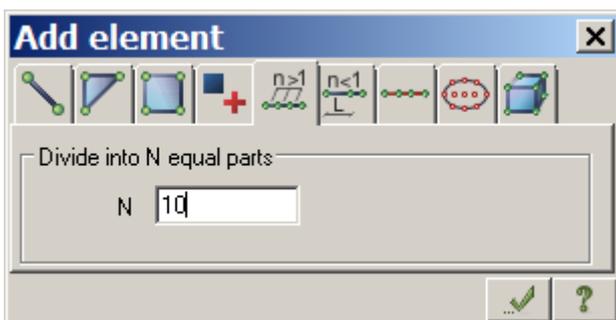


Figure 4.6 Add element dialog box



It is possible to present the **Add element** dialog box with the **Divide into N equal parts** tab open. To do this, on the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click **Divide into N equal parts**  command.

- ⇒ In the **PolyFilter** dialog box, select the **Filter for FE geometry** tab (the fourth tab).
- ⇒ To select horizontal bar elements of the smaller length, specify the following data:
 - in the **Criterion** box, select **Length of bar**;
 - under **Range of values**, click **Discretely** and specify the value **4**.

- ⇒ Click **Apply**.
- ⇒ To divide selected bars into 8 parts, in the **Add element** dialog box (see Fig.4.6), specify the value:
 - N = 8.
- ⇒ Click **Apply**.



For combined behaviour of slab and beam, bar elements are divided in a similar way as in slab.

To pack the model:

- ⇒ On the **Create and edit** ribbon tab, on the **Edit** panel, click **Pack model** .
- ⇒ In the **Pack model** dialog box (see Fig.4.7), click **Apply** . It is necessary to pack the model in order to 'throw together' coincident nodes and elements and to eliminate (that is, to remove completely) deleted nodes and elements from design model.

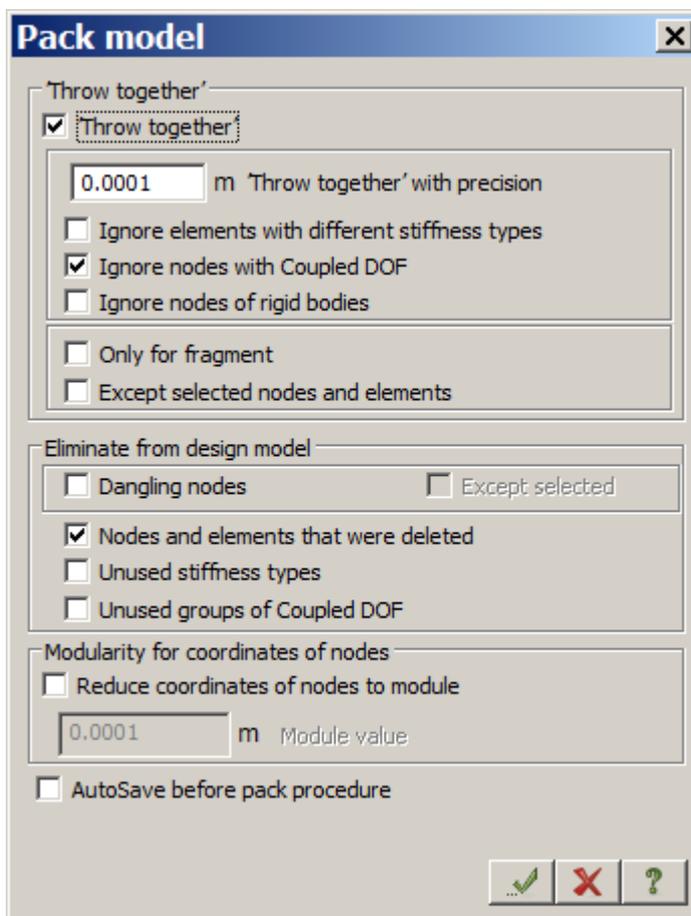


Figure 4.7 **Pack model** dialog box



***Pack model** dialog box is used to manage pack parameters after **Merge**, **Copy** and other commands with model geometry.*

To add beams and floor slab:

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click **Add bar** .
- ⇒ Under **Add bar**, select the **Specify nodes with pointer** and **Consider intermediate nodes** check boxes.

- ⇒ Add bars between nodes of the extreme left edge of floor slab (nodes No.24 and 461) and nodes of the extreme right edge of floor slab (nodes No.5 and 20). To do this, specify these pairs of nodes in sequence (in this case the rubber-band line is automatically stretched between the nodes that you select).
- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **PolyFilter** .
- ⇒ In the **PolyFilter** dialog box, select the **Section and cut off** tab (the last but one tab) (see Fig.4.8).
- ⇒ To select the cutting plane, under **Cutting plane**, click XOY (by default, under **Include, Nodes** and **Elements** check boxes are selected; under **Select mode**, **Section by plane** option is selected; **Specify node of the plane** check box is selected).
- ⇒ Specify with the pointer any node of floor slab of the first storey (the node will be coloured black).
- ⇒ Click **Apply**.

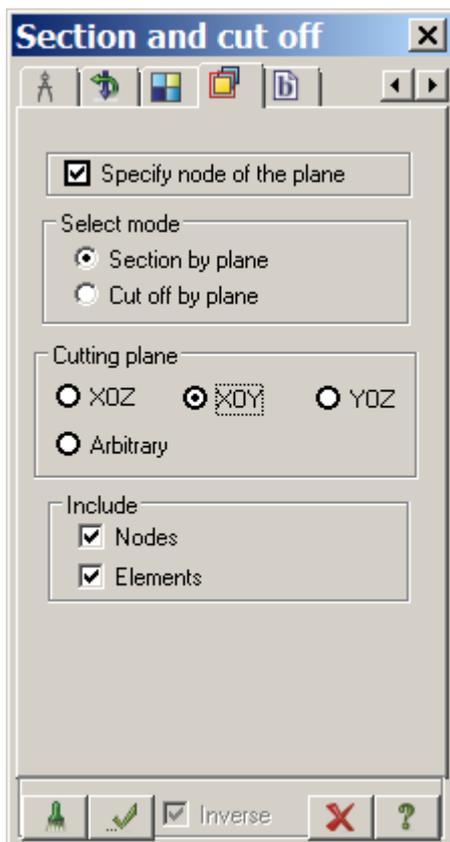


Figure 4.8 **PolyFilter** dialog box (Section and cut off)

- ⇒ On the **Create and edit** ribbon tab, on the **Edit** panel, point to **Copy** drop-down list and click **By one node** .
- ⇒ The **Copy objects** dialog box is presented with the **Copy by one node** tab open (see Fig.4.9).
- ⇒ Specify the node No.24 on the model with the pointer.
- ⇒ Then click the node where the fragment should be placed (the upper-left node of the frame - node No.6).

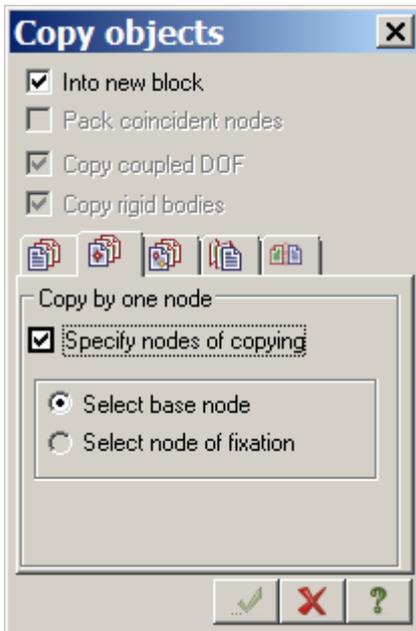


Figure 4.9 Copy objects dialog box

To define diaphragm:

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** drop-down list and click the **Create wall-beam**  command.
- ⇒ In the **Create plane fragments and grids** dialog box, specify **Angle of rotation about the Z-axis** equal to 90 degrees.
- ⇒ Under **Coordinates of the 1st node**, select the **Specify with pointer** check box and specify node No.2 with the pointer (the node will be coloured pink and its coordinates will be displayed in the dialog box).
- ⇒ In the same dialog box specify the following parameters for the diaphragm:
 - spacing along the first axis: spacing along the second axis:

L(m)	N	L(m)	N
0.5	24	0.5	12
- ⇒ Click **Apply** .
- ⇒ To unselect all nodes and elements, on the **Select** toolbar, click **Unselect all** .
- ⇒ To switch to projection on the XOZ-plane, on the **Projection** toolbar (by default, it is displayed at the bottom of the screen), click **Projection on XOZ-plane** .
- ⇒ On the **Select** toolbar, click **Select vertical elements** . Then select columns at the place where diaphragm is located.
- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click **Divide into N equal parts**  command.
- ⇒ In the **Add element** dialog box, on the **Divide into N equal parts** tab, specify the value:
 - N = 6.
- ⇒ Click **Apply**.
- ⇒ To present the model in isometric projection, on the **Projection** toolbar, click **Front isometric projection** .

To define base slab:

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** drop-down list and click the **Create slab**  command.
- ⇒ In the **Create plane fragments and grids** dialog box, under **Coordinates of the 1st node**, clear the **Specify with pointer** check box and define coordinates of the first node:

X(m)	Y(m)	Z(m)
-0.5	-0.5	0
- ⇒ In the same dialog box define the following parameters for the base slab:
 - spacing along the first axis: spacing along the second axis:

L(m)	N	L(m)	N
0.5	20	0.5	26
- ⇒ Click **Apply**  .
- ⇒ In the **Display** dialog box, clear the **Node numbers** check box on the **Nodes** tab.
- ⇒ Click **Redraw**  .
- ⇒ On the **Create and edit** ribbon tab, on the **Edit** panel, click **Pack model**  .
- ⇒ In the **Pack model** dialog box, click **Apply**  .
- ⇒ To unselect elements, on the **Select** toolbar, click **Unselect all** button  .

The model of framework is presented in Fig.4.10.

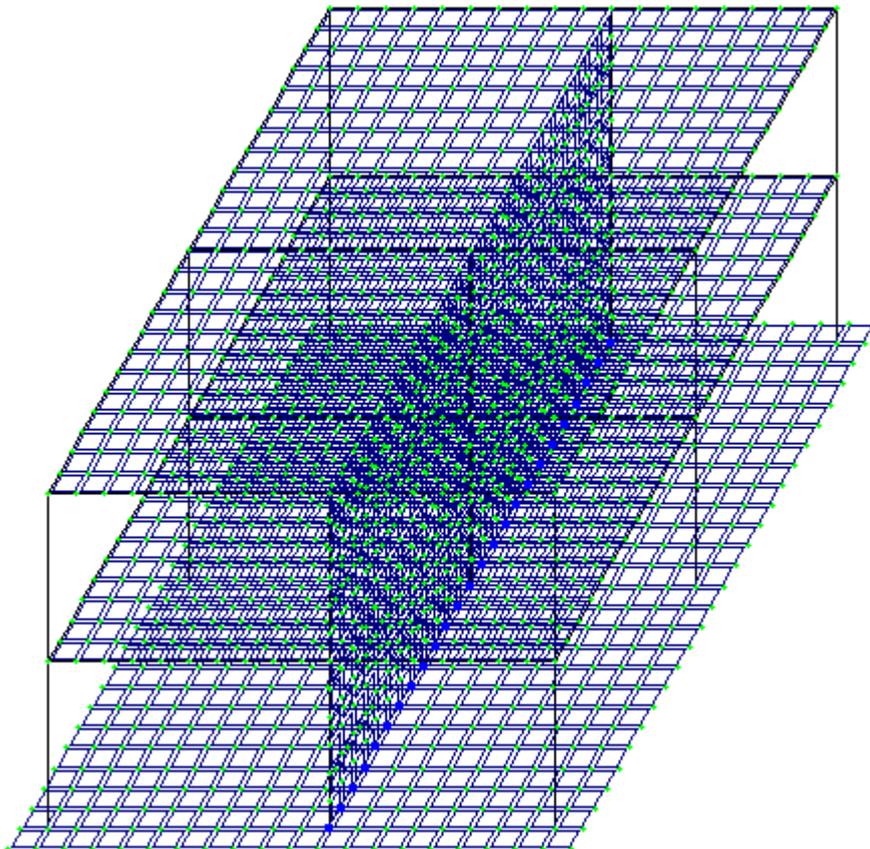


Figure 4.10 Design model of framework

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 – **Example4**;
 - location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

Step 3. Defining design options

- ⇒ On the **More edit options** ribbon tab, on the **Design** panel, click **Design options for main model** command .
- ⇒ In the **Design options** dialog box (see Fig.4.11), define parameters for the first design option:
- in the **Analysis of sections by** list, select **DCF**;
 - other parameters remain by default.
- ⇒ Click **Apply** .

