# Example 11_M. Analysis of structure with option to modify stiffness of soil (METEOR system)

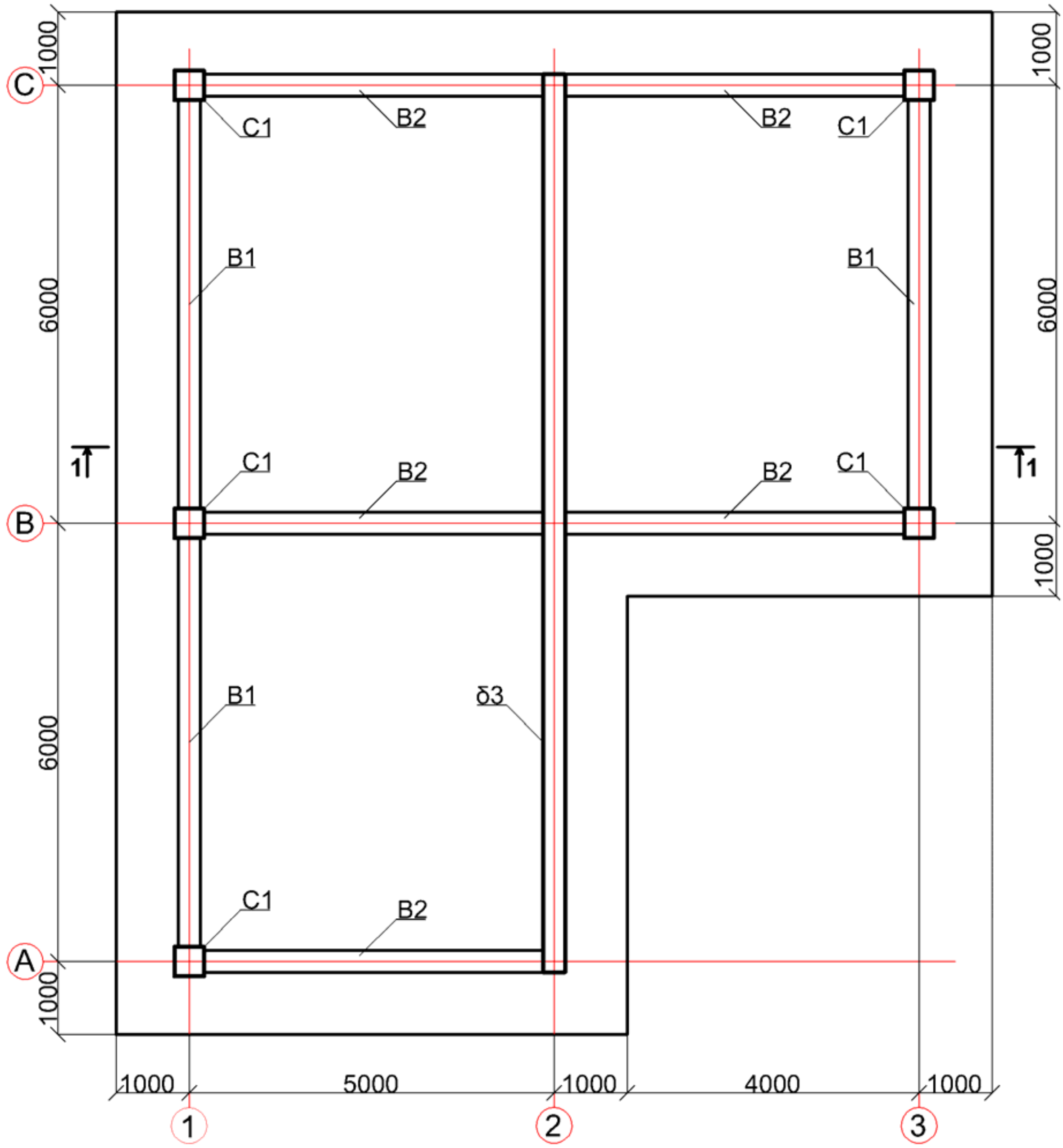
In this lesson you will learn how to:
– generate different variants of design model;
– use structural blocks (StB);
– generate integrated DCF according to results of multivariant analysis;
– analyse reinforcement according to integrated DCF of multivariant analysis.

**Description:**
Model of the framework and sections of elements in the framework are presented in Fig.11.1.
Material for framework – reinforced concrete B25.
3D framework with base slab (foundation slab) on elastic foundation with the following modulus of subgrade reaction (subgrade modulus):
– variant 1   C1 = 1000 $t/m^3$;
– variant 2   C1 = 5000 $t/m^3$;
– variant 3   C1 = 1000 $t/m^3$ at the second span, C1 = 0 $t/m^3$ at the first span;
– variant 4   C1 = 1000 $t/m^3$ at the first span, C1 = 0 $t/m^3$ at the second span.

Loads (variants 1, 2, 3 and 4):
– load case 1 – dead weight;
– load case 2 – dead uniformly distributed load $g_1$ = 1.5 $t/m^2$ applied to floor slabs of the 1st and the 2nd floor; dead uniformly distributed load $g_2$ = 2 $t/m^2$ applied to soil;
– load case 3 – snow load $g_3$ = 0.08 $t/m^2$.

Loads (variant 2):
– load case 4 – earthquake load. Seismicity of the area – 7 units of magnitude, soil category – 1. Unfavourable direction of earthquake load – along the smaller side of the structure.


1000
6000
6000
1000
1000
6000
1000
C
B
A
C1
B2
B2
C1
B1
B1
C1
B2
B2
C1
B1
δ3
C1
B2
1
1
1000
5000
1000
4000
1000
1
2
3

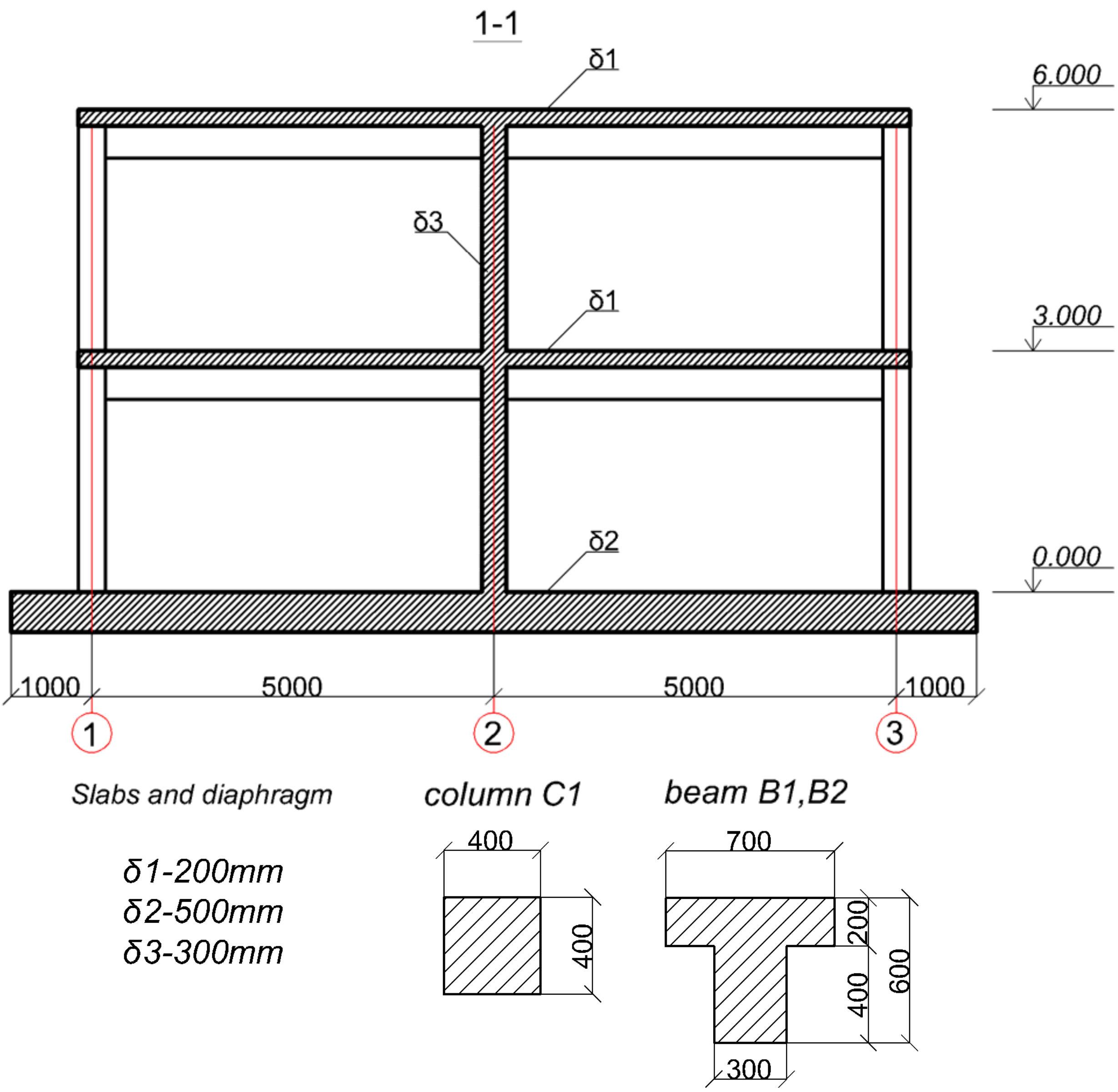



Figure 11.1 Model of the framework and sections of element

➪ 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.11.2), specify the following data:

- problem name – **Example11_M**;
- model type – **5 – Six degrees of freedom per node**.

➪ Click **OK** .

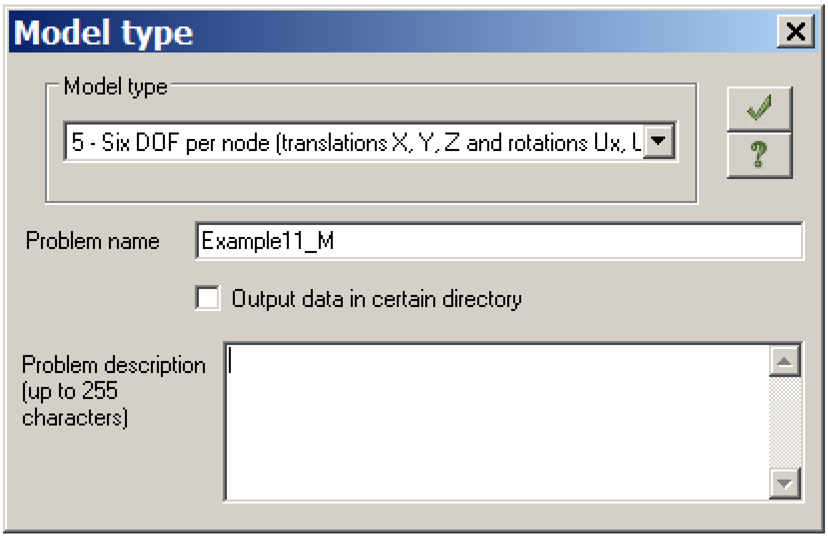



Figure 11.2 **Model type** dialog box

*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.*

*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

<u>To generate 3D frame:</u>

➪ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** list and click the **3D frame** command.

➪ In the **Create plane fragments and grids** dialog box (see Fig.11.3), specify the following data:

- spacing along X: spacing along Y: spacing along Z:

| spacing along X: **L(m)** | **N** | **M** | spacing along Y: **L(m)** | **N** | **M** | spacing along Z: **L(m)** | **N** | **M** |
|---|---|---|---|---|---|---|---|---|
| 5 | 2 | 10 | 6 | 2 | 12 | 3 | 2 | 1 |

➪ Clear the **Apply restraints** check box.

➪ In the **Parameters of foundation slab** area, select the **Generate side overhangs** check box and define the following data in the **Side overhangs** area:

- width of overhand along X – 1m;
- number of FE along X – 2;
- width of overhand along Y – 1m;
- number of FE along Y – 2;
- other parameters remain by default.

➪ Click **Apply** .

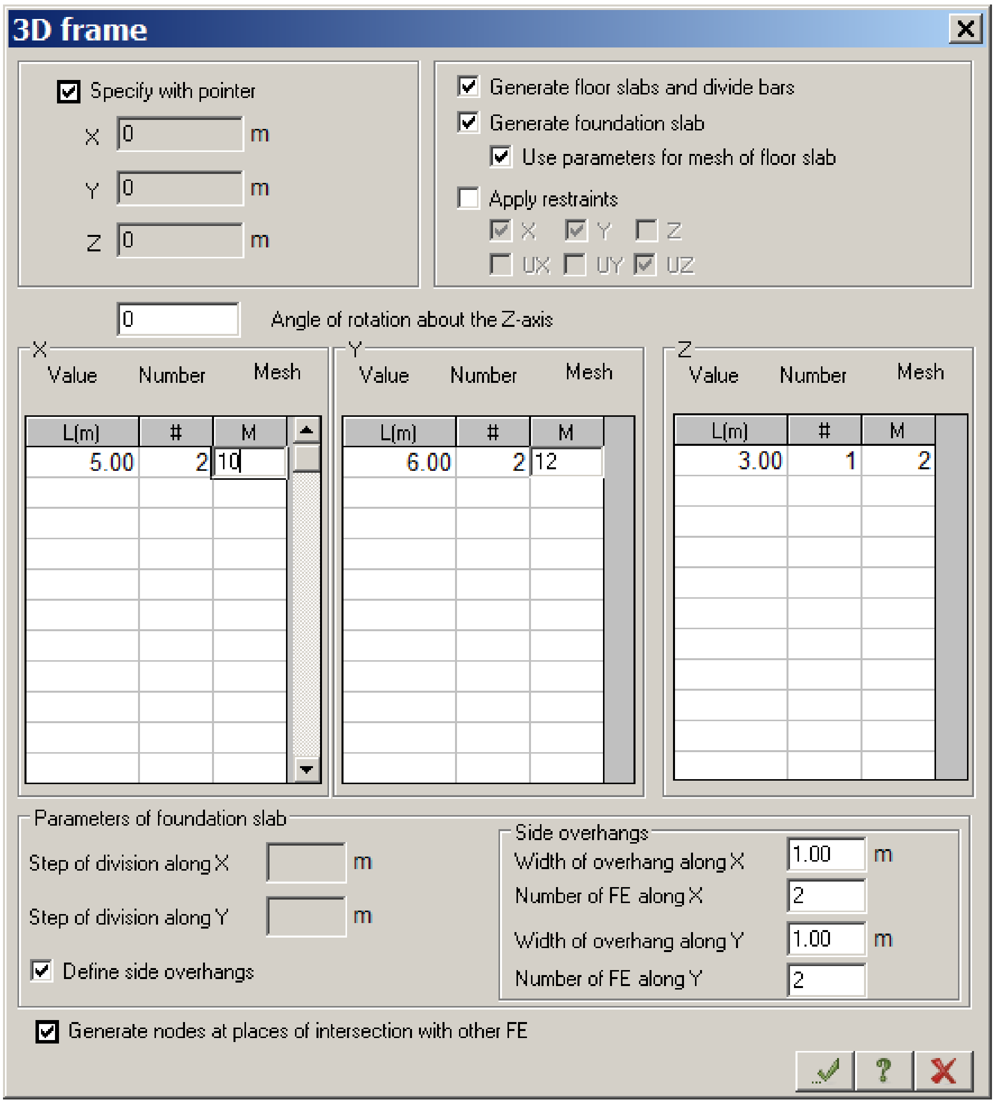


Figure 11.3 **3D frame** dialog box

<u>To present numbers of nodes on the screen:</u>

➪ On the **Select** toolbar, click **Flags of drawing** button .

➪ In the **Display** dialog box (see Fig.11.4), select the **Node numbers** check box on the **Nodes** tab (the second tab).

➪ Click **Redraw** .

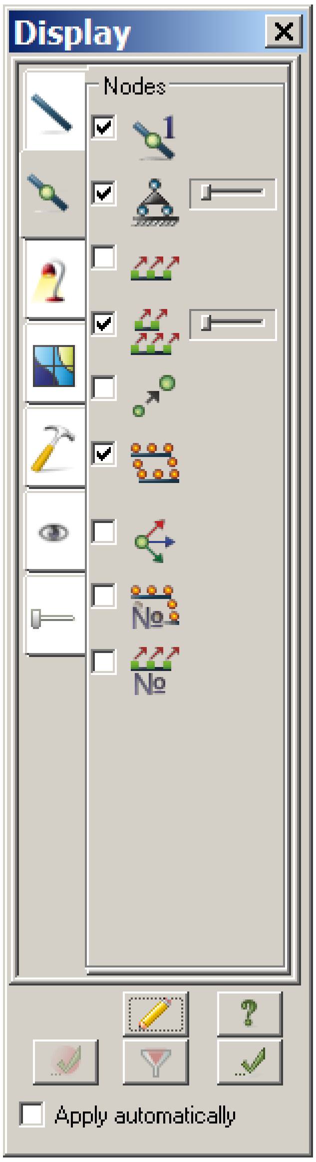


Figure 11.4 **Display** dialog box

<u>To generate diaphragm:</u>

- On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** list and click the **Create wall-beam**  command.
- In the **Create plane fragments and grids** dialog box (see Fig.11.5), specify **angle of rotation about the Z-axis** as equal to 90 degrees.
- Specify with the pointer node No.11 (the node will be coloured pink and coordinates of node become available in the dialog box).
- In the table of the dialog box specify the following data:
  - spacing along the first axis: spacing along the second axis:

| L(i) | N | L(i) | N |
|---|---|---|---|
| 0.5 | 24 | 0.5 | 12 |

- Click **Apply** .

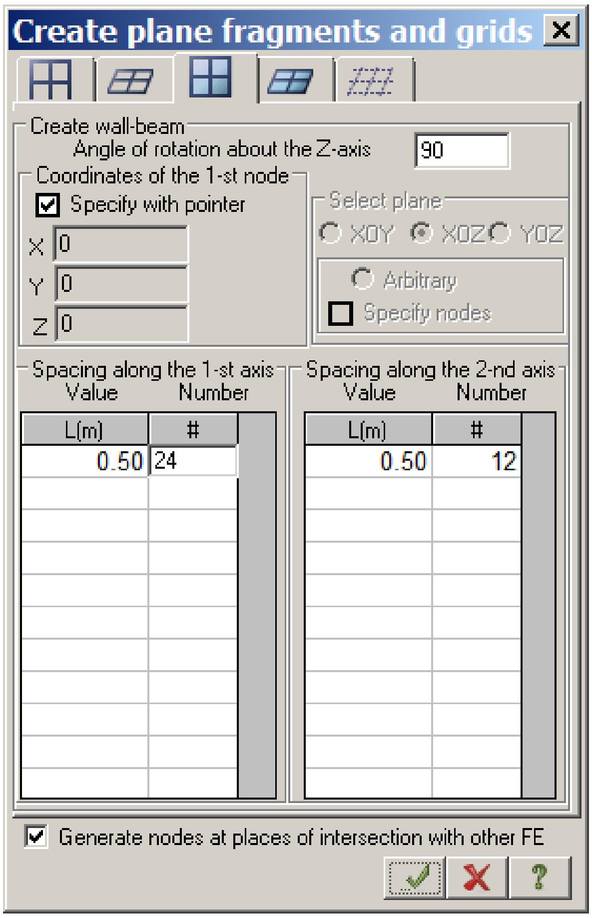



Figure 11.5 **Create plane fragments and grids** dialog box

To hide numbers of nodes on design model:

- On the **Select** toolbar, click **Flags of drawing** button .
- In the **Display** dialog box, clear the **Node numbers** check box on the **Nodes** tab.
- Click **Redraw** .