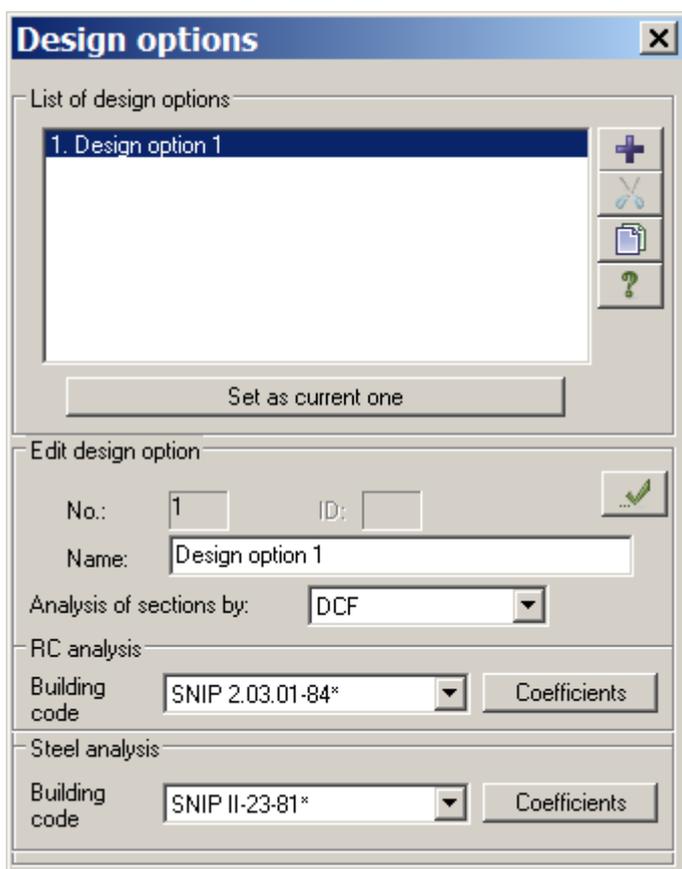


Figure 4.11 **Design options** dialog box



To define new design option, click **Create new design option** button  (by default, all parameters of the new design option will obtain values defined in the **Analysis parameters** dialog box on the appropriate tabs).

Then define the following data:

- name of design option;
- building codes for analyses of reinforced concrete (RC) and steel structures;
- type for analysis of sections (by DCF, DCL or Forces).

To input data for design option, click **Apply** .

To assign the selected design option as active one in the graphical environment, select it in the list and click **Set as current type** or double-click appropriate row in the list of design options.

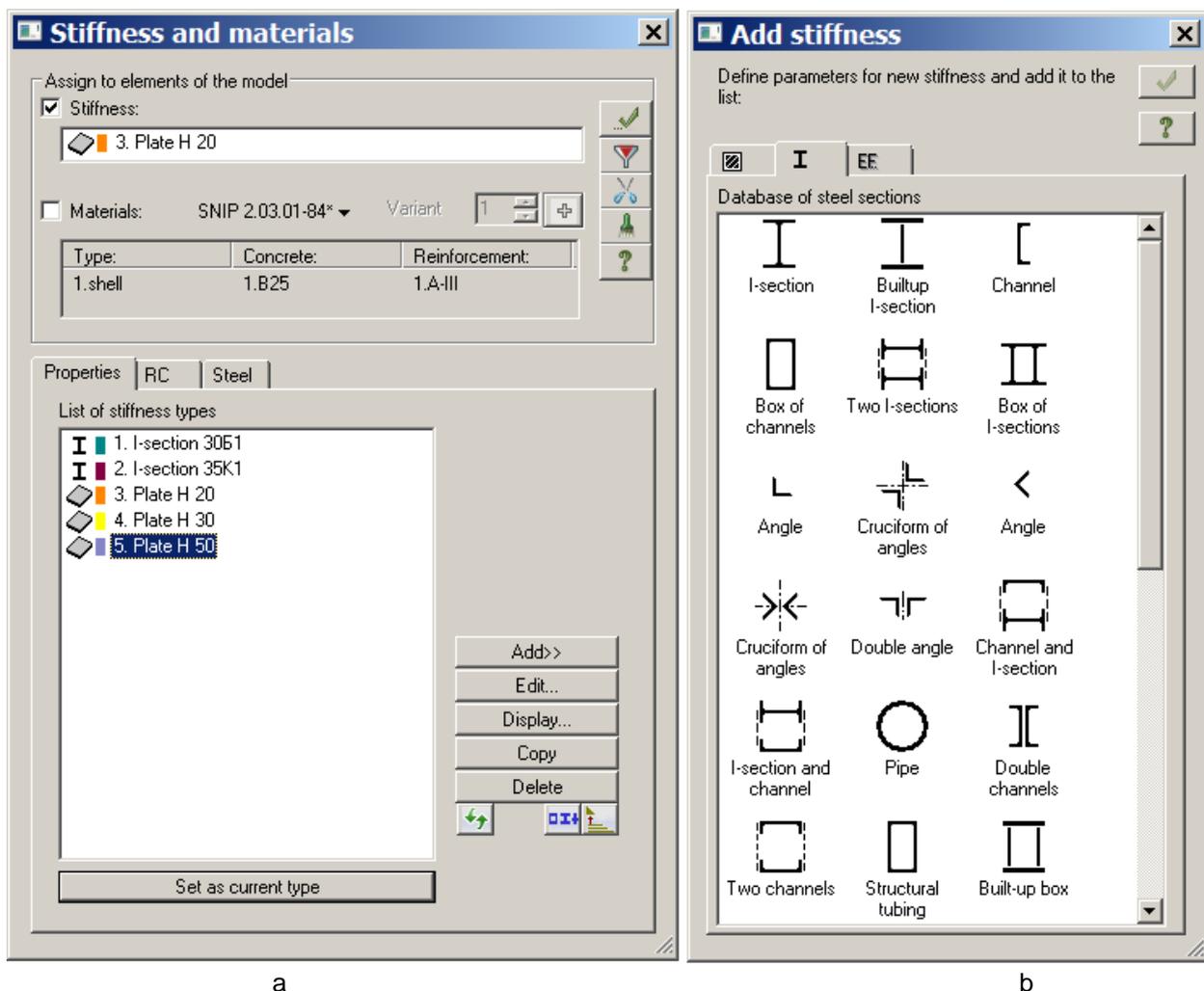
Materials for appropriate design option may be defined in the **Stiffness and materials** dialog box (see Fig.4.12a).

To define the second design option:

- ⇒ To define the second design option, in the **Design options** dialog box, click **Create new design option** button .
- ⇒ Then define parameters for the second design option:
 - under **Steel analysis**, define building code SP 16.13330.2011;
 - in the **Analysis of sections by** list, select **DCL**;
 - other parameters remain by default.
- ⇒ Click **Apply** .
- ⇒ To assign the first design option as a current one, in the **List of design options**, select appropriate row and click **Set as current type**.
- ⇒ To close the **Design options** dialog box, click the **Close** button.

Step 4. Defining material properties to elements of the modelTo 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.4.12a), click **Add**. The dialog box expands to display the library of stiffness parameters. In the **Add stiffness** dialog box (see Fig.4.12b), select the **Database of steel sections** tab (the second tab).

Figure 4.12 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ Double-click the **I-section** icon in the list. The **Steel cross-section** dialog box opens. In this dialog box you can define material properties for selected type of the section.
- ⇒ In the **Steel cross-section** dialog box (see Fig.4.13), specify the following parameters for I-section:
 - in the **Steel table** box, click **Двутавр с параллельными гранями полок типа Б (балочный). Актуализированный** ;
 - in the **Shape** box, click **30Б1** .
- ⇒ Click **OK**.