To edit the model:

- 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** and **Select horizontal elements** . Then select beams and columns at the place where diaphragm is located (84 elements should be selected).

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

- To delete selected elements, on the **Create and edit** ribbon tab, on the **Edit** panel, click **Delete selected objects** .
- To switch to projection on the XOY-plane, on the **Projection** toolbar, click **Projection on XOY-plane** .
- On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button . With selection window select nodes of slabs in the first right span of the model, as shown in Fig.11.6.

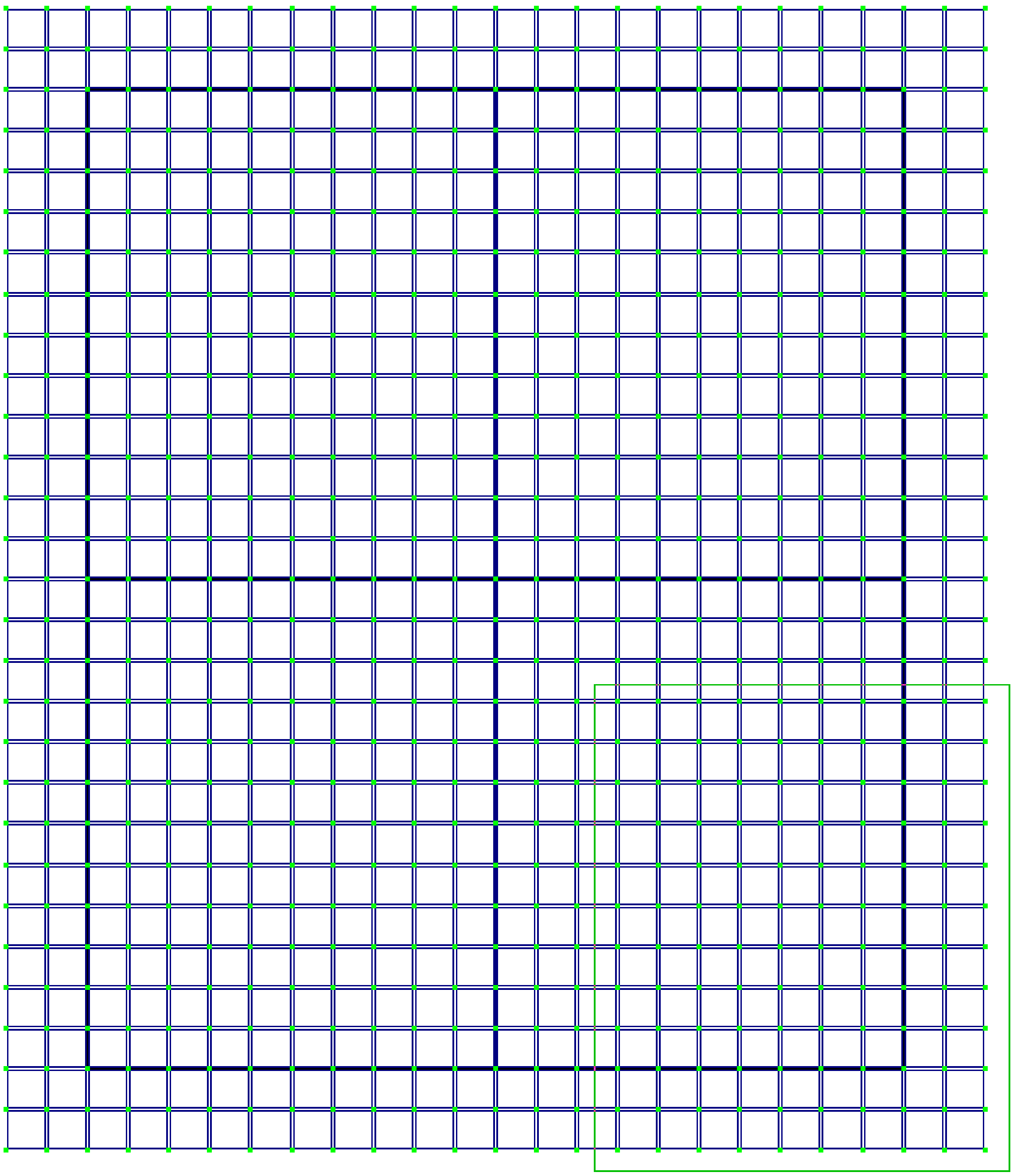


Figure 11.6 Select nodes of slabs

- To delete selected nodes, on the **Create and edit** ribbon tab, on the **Edit** panel, click **Delete selected objects**  . ***N.B.*** When you delete nodes, elements adjacent to these nodes will be automatically deleted.
- On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button  in order to make this command not active.
- On the **Select** toolbar, click **Select vertical elements** button  in order to make this command not active.
- On the **Select** toolbar, click **Select horizontal elements** button  in order to make this command not active.
- To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection**  .

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.11.7), 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.

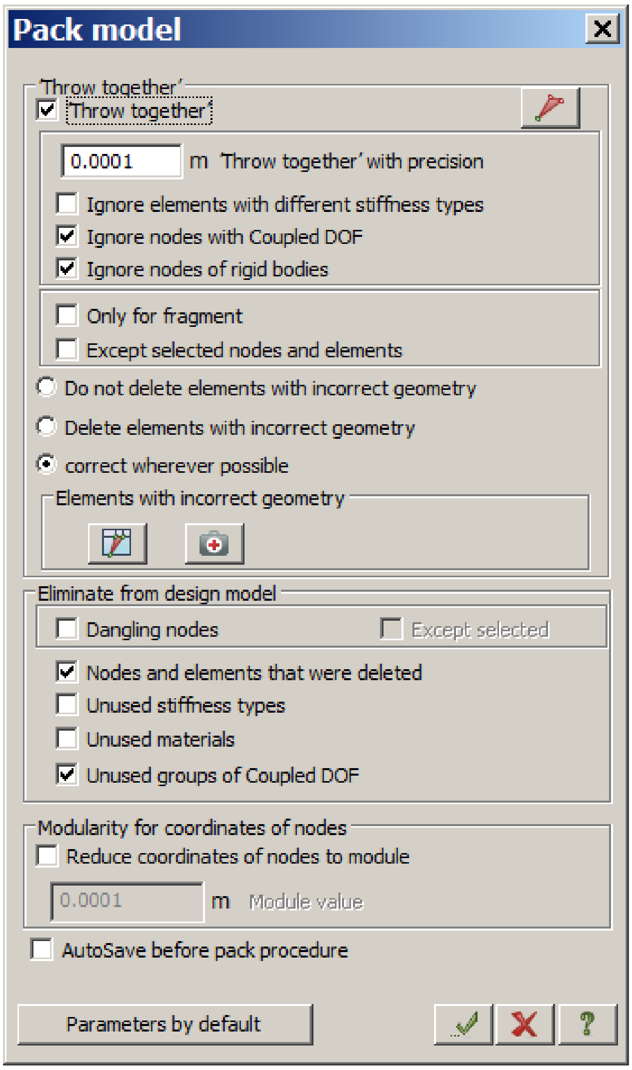


Figure 11.7 **Pack model** dialog box

The model shown in Fig.11.8 will be displayed on the screen.