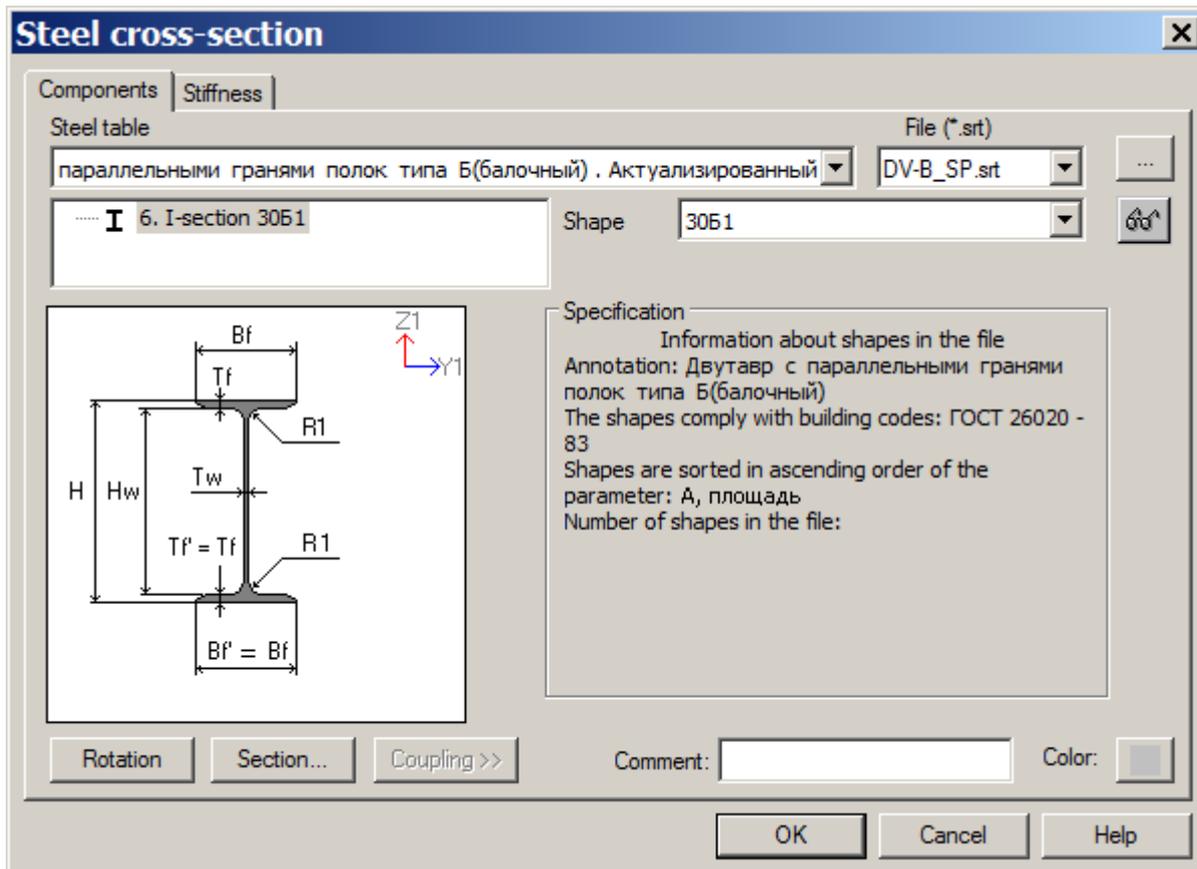


Figure 4.13 **Steel cross-section** dialog box

- ⇒ In the **Add stiffness** dialog box (see Fig.4.12b), on the **Database of steel sections** (the second tab), double-click the **I-section** icon once again.
- ⇒ In the **Steel cross-section** dialog box specify the following parameters for I-section:
 - in the **Steel table** box, click **Двутавр с параллельными гранями полок типа К (колонный). Актуализированный** ;
 - in the **Shape** box, click **35К1** .
- ⇒ Click **OK**.
- ⇒ In the **Add stiffness** dialog box, click the tab with numerical description of stiffness (the third tab).
- ⇒ Double-click the **Plates** icon in the list. The **Specify stiffness for plates** dialog box opens. In this dialog box you can define material properties for selected type of the section.
- ⇒ In the **Specify stiffness for plates** dialog box (see Fig.4.14), specify the following parameters for **Plates** (for floor slab):
 - modulus of elasticity – $E = 3e6 \text{ t/m}^2$ (for the U.S. keyboard layout);
 - Poisson's ratio – $V = 0.2$;
 - thickness – $H = 20 \text{ cm}$;

- unit weight of material – $R_o = 2.75 \text{ t/m}^3$.

⇒ To confirm the specified data, click **OK** .

Figure 4.14 **Specify stiffness for plates** dialog box

- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select '3.Plate H 20'.
- ⇒ Click **Copy** two times.
- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select '4.Plate H 20'.
- ⇒ Click **Edit**.
- ⇒ In another **Specify stiffness for plates** dialog box specify parameter (for diaphragm):
 - thickness - $H = 30 \text{ cm}$.
- ⇒ Click **OK**  .
- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select '5.Plate H 20'.
- ⇒ Click **Edit**.
- ⇒ In another **Specify stiffness for plates** dialog box specify the following parameter (for foundation slab):
 - thickness - $H = 50 \text{ cm}$.
- ⇒ Click **OK**.
- ⇒ To hide the library of stiffness parameters, click **Add** in the **Stiffness and materials** dialog box.

To define materials for reinforced concrete (RC) structures:

- ⇒ To define parameters for reinforced concrete structures, in the **Stiffness and materials** dialog box, click the second tab **Reinforced concrete (RC)**.
- ⇒ Select **Type** option and click **Add**.
- ⇒ In the **General parameters** dialog box (see Fig.4.15), define the following parameters for plates:
 - in the **Module of reinforcement** list, select **Shell**;

- in the **Comment** box, type comment - **Shells**;
- other parameters remain by default.

⇒ Click **OK** .

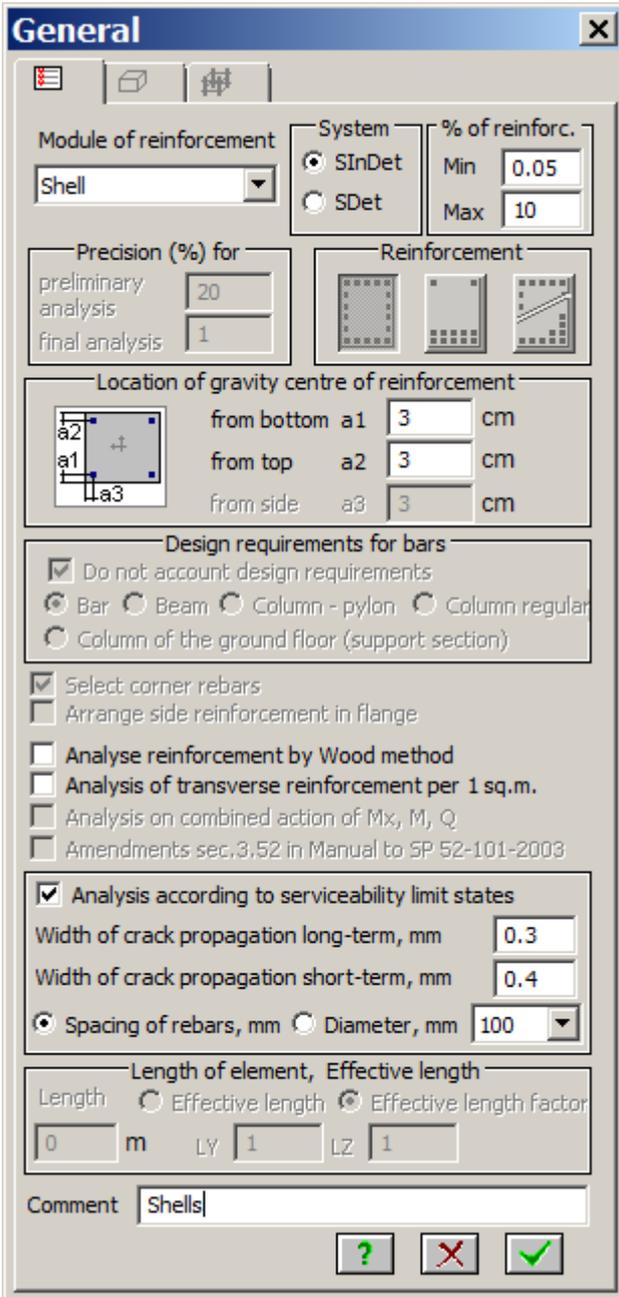


Figure 4.15 **General parameters** dialog box

- ⇒ In the **Stiffness and materials** dialog box, select the **Concrete** option.
- ⇒ Click **Default** (in this case, concrete B25 is accepted by default).
- ⇒ In the same dialog box, select the **Reinforcement** option.
- ⇒ Click **Default** (in this case, reinforcement A-III is accepted by default).



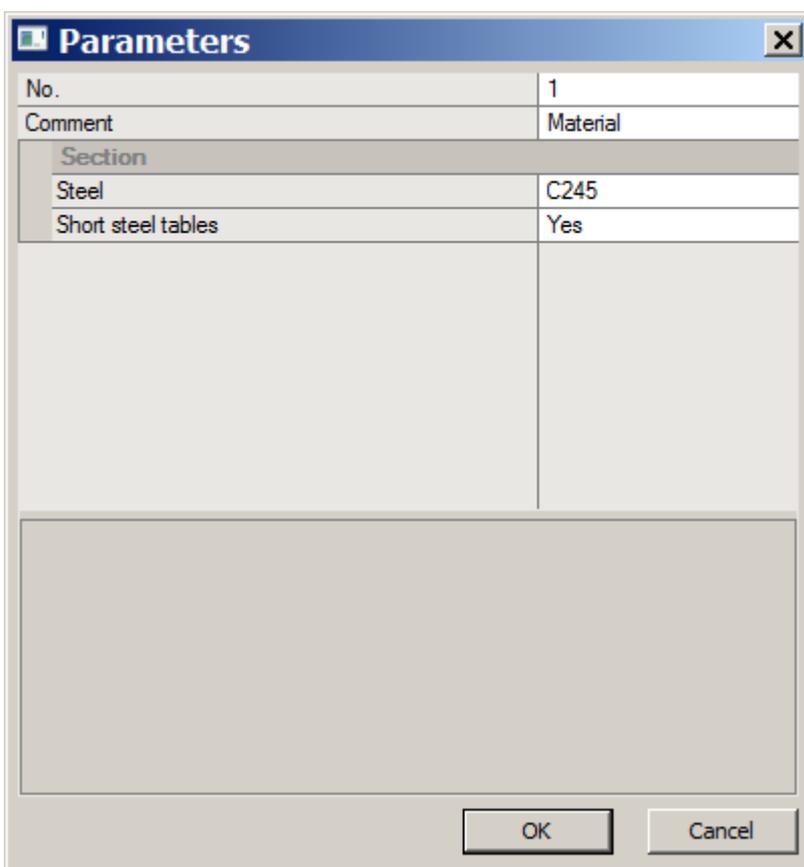
*If there are several design options in the problem, to switch to another design option, in the **Stiffness and materials** dialog box (see Fig.4.12a) use the **No. of current design option** box (when **Materials** check box is selected.)*

Every design option has its own parameters for materials.

To create new design option, click **Create new design option** button . Then in the **Design options** dialog box (see Fig.4.11), define all necessary parameters for new design option.

To define materials for the first design option for steel structures:

- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select type '1.I-section 3051' and 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. To assign current type of stiffness, you could also double-click appropriate row in the list.
- ⇒ Then, to assign materials for steel structures, in the **Stiffness and materials** dialog box, select the **Steel** tab (the third tab).
- ⇒ Select **Material** option and click **Add**.
- ⇒ In the **Parameters** dialog box (see Fig.4.16), in the **Steel** list box, define the steel grade C245 (it will be applied to beams and columns of the first and second design options).
- ⇒ To confirm the data, click **OK**.



No.	1
Comment	Material
Section	
Steel	C245
Short steel tables	Yes

Figure 4.16 **Parameters** (for materials) dialog box

- ⇒ In the **Stiffness and materials** dialog box, select **Additional parameters** option and click **Add**.
- ⇒ In the **Parameters** dialog box (see Fig.4.17), define the following parameters for beams:
 - under **Element type**, click **Beam**;
 - in the **Comment** line, type **Beams**;
 - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.

Parameters	
Building code	SNIP II-23-81*
No.	1
Comment	Beams
Element type	
Truss	<input type="radio"/>
Column	<input type="radio"/>
Beam	<input checked="" type="radio"/>
Service conditions factors and safety factor	
Yc buckling	0.95
Yc strength	1
Yn	1
Analysis is carried out	
within elasticity limits	<input checked="" type="radio"/>
with account of plasticity	<input type="radio"/>
Pure bending	<input type="checkbox"/>
Stiffeners	
place stiffeners	<input type="checkbox"/>
stiff. spacing, m	0
Deflection analysis	
Span length L, m	Auto
Maximum allowed deflection	1/400
Cantilever	<input type="checkbox"/>
Data for bucking analysis	
Lef b, m	0
use length factors	<input type="checkbox"/>
Comment Arbitrary text that describes this set of additional parameters	
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Figure 4.17 Parameters (for beams) dialog box

- ⇒ In the **Stiffness and materials** dialog box, click **Add** once more.
- ⇒ In the **Parameters** dialog box (see Fig.4.18), define the following parameters for columns:
 - under **Element type**, click **Column**;
 - under **Effective lengths**, select the **Use length factor** check box;
 - define effective length factor relative to Z1-axis $K_z=1$;
 - define effective length factor relative to Y1-axis $K_y=1$;
 - define effective length factor for check by lateral-torsional buckling (calculation of factor F_b) $K_b=0.85$;
 - in the **Comment** line, type **Columns**;
 - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.

Parameters	
Building code	SNIP II-23-81*
No.	2
Comment	Columns
Element type	
Truss	<input type="radio"/>
Column	<input checked="" type="radio"/>
Beam	<input type="radio"/>
Service conditions factors and safety factor	
Yc buckling	1
Yc strength	1
Yn	1
Ultimate slenderness	
principal column	<input checked="" type="radio"/>
secondary column	<input type="radio"/>
other	<input type="radio"/>
In compression	180-60a
In tension	300
Analysis is carried out	
within elasticity limits	<input checked="" type="radio"/>
with account of plasticity	<input type="radio"/>
Effective lengths	
Kz	1
Ky	1
Kb	0.85
use length factors	<input checked="" type="checkbox"/>
Comment Arbitrary text that describes this set of additional parameters	
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Figure 4.18 Parameters (for columns) dialog box

To define materials for the second design option for steel structures:

- ⇒ To switch to the second design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box define number 2.
- ⇒ Then in the **Stiffness and materials** dialog box, select **Additional parameters** option and click **Add**.
- ⇒ In the **Parameters** dialog box, define the following parameters for beams:
 - under **Element type**, click **Beam**;
 - in the **Comment** line, type **Beams**;
 - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.
- ⇒ In the **Stiffness and materials** dialog box, click **Add** once more.

- ⇒ In the **Parameters** dialog box, define the following parameters for columns:
 - under **Element type**, click **Column**;
 - under **Effective lengths**, select the **Use length factor** check box;
 - define effective length factor relative to Z1-axis $K_z=1$;
 - define effective length factor relative to Y1-axis $K_y=1$;
 - define effective length factor for check by lateral-torsional buckling (calculation of factor F_b) $K_b=0.85$;
 - in the **Comment** line, type **Columns**;
 - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.

To assign material properties to elements of the model:

- ⇒ In the **Stiffness and materials** dialog box, in the list of additional parameters for steel structures, select '**3.Beams**'.
- ⇒ Click **Set as current type**. Selected type of additional parameters will be displayed in the **Materials** box under **Assign to elements of the model**. You can also specify the current type of additional parameters by double-clicking the necessary type in the list.
- ⇒ On the **Select** toolbar, click **Select horizontal bars** button .
- ⇒ Select all horizontal elements of the model. Selected elements will be coloured red.



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

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



The same material may be assigned to elements of design model regardless of the section type, if this material is consistent with the section type. Otherwise, it is not possible to assign material and you will see appropriate message.

- ⇒ On the **Select** toolbar, click **Select horizontal bars**  in order to make this command not active.
- ⇒ In the **Stiffness and materials** dialog box, in the list of additional parameters for steel structures, select '**4.Columns**'.
- ⇒ Click **Set as current type**.
- ⇒ In the **Stiffness and materials** dialog box, on the **Properties** tab (the first tab), select '2.I-section 35K1' in the **List of stiffness types**.
- ⇒ Click **Set as current type**.
- ⇒ On the **Select** toolbar, click **Select vertical bars** button .
- ⇒ Select all vertical elements of the model with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ To switch to the first design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box define number 1.
- ⇒ To assign materials to *steel structures* for the first design option, clear the **Stiffness** box in the **Assign to elements of the model** area.
- ⇒ In the **Stiffness and materials** dialog box, in the list of additional parameters for steel structures, select '**2.Columns**'.

- ⇒ Click **Set as current type**.
- ⇒ Select all vertical elements of the model with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ On the **Select** toolbar, click **Select vertical bars**  in order to make this command not active.

- ⇒ In the **Stiffness and materials** dialog box, in the list of additional parameters for steel structures, select '**1.Beams**'.
- ⇒ Click **Set as current type**.
- ⇒ On the **Select** toolbar, click **Select horizontal bars** button .
- ⇒ Select all horizontal elements of the model.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .

- ⇒ To assign materials to *reinforced concrete structures* for the first design option, click the **Properties** tab (the first tab).
- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select stiffness type '3.Plate H20'.
- ⇒ Click **Set as current type**.
- ⇒ In the **Stiffness and materials** dialog box, select **RC** tab (the second tab). In this case, make sure that in the list of current materials the following data should be defined as current one: type – **1.shell**, concrete class – **1.B25** and class of reinforcement – **1.A-III**.
- ⇒ On the **Select** toolbar, point to **Select blocks** drop-down list and click **Select blocks** button .
- ⇒ Specify with the pointer any node or element of the floor slab on the 1st storey and then on the 2nd storey.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the dialog box with Warning message, click **OK**.
- ⇒ To unselect nodes and elements, on the **Select** toolbar, click **Unselect all** button .

- ⇒ Set the stiffness type '4.Plate H30' as a current one.
- ⇒ To select diaphragm, on the **Select** toolbar, click **PolyFilter** button .
- ⇒ In the **PolyFilter** dialog box, click the **Filter for elements** tab (the second tab).
- ⇒ On the **Filter for elements** tab (see Fig.4.19), select **By FE shape** check box and select **4-node FE (plates)** option in the list.
- ⇒ Then select **By orientation of FE** check box and define **II YOZ**.
- ⇒ In the **PolyFilter** dialog box, click **Apply**.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply**.

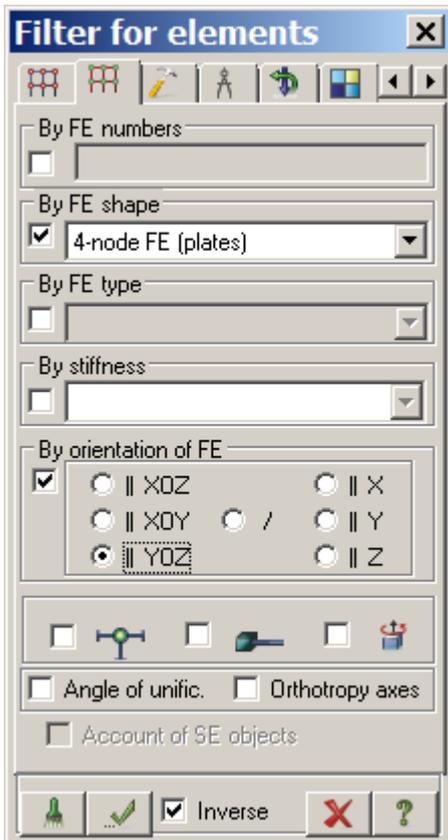


Figure 4.19 **Filter for elements** dialog box

- ⇒ Set the stiffness type '5.Plate H50' as a current one.
- ⇒ When the **Select blocks** command  is active, specify with the pointer any node or element of base slab.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply**.

Step 5. Defining parameters of elastic foundation

- ⇒ When the **Select blocks** command  is active, specify with the pointer any node or element of 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.4.20), make sure that the **Plates** check box and **Assign** option are selected. To define moduli of subgrade reaction, in the **C1z** box specify its value as equal to $C1z=1000 \text{ t/m}^3$.
- ⇒ Click **Apply** .

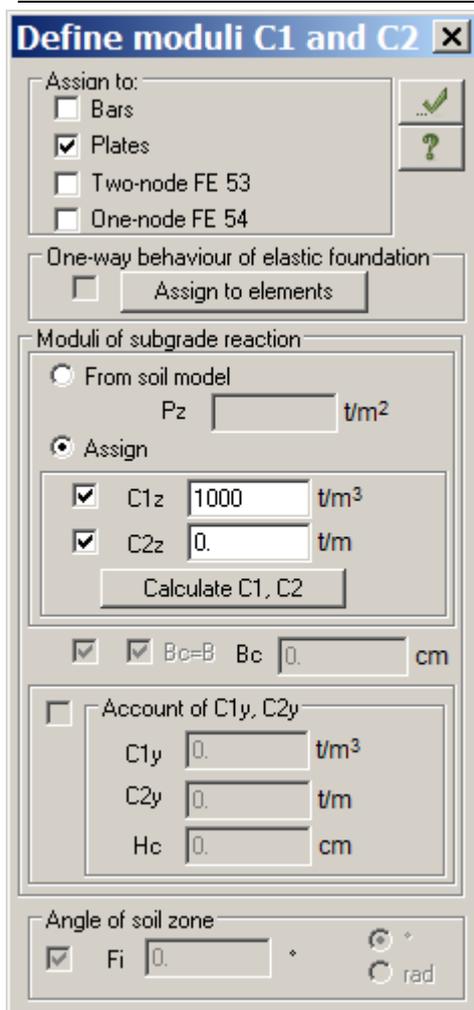


Figure 4.20 Define moduli C1 and C2 dialog box

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

Step 6. Defining boundary conditions



To avoid geometric instability in the XOY-plane, additional boundary conditions are defined for base slab.

To select nodes:

- ⇒ In the **PolyFilter** dialog box, click **Section and cut off** tab (the last tab).
- ⇒ Under **Cutting plane**, click YOZ.
- ⇒ Click any node that is common for diaphragm and base slab.
- ⇒ Click **Apply** .
- ⇒ To present on the screen only selected nodes and elements of the model, on the **Select** toolbar, click **Fragmentation** .
- ⇒ To present design model in projection on the YOZ-plane, on the **Projection** toolbar, click **Projection on YOZ-plane** .
- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button . Then select nodes that are common for diaphragm and foundation slab.

To define boundary conditions:

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Restraints** .
- ⇒ In the **Restraints on nodes** dialog box (see Fig.4.21), specify directions along which displacements of nodes are not allowed (X). To do this, select appropriate check boxes.
- ⇒ Click **Apply**  (the nodes will be coloured blue).



Figure 4.21 Restraints on nodes dialog box

- ⇒ Select the node that is common for middle column and base slab.
- ⇒ In the **Restraints on nodes** dialog box specify directions along which displacements of nodes are not allowed (Y, UZ). To do this, select appropriate check boxes.
- ⇒ Click **Apply**.
- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes**  in order to make this command not active.
- ⇒ To restore design model in initial view after fragmentation, on the **Select** toolbar, click **Restore model** .
- ⇒ To present the model in isometric projection, on the **Projection** toolbar, click **Front isometric projection** .

Step 7. Applying loadsTo 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.4.22), click **All elements** and specify **Load factor** as equal to 1. Then click **Apply**  (dead weight is automatically applied to elements).