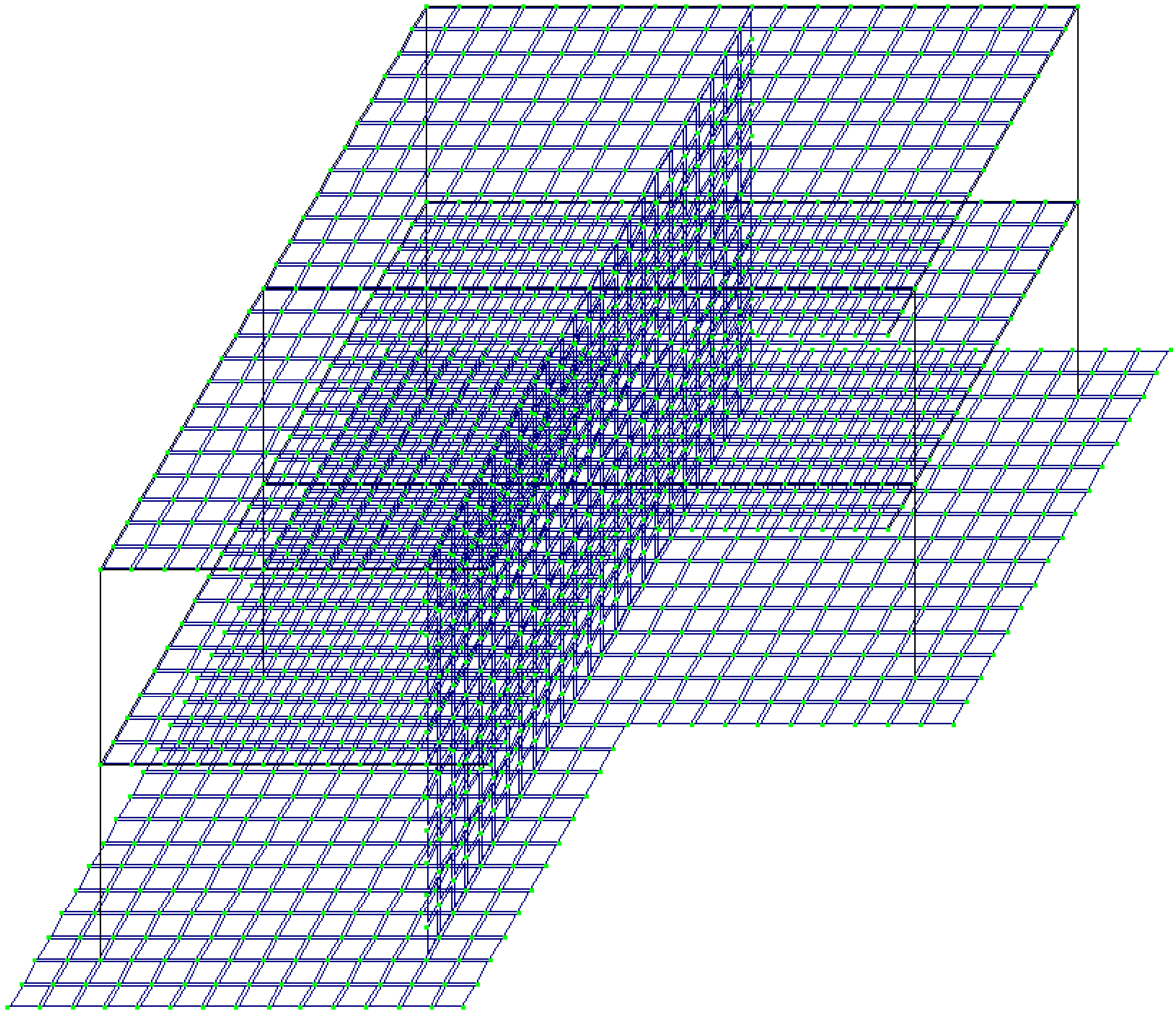


Figure 11.8 Design model of the framework

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

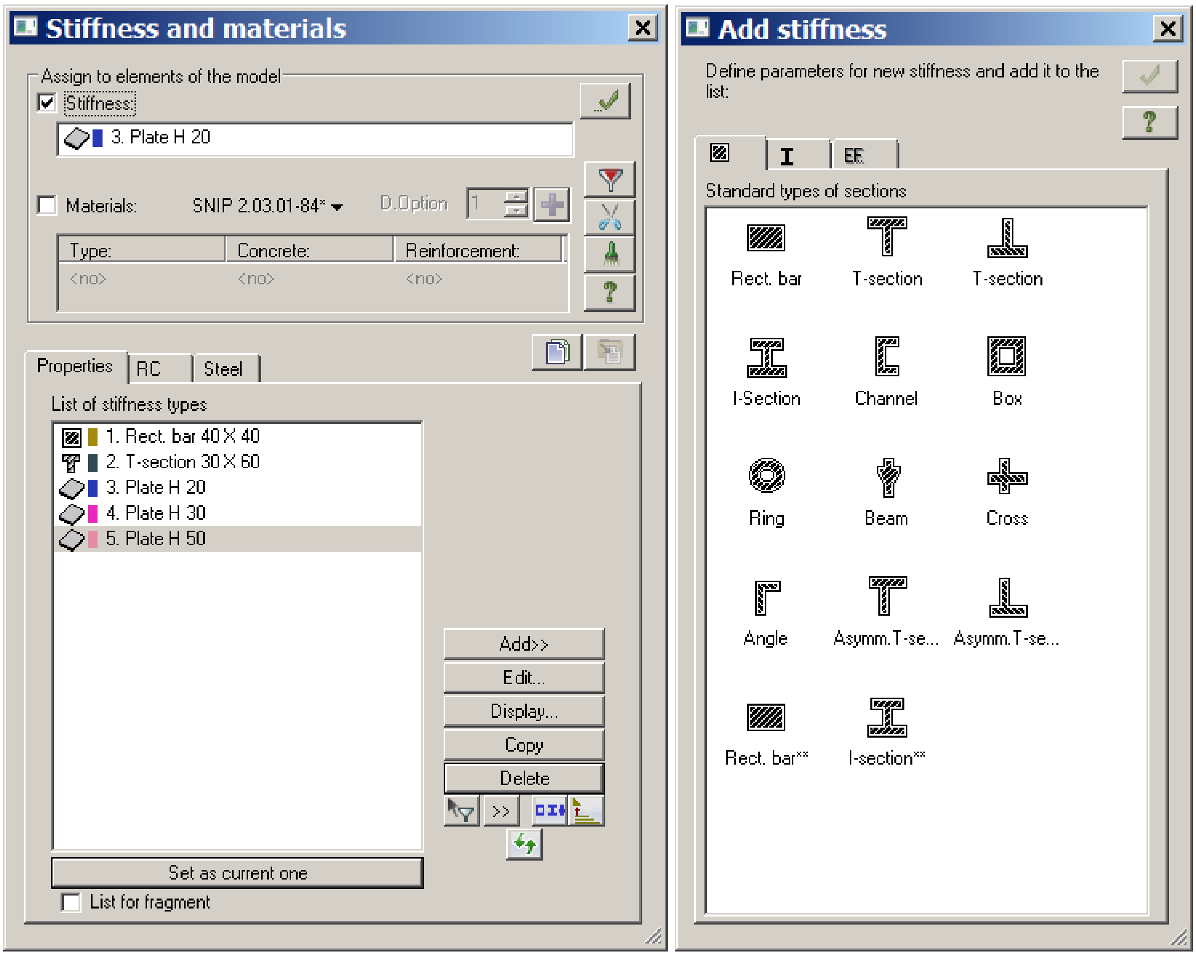
- On the **LIRA-SAPR menu** (Application menu), click **Save** command .
- In the **Save as** dialog box specify the following data:
  - file name – **Example11_M**;
  - location where you want to save this file (**Data** folder is displayed by default).
- Click **Save**.

### Step 3. Defining material properties to elements of the model

<u>To create material data sets:</u>

- 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.11.9a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.11.9b).




a b

Figure 11.9 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.11.10):
  - modulus of elasticity – E = 3e6 t/m$^2$ (for the U.S. keyboard layout);
  - geometric properties – B = 40 cm; H = 40 cm;
  - unit weight of material – Ro = 2.75 t/m$^3$.
- To preview schematic presentation, click **Draw**.
- To confirm the specified data, click **OK** .

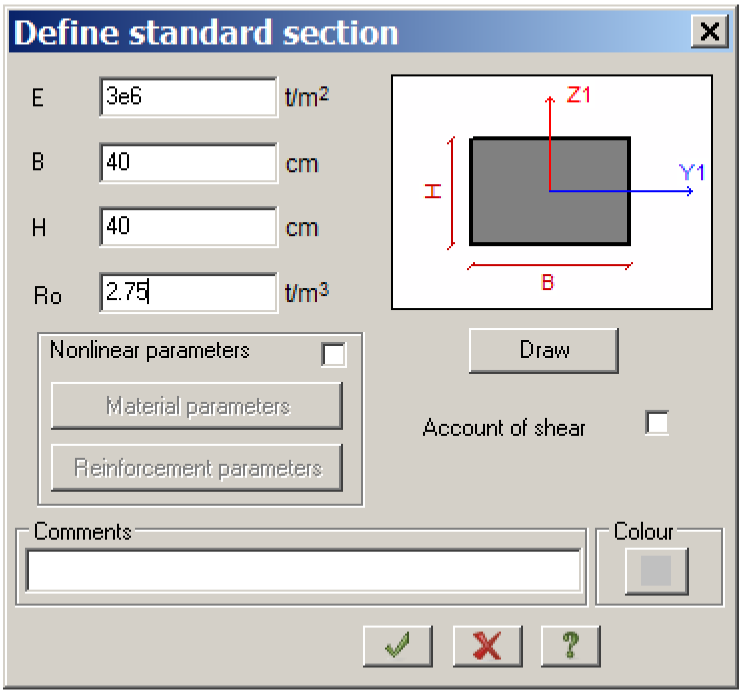



Figure 11.10 **Define standard section** dialog box

- Then in the **Add stiffness** dialog box, double-click the **T-section (table at the top)** icon in the list.

- In another **Define standard section** dialog box, specify the following parameters for **T-section (table at the top)**:
  - modulus of elasticity – E = 3e6 t/m$^2$;
  - geometric properties – B = 30 cm; H = 60 cm; B1 = 70 cm; H1 = 20 cm;
  - unit weight of material – Ro = 2.75 t/m$^3$.
- To confirm the specified data, click **OK**  .

- 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.11.11), specify the following parameters for **Plate** (floor slab):
  - modulus of elasticity – E = 3e6 t/m$^2$ (for the U.S. keyboard layout);
  - Poisson's ratio – v = 0.2;
  - thickness – H = 20 cm;
  - unit weight of material – $R_o$ = 2.75 t/m$^3$.
- To confirm the specified data, click **OK**  .

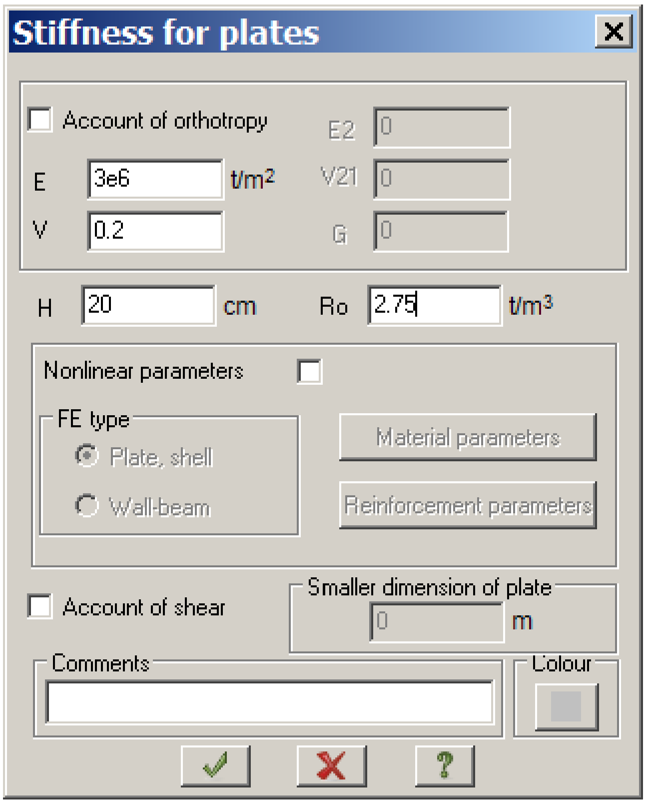



Figure 11.11 **Stiffness for plates** dialog box

- 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 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 cm.
- Click **OK**.

- To hide the library of stiffness parameters, click **Add** in the **Stiffness and materials** dialog box.

<u>To assign stiffness to elements of the model:</u>

- To select elements of the model, on the **Select** toolbar, click **Structural blocks (StB)** button .
- In the **Structural blocks** dialog box (see Fig.11.12), select the first row **Block (1)** with the comment **3D frame**. In this case, floor slabs and bars will be selected on the model.

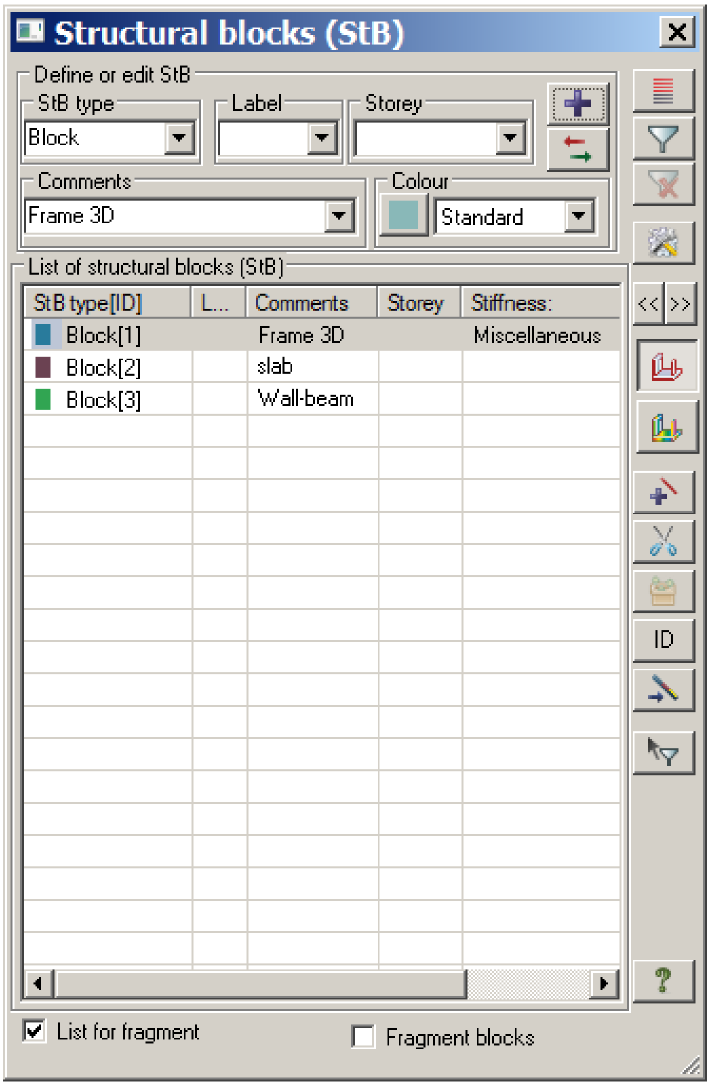



Figure 11.12 **Structural blocks (StB)** dialog box

- In the **Stiffness and materials** dialog box, when the current stiffness is defined as ‘3.Plate H 20’, click **Apply** .
- In the **Warning** box, click **OK**.

- In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select stiffness type '2.T-section 30x60'.
- 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.
- In the **Stiffness and materials** dialog box, click **Apply** .

- In the **Stiffness and materials** dialog box, in the list of stiffness types, select ‘1.Rect.bar 40x40’.
- Click **Set as current type**.
- On the **Select** toolbar, click **Select vertical elements** button .
- Select all vertical elements with the pointer.
- In the **Stiffness and materials** dialog box, click **Apply** .
- In the **Warning** box, click **OK**.

- In the **Stiffness and materials** dialog box, in the list of stiffness types, select ‘4.Plate H 30’.

- Click **Set as current type**.
- In the **Structural blocks** dialog box (see Fig.11.12), select the second row **Block (3)** with the comment **wall-beam**.
- In the **Stiffness and materials** dialog box, click **Apply** .

- In the **Stiffness and materials** dialog box, in the list of stiffness types, select ‘5.Plate H 50’.
- Click **Set as current type**.
- In the **Structural blocks** dialog box (see Fig.11.12), select the second row **Block (2)** with the comment **slab**.
- In the **Stiffness and materials** dialog box, click **Apply** .

### Step 4. Defining parameters of elastic foundation

- Select elements of the base slab once again (use the **Structural blocks** dialog box).
- 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.11.13), 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 t/$m^3$.
- Click **Apply** .

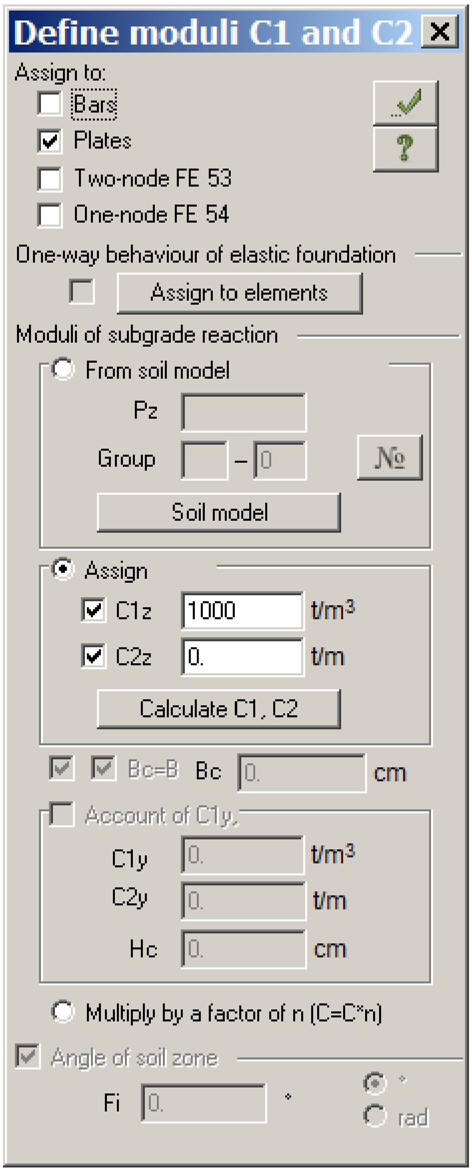
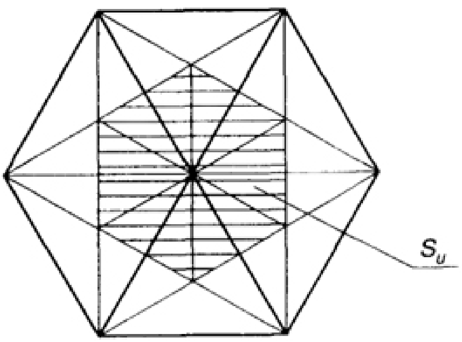



Figure 11.13 **Define moduli C1 and C2** dialog box

## Step 5. Defining boundary conditions

*To avoid geometric unstability 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.*
***N.B.*** *'Load area' is herein taken to mean the area of loaded soil, area that is associated with one-node FE. See the hatched area in the Figure below.*



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.11.14), specify the following parameters for the section:
  - stiffness of FE per unit length in tension-compression along the global X-axis – Rx = 175 t/m;
  - stiffness of FE per unit length in tension-compression along the global Y-axis – Ry = 175 t/m.
- To confirm the specified data, click **OK** .

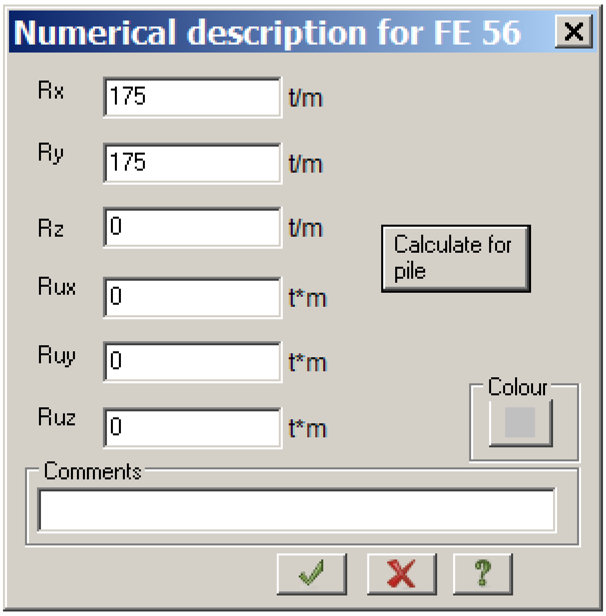



Figure 11.14 **Numerical description for FE 56** dialog box

To add FE 56:

- Make sure that nodes of the base slab are selected on the model.
- 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.11.15).
- In this dialog box, specify with the pointer FE '56' option.
- Click **Apply**  .

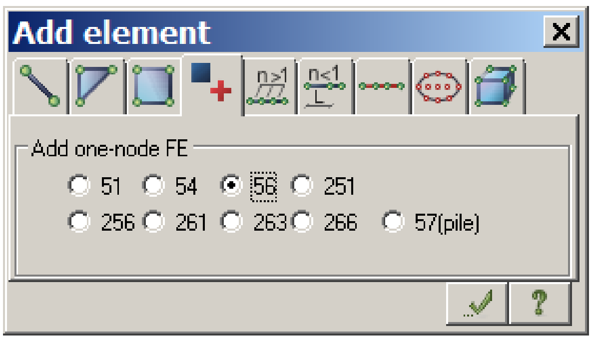



Figure 11.15 **Add element** dialog box

*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.*

## Step 6. Applying loads

To create load case No.1:

- On the **Create and edit** ribbon tab, select the **Loads** panel and click **Add dead weight** .
- In the **Add dead weight** dialog box (see Figure 11.16), click **All elements** and specify **Load factor** as equal to **1**. Then click **Apply** (dead weight of elements is added).