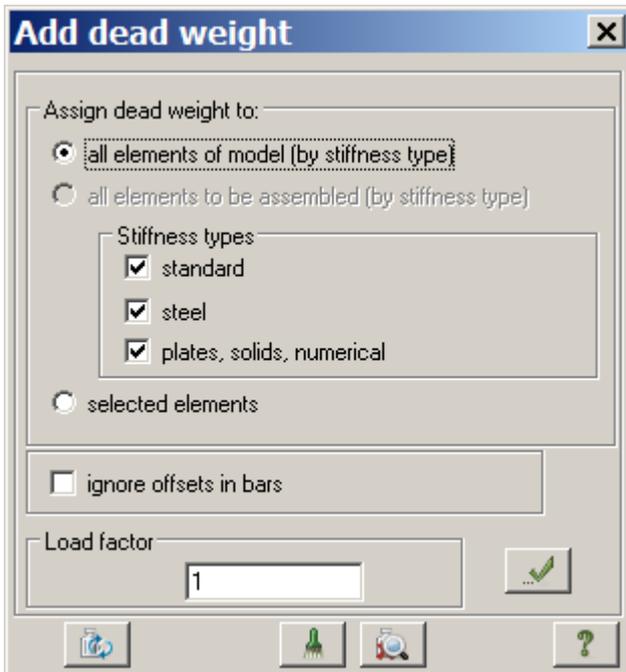


Figure 4.22 Add dead weight dialog box

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, click **Select blocks** button . Then select floor slabs of the 1st and the 2nd storey.
- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel, then select **Load on plates** command  from the **Loads on nodes and elements** drop-down list.
- ⇒ In the **Define loads** dialog box (see Fig.4.23), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

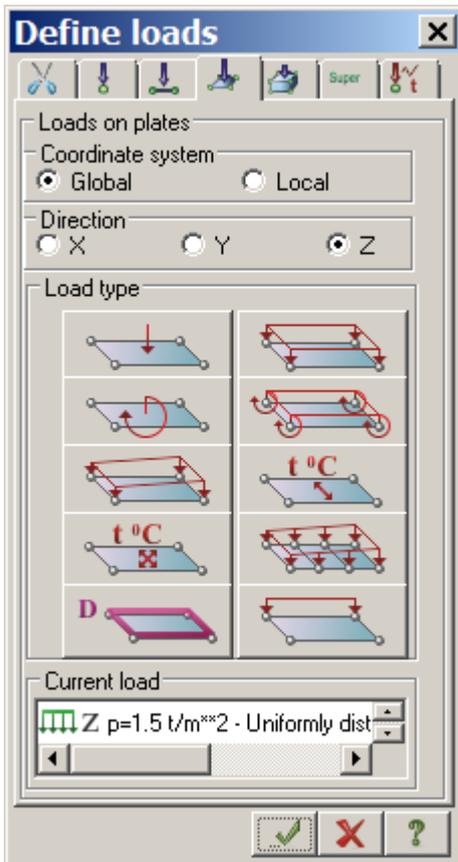
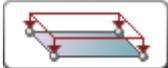


Figure 4.23 Define loads dialog box

- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 1.5 \text{ t/m}^2$ (see Fig.4.24).
- ⇒ Click **OK** .

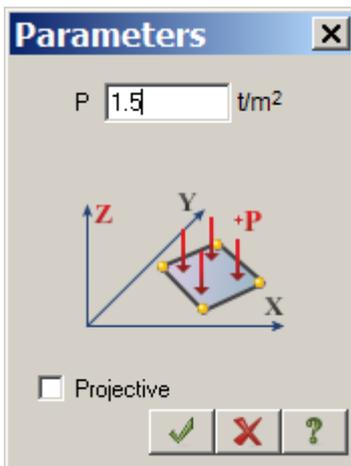


Figure 4.24 Load parameters dialog box

- ⇒ The **Warning** message box is displayed (see Fig.4.25). Click **OK**.

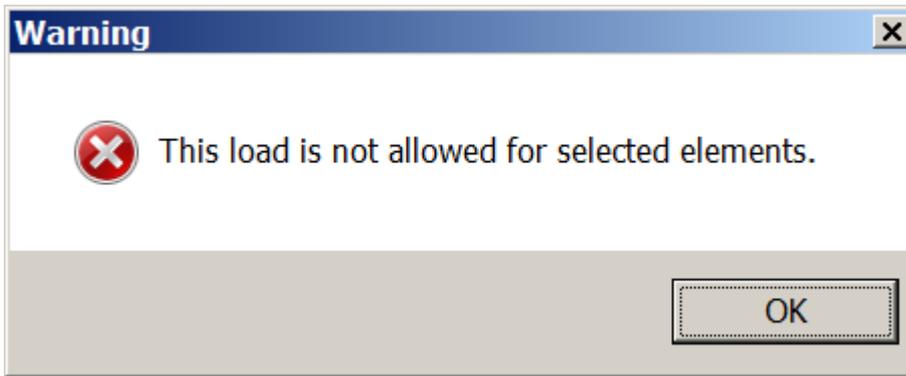


Figure 4.25 Warning message box



The warning message appears because when you select floor slabs, bars and plates are also selected at the same time. Load applied to plates is not allowed for selected bar elements.

- ⇒ To unselect nodes and elements, on the **Select** toolbar, click **Unselect all** button .
- ⇒ Select all elements of foundation slab when the **Select block** option is active.
- ⇒ In the **Define loads** dialog box, in the **Load type** area, click the **Uniformly distributed load** button.
- ⇒ In the **Load parameters** dialog box specify $P = 2 \text{ t/m}^2$.
- ⇒ Click **OK**.
- ⇒ In the **Define loads** dialog box, click **Apply** .

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.
- ⇒ Select floor slabs of the 2nd storey when the **Select block** option is active.
- ⇒ In the **Define loads** dialog box, in the **Load type** area, click the **Uniformly distributed load** button.
- ⇒ In the **Load parameters** dialog box specify $P = 0.08 \text{ t/m}^2$.
- ⇒ Click **OK**.
- ⇒ In the **Define loads** dialog box, click **Apply** .
- ⇒ The **Warning** message box is displayed. Click **OK**.
- ⇒ To unselect nodes and elements, on the **Select** toolbar, click **Unselect all** button .

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.4.26).
- ⇒ 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** .
- ⇒ To add the fourth load case, in the **List of load cases** area, click **Add load case (to the end)** .

- ⇒ For load case 4 – in the **Edit selected load case** area, in the **Type** box, select **Earthquake** and click **Apply** .

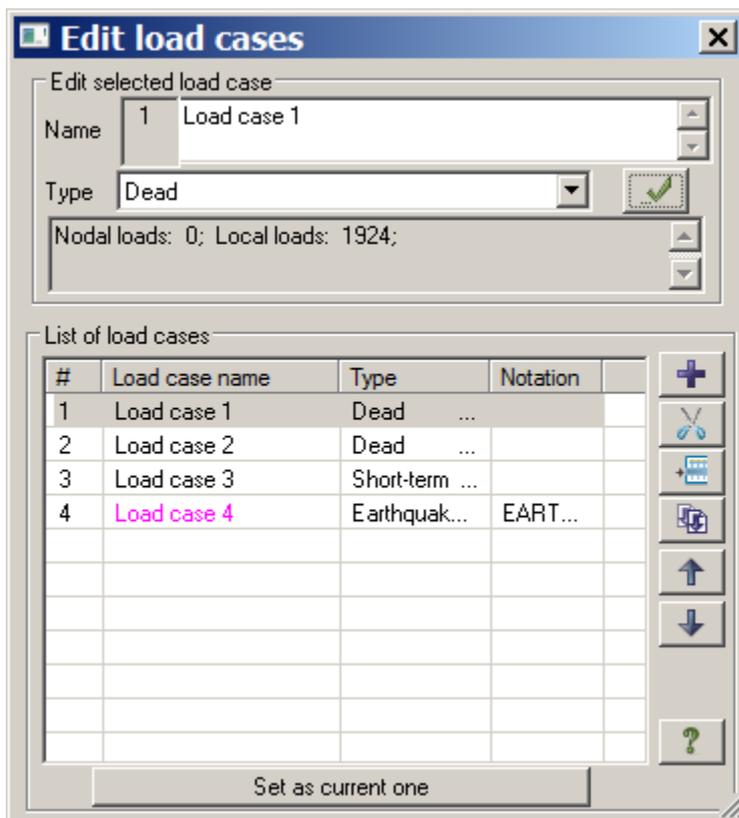


Figure 4.26 **Edit load cases** dialog box

Defining parameters for earthquake analysis of the frame

Step 8. Creating dynamic load cases from the static ones

- ⇒ On the **Analysis** ribbon tab, on the **Dynamics** panel, click **Account of static load cases** .
- ⇒ In the **Create dynamic load cases from the static ones** dialog box (see Fig.4.27), under **Generate mass matrix according to**, click **Load case (code 1)** and to create the first line of the summary table, specify the following data:
- dynamic load case No. – 4;
 - No. of corresponding static load case – 1;
 - conversion factor – 0.9.
- ⇒ Click **Add**.
- ⇒ To create the second line of the summary table, specify the following data:
- dynamic load case No. – 4;
 - No. of corresponding static load case – 2;
 - conversion factor – 0.9.
- ⇒ Click **Add**.
- ⇒ To create the third line of the summary table, specify the following data:
- dynamic load case No. – 4;
 - No. of corresponding static load case – 3;
 - conversion factor – 0.5.
- ⇒ Click **Add** and then click **OK**.

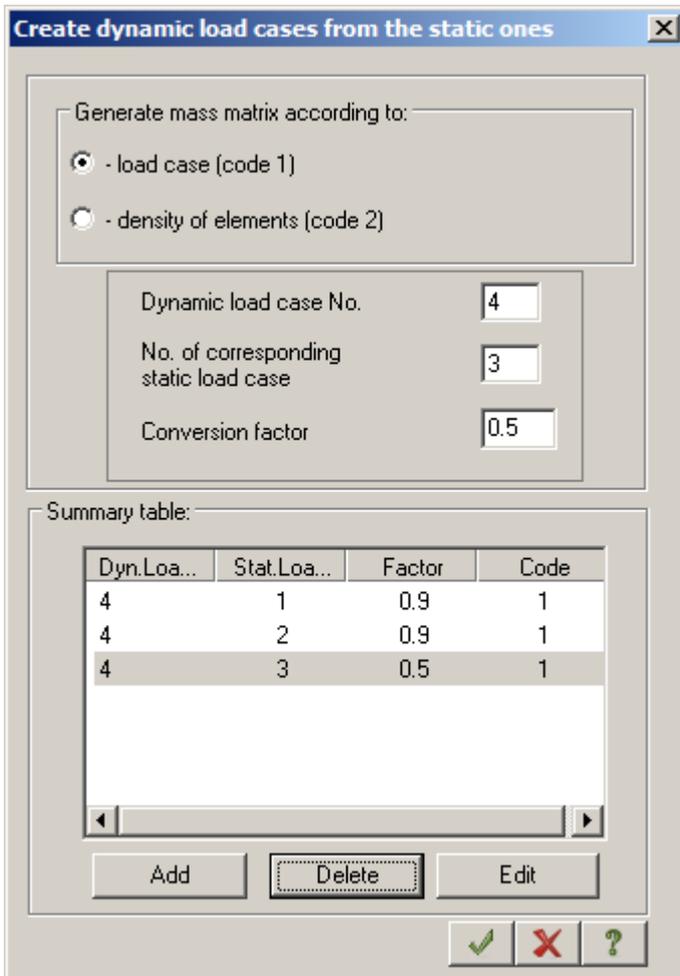


Figure 4.27 Create dynamic load cases from the static ones dialog box

Step 9. Generating table of dynamic load cases



Unfavourable direction of earthquake load is along the smaller side of the structure. As the structure has dimensions 9 x12 m in plan, the X direction will be the most unfavourable.

- ⇒ On the **Analysis** ribbon tab, select the **Dynamics** panel and click **Table of dynamic load cases** button .
- ⇒ In the **Table of dynamic load cases** dialog box (see Fig.4.28), define the following data:
 - load case No. – 4;
 - dynamic load type – **Earthquake /01.01.2000/SP 14.13330.2011/ (35)**;
 - number of analysed mode shapes – 10.
- ⇒ Click **Parameters**.