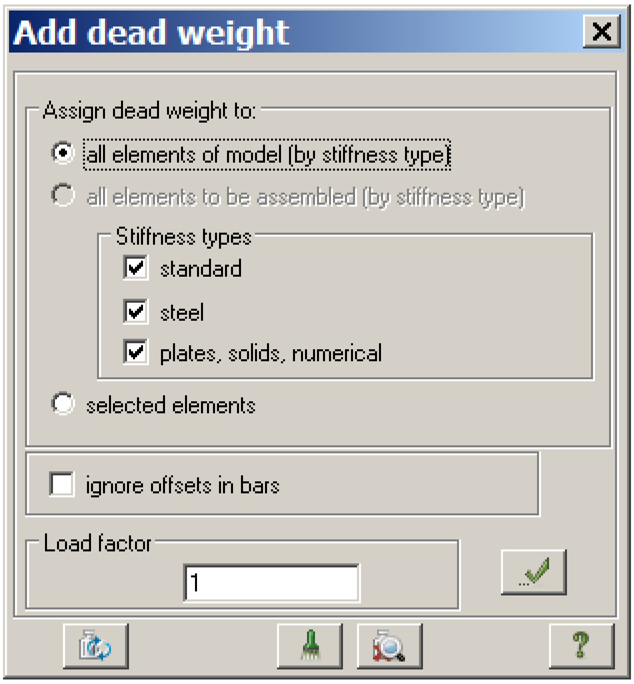


Figure 11.16 **Add dead weight** dialog box

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.
- To select floor and roof slabs, 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 elements** tab (the second tab) (see Fig.11.17).
- Select **By stiffness** check box and specify the line '3.Plate H 20'.
- Click **Apply** .

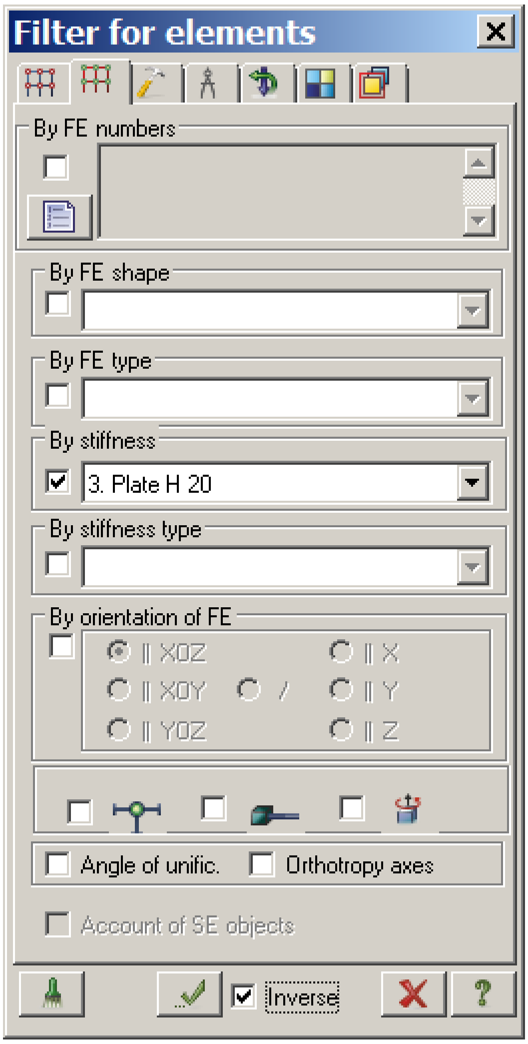



Figure 11.17 **Filter for elements** tab

- 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.11.18), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

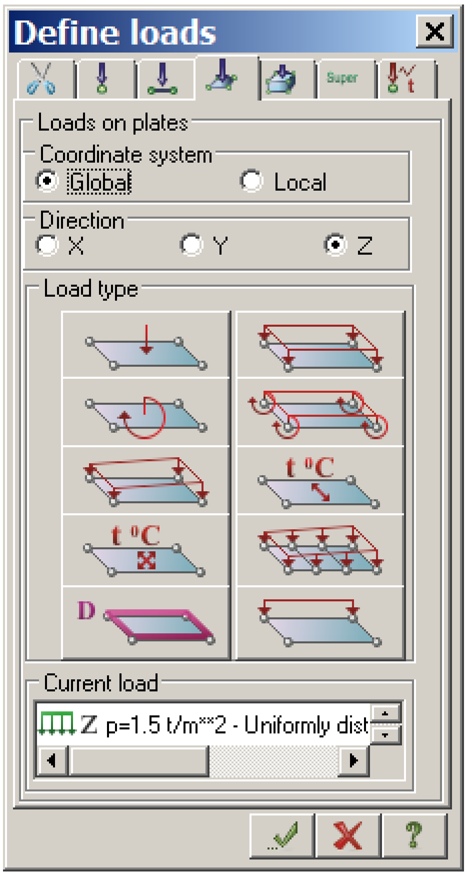
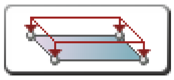



Figure 11.18 **Define loads** dialog box

- In the **Load type** area, click **Uniformly distributed load** button .
- In the **Load parameters** dialog box specify p = 1.5 t/m$^2$ (see Figure 11.19).
- Click **OK** .

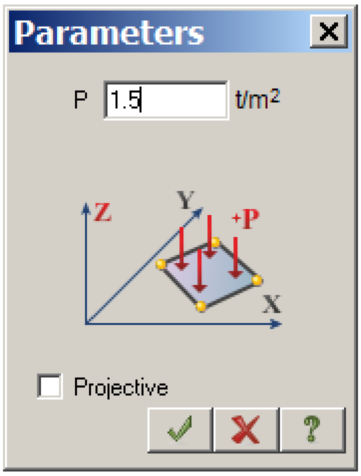



Figure 11.19 **Load parameters** dialog box

- In the **PolyFilter** dialog box, on the **Filter for elements** tab, in the **By stiffness** check box, specify the line '5.Plate H 50'.
- Click **Apply** .

- In the **Define loads** dialog box, click **Uniformly distributed load** button.
- In the **Load parameters** dialog box specify p = 2.0 t/m$^2$.
- Click **OK** .

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.
- In the **PolyFilter** dialog box, on the **Filter for elements** tab (the second tab), in the **By stiffness** check box, specify the line '3.Plate H 20'.
- On the **Projection** toolbar, click **Projection on XOZ-plane** .
- On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .
- Select all elements of the roof with the pointer.

- In the **Define loads** dialog box, click **Uniformly distributed load** button.
- In the **Load parameters** dialog box specify p = 0.08 t/m$^2$.
- Click **OK** .
- To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection** .

To define detailed information about load cases:

- On the **Create and edit** ribbon tab, select the **Loads** panel and click **Edit load cases** . The **Edit load cases** dialog box is displayed on the screen (see Fig.11.20).
- 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** .

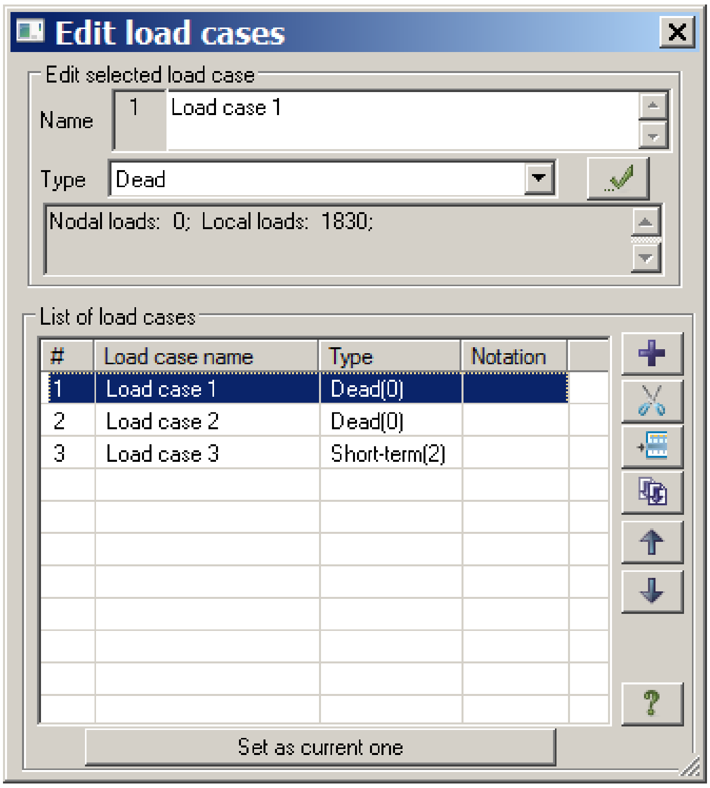



Figure 11.20 **Edit load cases** dialog box

## Step 7. Generating DCF table

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

*As type of load cases was defined in the **Edit load cases** dialog box (see Fig.11.20), then DCF table was automatically generated with parameters accepted by default for every load case. Now you should modify parameters only for the third load case.*

➪ In the **Design combinations of forces** dialog box (see Figure 11.21), select building code **SNIP 2.01.07-85*** and specify the following data:

- for Load case 3 – in the **Summary table for DCF calculation**, define **Load factor** as equal to 1.4 and then click **Apply** .

➪ Click **OK** .