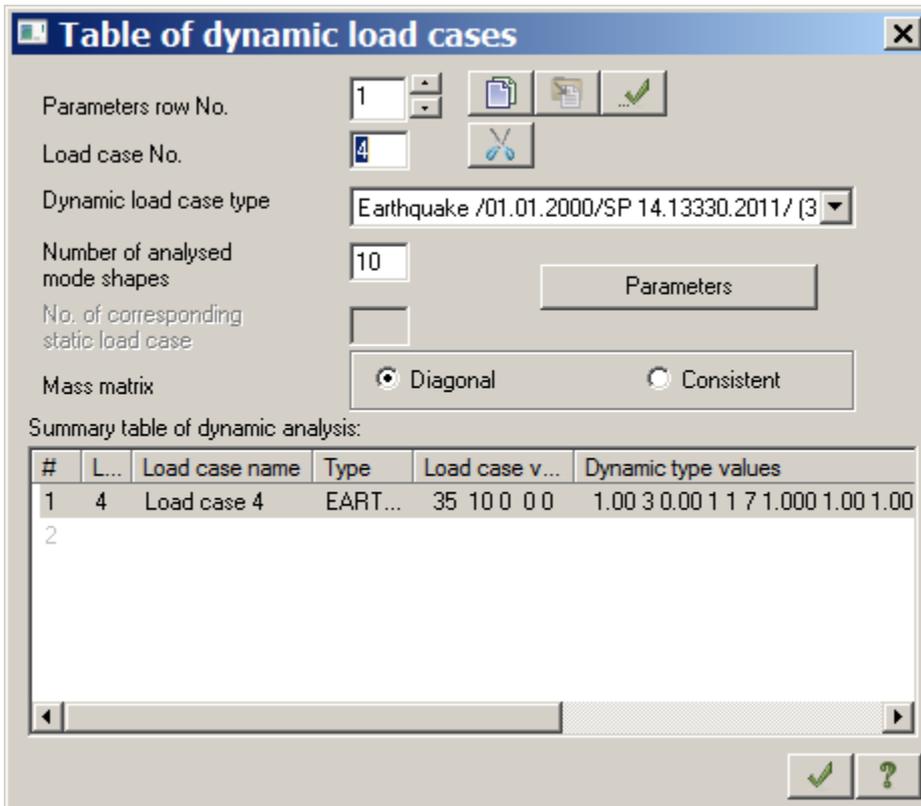


Figure 4.28 Table of dynamic load cases dialog box

- ⇒ In the **Earthquake analysis parameters** dialog box (see Fig.4.29), define the following data:
 - direction cosines of earthquake load resultant in global coordinate system – CX = 1;
 - other parameters remain by default.
- ⇒ Click **OK** .

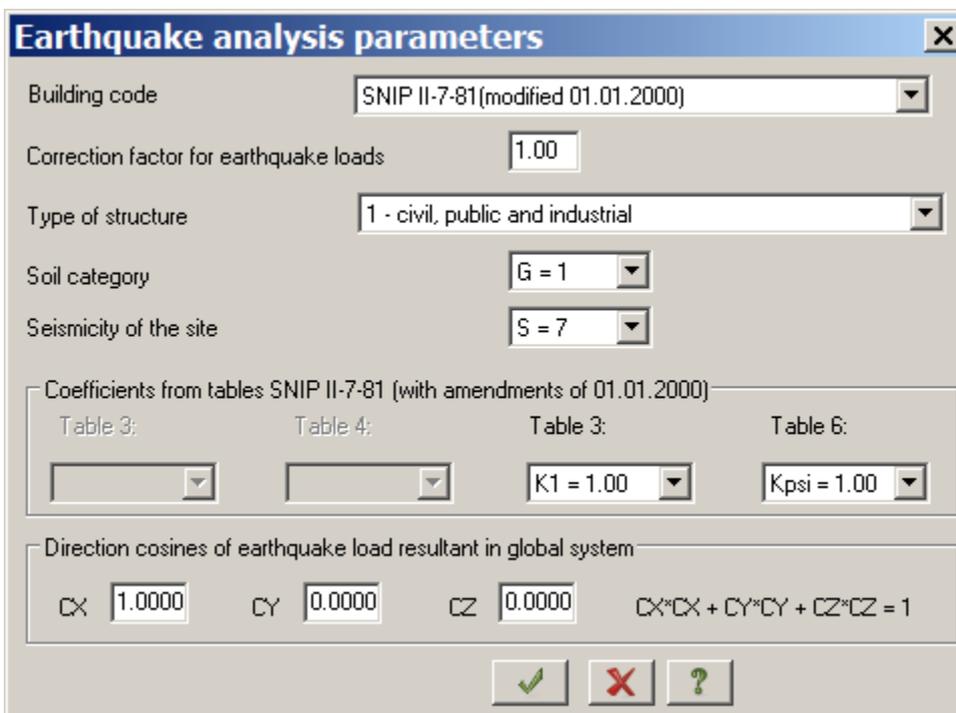


Figure 4.29 Earthquake analysis parameters dialog box

- ⇒ In the **Table of dynamic load cases** dialog box, click **OK**.

Step 10. Generating DCF table

⇒ On the **Analysis** ribbon tab, select the **DCF** panel and click **DCF table** button .

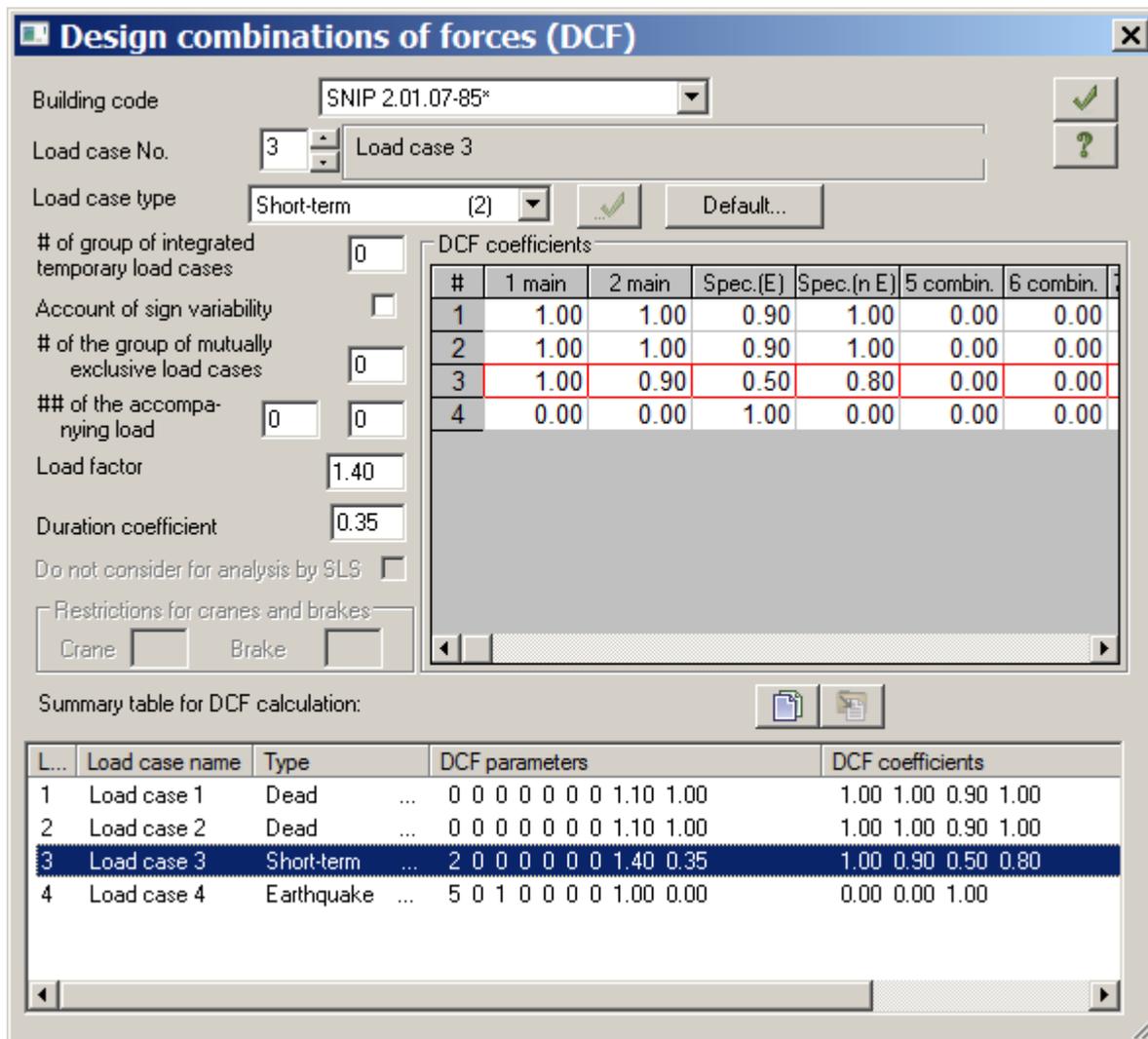


As the type of load cases was defined in the **Edit load cases** dialog box (see Fig.4.26), the DCF table is generated automatically with parameters accepted by default for every load case. Now you have to modify parameters for the third load case.

⇒ In the **Design combinations of forces** dialog box (see Fig.4.30), select building code **SNIP 2.01.07-85*** and specify the following data:

- for Load case 3 – in the **Load factor** box specify **1.4** and then click **Apply** .

⇒ Click **OK** .



Design combinations of forces (DCF)

Building code: SNIP 2.01.07-85*

Load case No.: 3 Load case 3

Load case type: Short-term (2) Default...

of group of integrated temporary load cases: 0

Account of sign variability:

of the group of mutually exclusive load cases: 0

of the accompanying load: 0 0

Load factor: 1.40

Duration coefficient: 0.35

Do not consider for analysis by SLS:

Restrictions for cranes and brakes: Crane Brake

DCF coefficients:

#	1 main	2 main	Spec.(E)	Spec.(n E)	5 combin.	6 combin.
1	1.00	1.00	0.90	1.00	0.00	0.00
2	1.00	1.00	0.90	1.00	0.00	0.00
3	1.00	0.90	0.50	0.80	0.00	0.00
4	0.00	0.00	1.00	0.00	0.00	0.00

Summary table for DCF calculation:

L...	Load case name	Type	DCF parameters	DCF coefficients
1	Load case 1	Dead	... 0 0 0 0 0 0 1.10 1.00	1.00 1.00 0.90 1.00
2	Load case 2	Dead	... 0 0 0 0 0 0 1.10 1.00	1.00 1.00 0.90 1.00
3	Load case 3	Short-term	... 2 0 0 0 0 0 1.40 0.35	1.00 0.90 0.50 0.80
4	Load case 4	Earthquake	... 5 0 1 0 0 0 1.00 0.00	0.00 0.00 1.00

Figure 4.30 Design combinations of forces dialog box

Step 11. Generating DCL table

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

⇒ In the **Design combinations of loads** dialog box (see Fig.4.31), select building code **SP 20.13330.2011** and specify the type of load for every load case by double-clicking the **Type** cell in the table for every load case:

- for Load case 1 – **Dead (P)** ;
- for Load case 2 – **Dead (P)** ;

- for Load case 3 – **Short-term leading 1 (Pt1)**;
 - for Load case 4 – **Earthquake (Pse)** .
- ⇒ For load case 4, double-click the **Sign variable** cell and define +/- .
- ⇒ For load case 3, in the **Load factor** cell, define factor as equal to **1.4**.

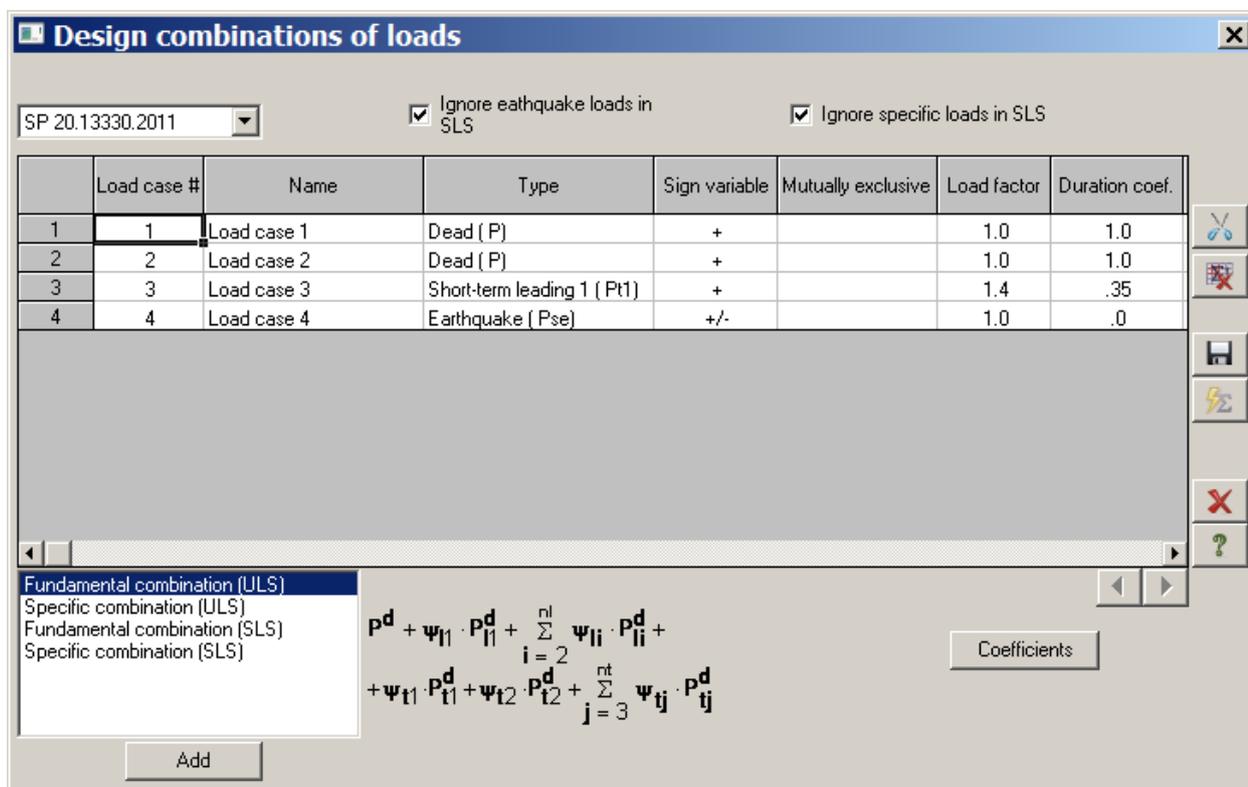


Figure 4.31 Design combination of loads dialog box

- ⇒ To add combinations, select every row (**Fundamental combination (ULS)**, **Specific combination (ULS)**, **Fundamental combination (SLS)**, **Specific combination (SLS)**) in turn and then click **Add** (columns with coefficient values according to SP 20.13330-2011 will appear in the table).
- ⇒ Click **Save data**.
- ⇒ To close the **Design combinations of loads** dialog box, click **Close**.

Step 12. Defining structural elements



Structural element (STE) is a set of several finite elements that during design procedure will be considered as a single unit. Elements that form the part of the structural element should have no gaps, have the same stiffness type, should not be included into other structural elements and unification groups, have common nodes and belong to the same line. In this version it is possible to select all elements of the model and unite them into structural ones.

To define structural element BEAM:

- ⇒ On the **Select** toolbar, click **Select horizontal bars** button  .
- ⇒ Select all horizontal elements of the model with the pointer.
- ⇒ To define structural elements, on the **More edit options** ribbon tab, select the **Design** panel and click **Structural elements** button  .
- ⇒ In the **Structural elements** dialog box (see Fig.4.32), under **Design option**, select **For all** check box. Then under **Edit structural elements**, click **Create**.

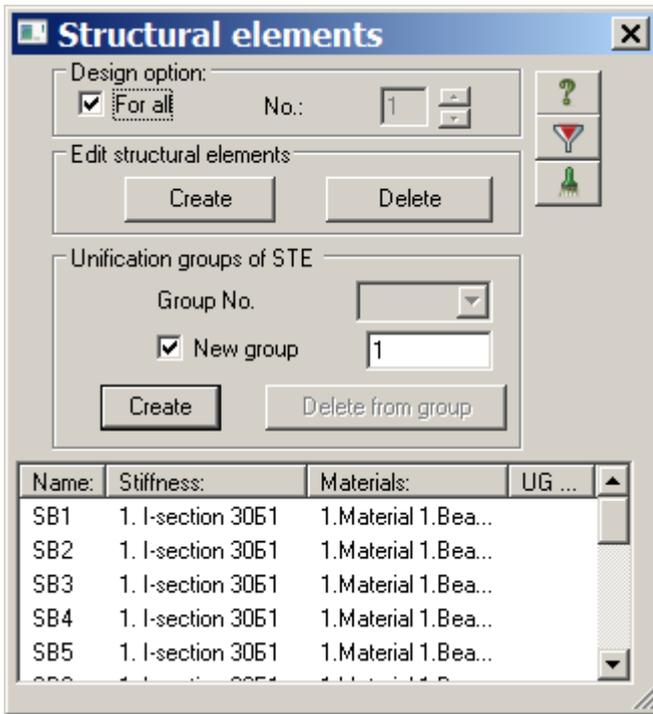


Figure 4.32 Structural elements dialog box

- ⇒ On the **Select** toolbar, click **Select horizontal elements** button  once again in order to make this command not active.

To define structural element COLUMN:

- ⇒ To switch to projection on the XOZ-plane, on the **Projection** toolbar (by default, it is displayed at the bottom of the screen), click **Projection on XOZ-plane** .
- ⇒ On the **Select** toolbar, click **Select vertical bars** button .
- ⇒ Select with the pointer columns of the first storey at the place where the diaphragm is located.
- ⇒ In the **Structural elements** dialog box, under **Design option**, select **For all** check box. Then under **Edit structural elements**, click **Create**.
- ⇒ Select with the pointer columns of the second storey at the place where the diaphragm is located.
- ⇒ In the **Structural elements** dialog box, under **Design option**, select **For all** check box. Then under **Edit structural elements**, click **Create**.
- ⇒ To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection** .
- ⇒ On the **Select** toolbar, click **Select vertical elements** button  once again in order to make this command not active.

Step 13. Defining deflection fixities at nodes of bending elements

- ⇒ On the **Select** toolbar, click **Select horizontal bars** button .
- ⇒ Select all horizontal elements of the model with the pointer.
- ⇒ On the **More edit options** ribbon tab, on the **Design** panel, click **Deflection fixities** command .
- ⇒ In the **Deflection fixities** dialog box (see Fig.4.33), select **Create at nodes with NotSLA bars** option in the list. Then select Y1 and Z1 check boxes.
- ⇒ Click **Apply** (deflection of element sections is determined relative to the line that connects fixities at the ends of the element).

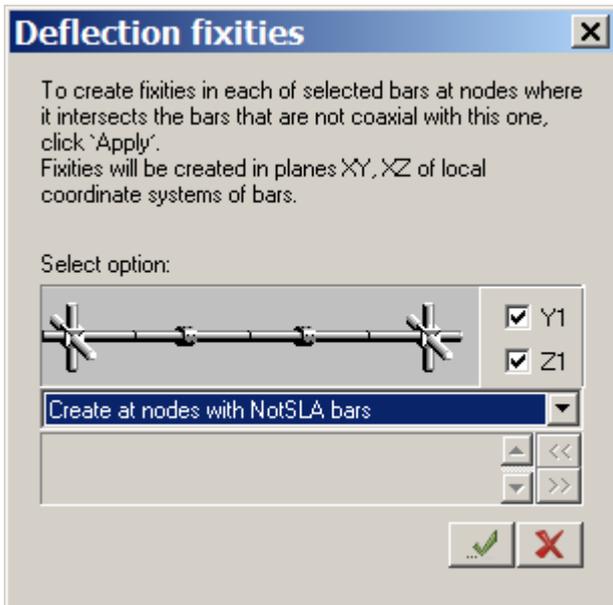


Figure 4.33 **Deflection fixities** dialog box

⇒ To close the **Deflection fixities** dialog box, click **Close**.

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

Step 14. Complete analysis of frame

⇒ To carry out complete analysis of frame, on the **Analysis** ribbon tab, select the **Analysis** panel and in the **Analyse problem** drop-down list, click **Complete analysis** .

Step 15. Review and evaluation of static & dynamic analyses results



*When analysis procedure is complete, to review and evaluate results of static and dynamic analyses, 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.4.34).

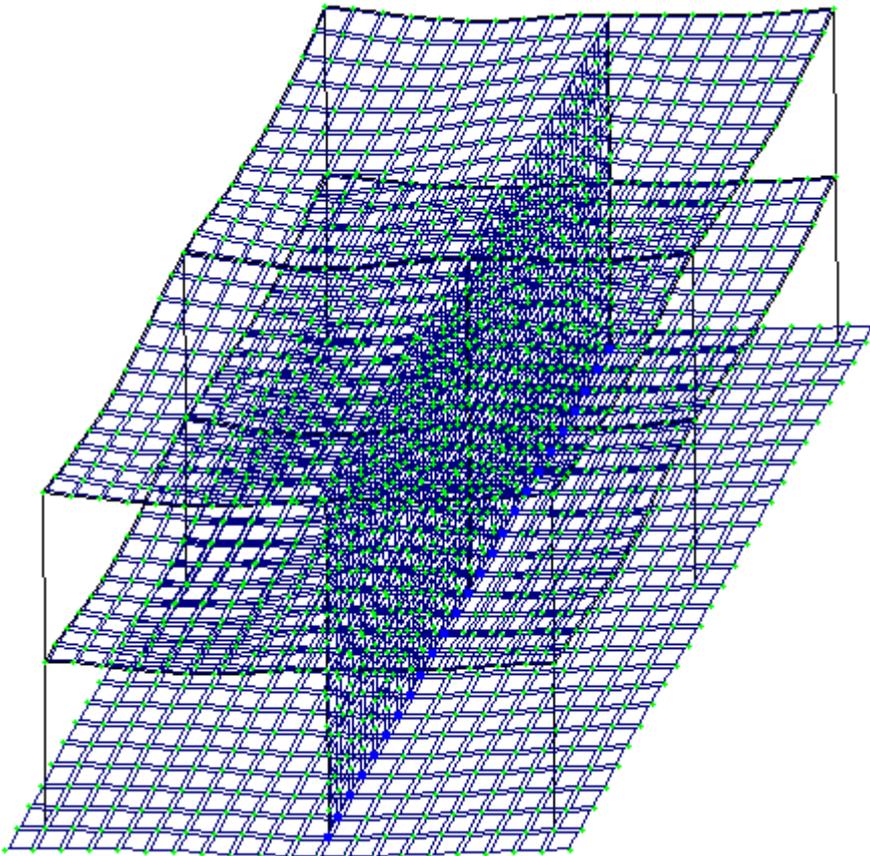


Figure 4.34 Design model with account of nodal displacements

To present diagrams of internal forces:

- ⇒ To display bars, on the **Select** toolbar, click **PolyFilter** .
- ⇒ In the **PolyFilter** dialog box, select the **Filter for elements** tab (the second tab).
- ⇒ Click **By FE shape** option and select **2-node FE (bars)** in the list.
- ⇒ Click **Apply** .
- ⇒ To present on the screen only selected bars, on the **Select** toolbar, click **Fragmentation** .
- ⇒ 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 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 restore design model in initial view, on the **Select** toolbar, click **Restore model** .