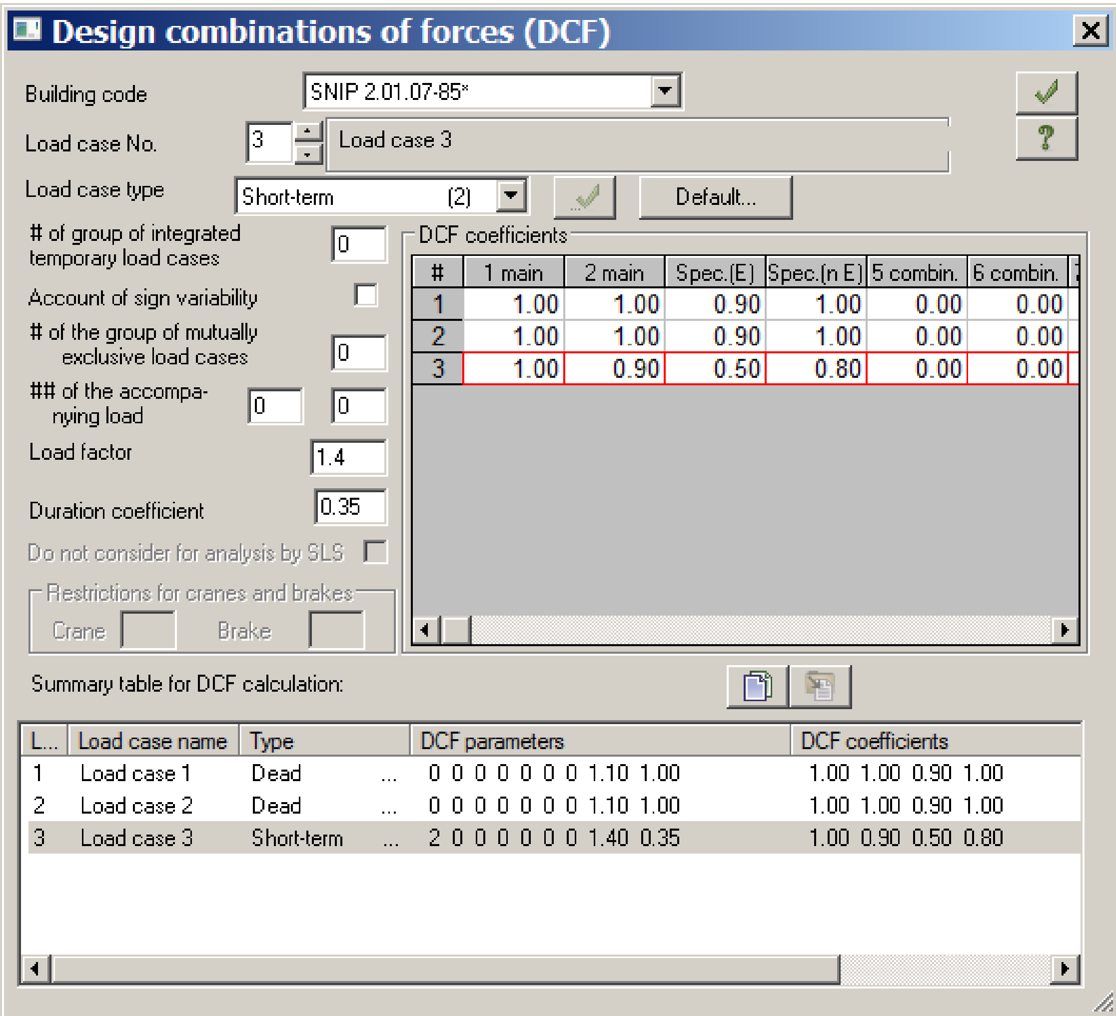



Figure 11.21 **Design combinations of forces** dialog box

## Step 8. Complete analysis of the model

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

## Step 9. 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.11.22).

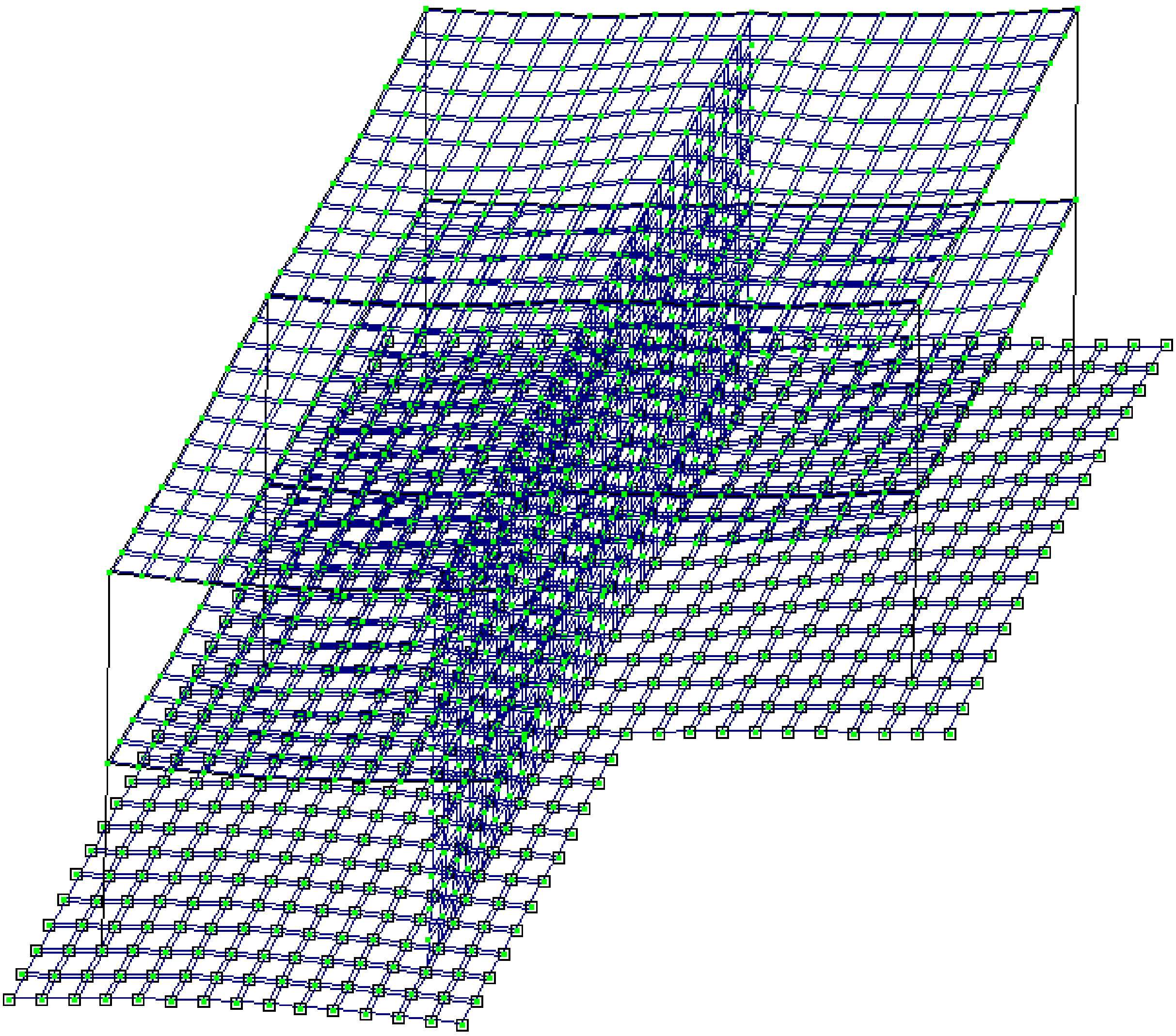


Figure 11.22 Design model with account of nodal displacements

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).
- Clear **By stiffness** option, then select **By FE shape** check box and select **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 Mx, 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 Mx** button on the same panel.
- To present stress mosaic plot for Nx, click **Stress Nx** button on the same panel.
- To present stress mosaic plot for Rz (soil pressure), click **Stress Rz** button on the same panel.
- To present the full picture of stress mosaic plots for Rz in base slab, select the slab with the structural block option and perform fragmentation.
- To restore design model in initial view, on the **Select** toolbar, click **Restore model** .

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.11.23), select **Forces** in the list.
- Click **Apply**.

*By default, standard tables are generated in the ***.csv** format. Information presented in these tables is divided into different tabs: input data (optional), e.g. DCF coefficients; output data for bars; output data for plates; etc.*
*To generate table in ***.csv** format and add it to the Report Book, select the **Generate updatable table in Report Book** check box. If the table is located in the Report Book, it is possible to update it later (if*

*required) and add it to the report file with the Report Book options.*
*To modify format of the table, in the **Standard tables** dialog box, click **Select format**. Then in the **Table format** dialog box, select appropriate option and click **OK**. To generate table in Document Maker (DOC-SAPR module), select RPT format.*
*Selected format is saved and will be applied by default in further work with standard tables.*

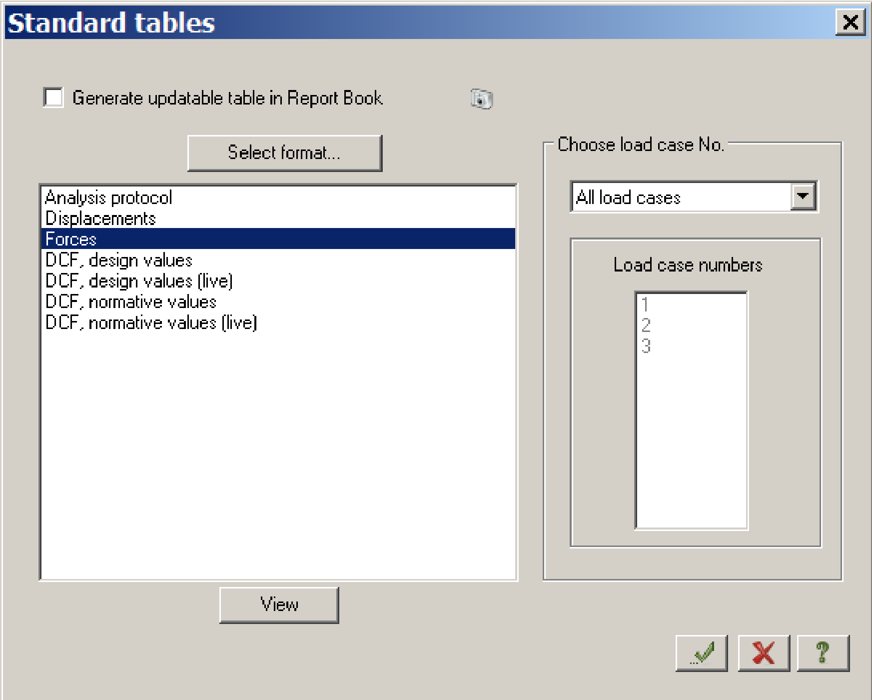


Figure 11.23 **Standard tables** dialog box

- To close the table, on the FILE menu, click **Close**.
- Then save current problem.