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 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** .

To present mode shapes of the structure:

- ⇒ To change the number of active load case, on the Status bar (displayed at the bottom of the screen), in the **Load case No.** list, select No. 4 and click **Apply** .
- ⇒ To display the first mode shape, on the **Results** ribbon tab, on the **Deformations** panel, select the **Mode shapes** command  in the **Stress strain state** drop-down list.
- ⇒ To display the 2nd mode shape of the 4th load case, on the Status bar (displayed at the bottom of the screen), in the **Mode shape No. (component, period)** list, change number of mode shape for 2 and click **Apply** .

To animate the 2nd mode shape:

- ⇒ To switch to the mode of 3D visualization, either select **3D model** command on the **Application menu** or click the **3D model** button  on the Quick Access Toolbar.
- ⇒ To animate the 2nd mode shape of the 4th load case, on the **3D view** ribbon tab, on the **Animation** panel, click **Animate mode shape** button .
- ⇒ In the **Animate mode shape** dialog box (see Fig.4.35), click **Play** button .
- ⇒ To close the **Animate mode shape** dialog box, click **Close**.

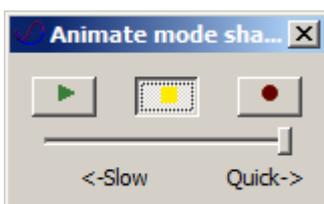


Figure 4.35 Animate mode shape dialog box

- ⇒ To return to the mode of analysis results visualization, on the **3D view** ribbon tab, on the **Back** panel, click **Finite element model** button .

To generate and review tables of analysis results:

- ⇒ To present table with forces 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.4.36), select **Forces** in the list.
- ⇒ Click **Apply** (to generate tables in HTML format, select appropriate option). To generate table and work further in **Document Maker** (**DOC-SAPR** module), select RPT format.

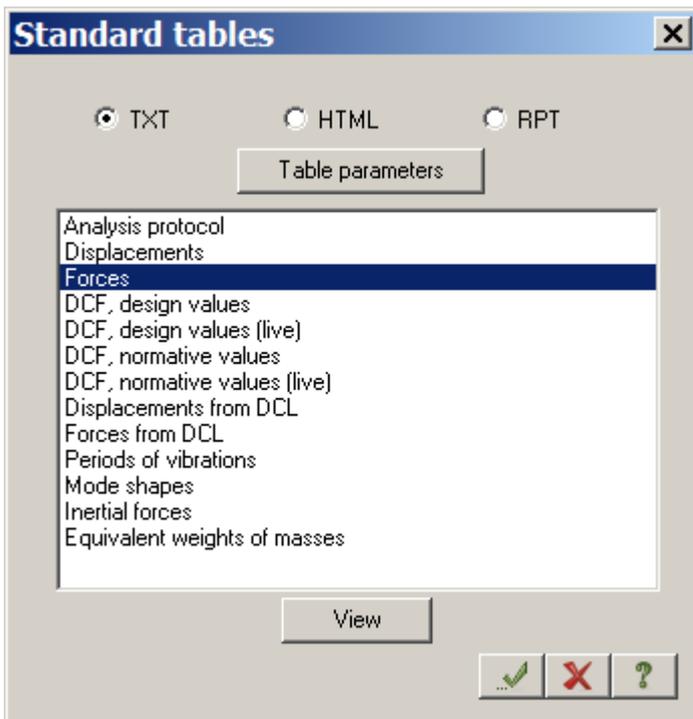


Figure 4.36 **Standard tables** dialog box

- ⇒ In the **Choose load case No.** dialog box (see Fig.4.37), select **All load cases** option and click **OK** .

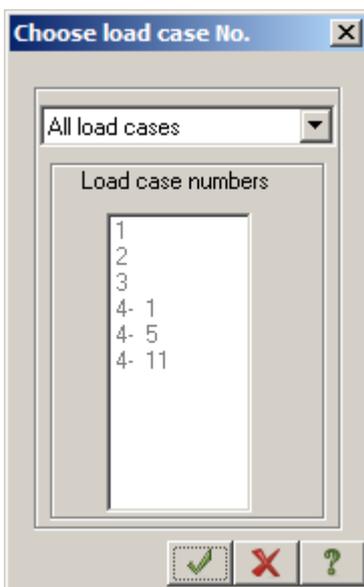


Figure 4.37 **Choose load case No.** dialog box

- ⇒ To close the table, on the FILE menu, click **Close**.
- ⇒ To present table with periods of vibrations, in the **Standard tables** dialog box, select appropriate row in the list.
- ⇒ Click **Apply** .
- ⇒ To save the table in *.txt format, on the Quick Access Toolbar, click **Save**.
- ⇒ In the **Save as** dialog box specify the following data:
 - file name (name of the table) – **Periods4**;
 - location where you want to save this file (**Work** folder is displayed by default).
- ⇒ Click **Save**.

Step 16. Review and evaluate results from analysis of reinforcement and steel analysis



*When analysis procedure is complete, to review and evaluate results from analysis of reinforcement and steel analysis, select the **Design** ribbon tab (for standard ribbon interface).*

To present results from analysis of reinforcement:

- ⇒ To present information about determined reinforcement in a certain plate element, on the **Select** toolbar, click **Information about nodes and elements** button  and specify with a pointer any plate element on the model.
- ⇒ In the dialog box that appears on the screen, select the **Information about reinforcement** tab. This dialog box contains complete information about selected element, including results for reinforcement.
- ⇒ To close the dialog box, click **Close** button.
- ⇒ To display mosaic plot for area of lower reinforcement in plates along the X1-axis, click the **Lower reinforcement in plates along X1** button  (on the **Design** ribbon tab, the **RC: Plates** panel).
- ⇒ To display mosaic plot for area of lower reinforcement in plates along the Y1-axis, click the **Lower reinforcement in plates along Y1** button  (on the **Design** ribbon tab, the **RC: Plates** panel).

To generate and review table with analysis results for reinforcement:

- ⇒ On the **Design** ribbon tab, select the **Tables** panel and click **Analysis results tables for RC** command  in the **Documents** drop-down list.
- ⇒ In the **Tables of analysis results** dialog box (see Fig.4.38), click the **Plates** option in the **Elements** area. The following data is mentioned by default: option **For all elements** in the **Create table** area and **Text** option in the **Table format** area.
- ⇒ Click **Table - on the screen** (It is also possible to generate tables with analysis results in other formats in the same way, just select appropriate format option).

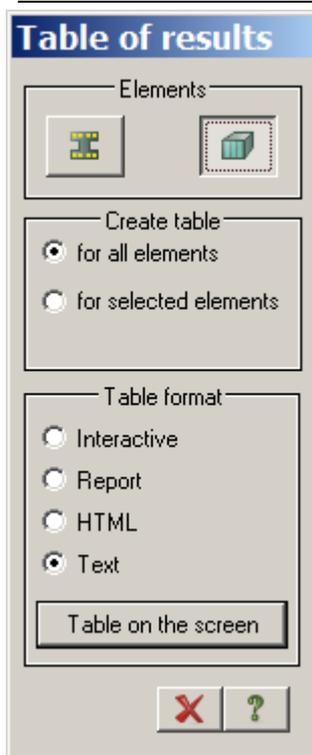


Figure 4.38 **Tables of analysis results** dialog box

To present mosaic plots for the check of assigned sections of steel bars:

- ⇒ With the **PolyFilter** command, select all bars of the model.
- ⇒ Perform fragmentation for selected elements.
- ⇒ To present mosaic results (for assigned cross-sections, check for ultimate limit state), on the **Design** ribbon tab, on the **Steel: check and select** panel, click **Check, ULS** button .
- ⇒ To present mosaic results (for assigned cross-sections, check for local buckling), on the **Design** ribbon tab, on the **Steel: check and select** panel, click **Check, LB** button .

To generate the table with results (assigned sections):

- ⇒ To generate tables with analysis results, on the **Results** ribbon tab, select **Tables** panel and click **Analysis results tables for steel**  in the **Documents** drop-down list.
- ⇒ In the **Table of results** dialog box (see Fig.4.39), select the **Check** option.
- ⇒ Click **Apply**  (to generate tables in HTML format, select appropriate option). To generate table and work further in **Document Maker (DOC-SAPR module)**, select RPT format. It is also possible to present tables in Excel format.

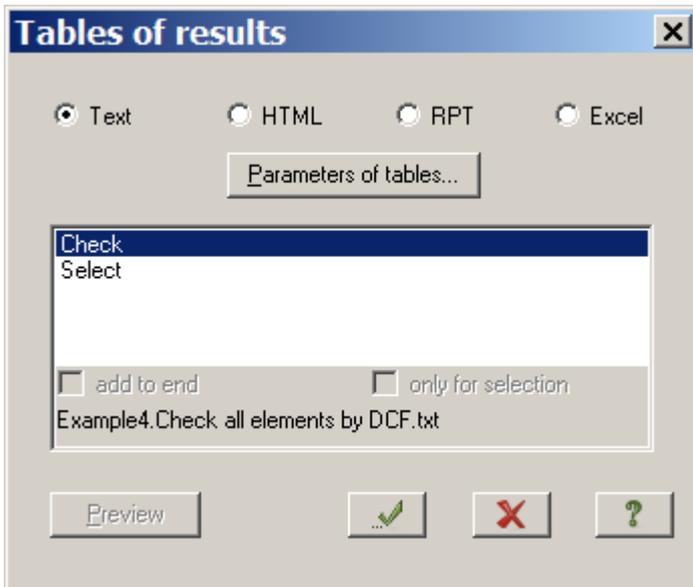


Figure 4.39 **Tables of results (steel analysis)** dialog box

⇒ To close the table, on the FILE menu, click **Close**.

To generate the table with results (selected sections):

- ⇒ In the **Table of results** dialog box (see Fig.4.39), define the **Select** option.
- ⇒ Click **Apply** .

To change the No. of design option:

- ⇒ On the Status bar, in the **Design option No.** box, select the row corresponding to the second design option.



To review and evaluate analysis results for another design option, do one of the following:

- either use the **Design options**  command. Then in the **Design options** dialog box, select appropriate row in the **List of design options** and click **Set as current type**;
- or on the Status bar, in the **Design option No.** box, select the row corresponding to appropriate design option. The design option with this number will become current one.