## Step 10. Generating the second variant of the problem

To save the problem under another name:

- On the **LIRA-SAPR menu** (Application menu), click **Save as** command .
- In the **Save as** dialog box specify the following data:
  - file name – **Example11_M2**;
  - location where you want to save this file (**Data** folder is displayed by default).
- Click **Save**.

## Step 11. Editing moduli of subgrade resction (subgrade moduli)

- Select elements of the base slab (use the **Structural blocks** dialog box).
- 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, 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=5000 t/m$^3$.
- Click **Apply** .
- To unselect nodes, on the **Select** toolbar, click **Unselect all** button .

## Defining parameters for earthquake analysis of the model

### Step 12. 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.11.24), 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**.

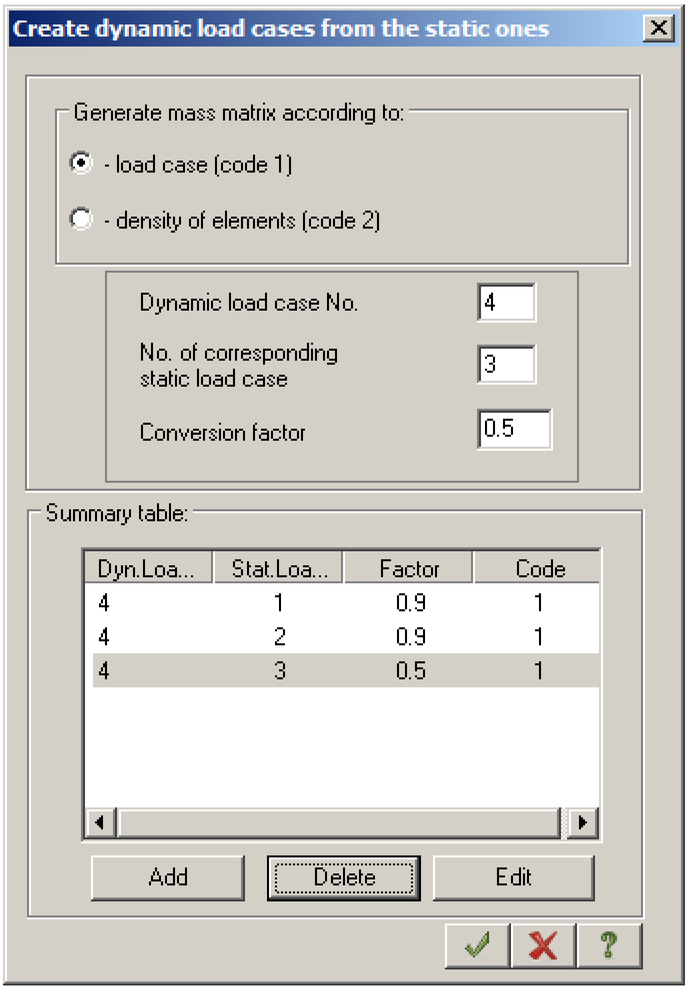
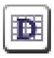



Figure 11.24 **Create dynamic load cases from the static ones** dialog box

## Step 13. Generating table of dynamic load cases

*Unfavourable direction of earthquake load is along the smaller side of the structure. As the structure has dimensions 10 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.11.25), 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**.

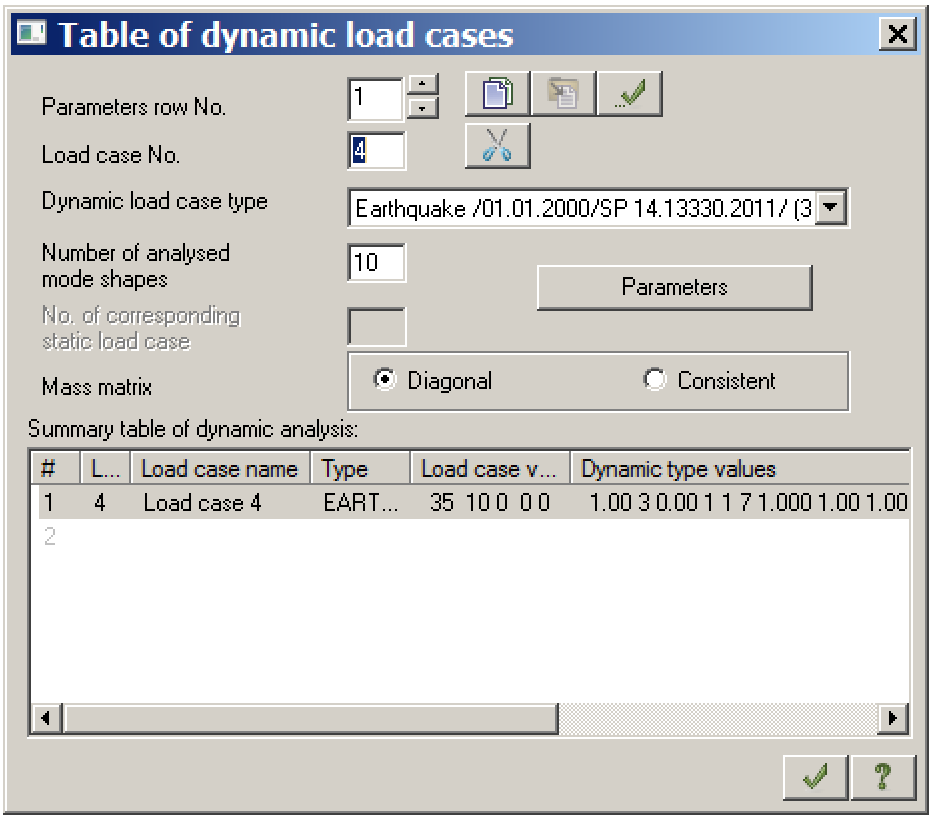



Figure 11.25 **Table of dynamic load cases** dialog box

- In the **Earthquake analysis parameters** dialog box (see Fig.11.26), define the following data:
  - direction cosines of earthquake load resultant in global coordinate system – CX = 1;
  - other parameters remain by default.
- Click **OK**  .

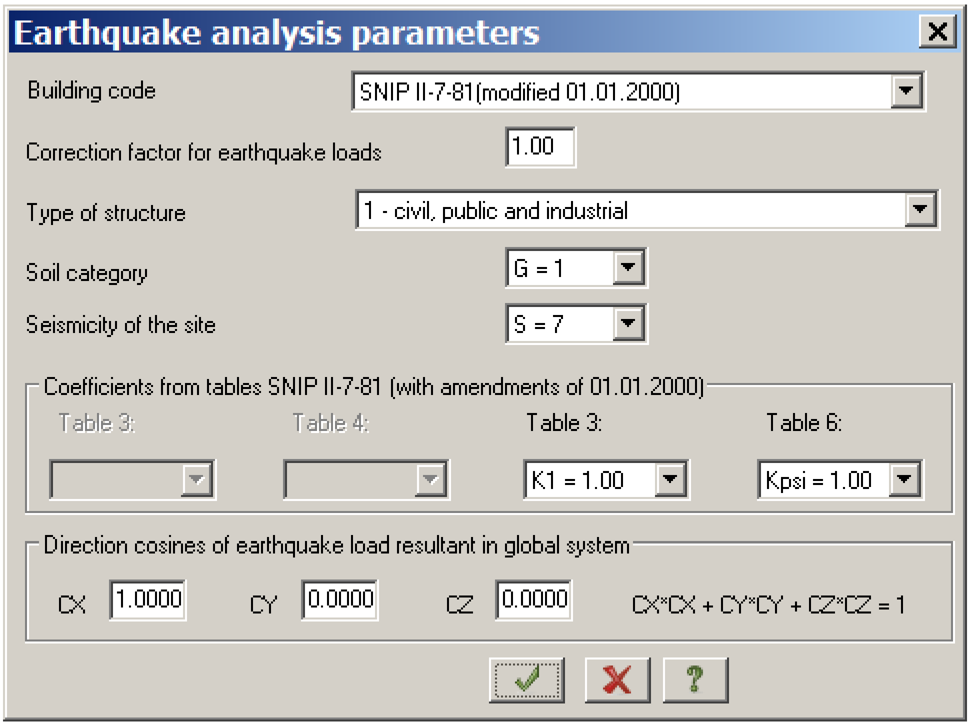



Figure 11.26 **Earthquake analysis parameters** dialog box

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

### Step 14. Editing DCF table

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

➪ In the **Design combinations of forces** dialog box, specify the following data:

- in the **Summary table for DCF calculation**, select the row for the 4th load case.
- for Load case 4 – in the **Load case type** list, click **Earthquake (5)** and then click **Apply** .

➪ To generate DCf table, click **OK** .

### Step 15. Complete analysis: the 2nd variant of the model

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

### Step 16. Review and evaluation of analysis results for the 2nd variant of the 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 1st 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.11.27), click **Play** button .
- To close the **Animate mode shape** dialog box, click **Close**.

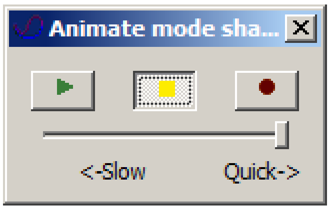



Figure 11.27 **Animate mode shape** dialog box

- 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 periods of vibrations, 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.11.23), select **Periods of vibrations** 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.

- To close the table, on the FILE menu, click **Close**.
- Then save current problem.

- To close the current problem, on the **Application menu**, click **Close**.

## Step 17. Generating the third variant of the problem

To save the problem under another name:

- To open problem **Example11_M**, click it on the **LIRA-SAPR menu** (Application menu) in the list of recent files.
- To save the problem under another name, on the **LIRA-SAPR menu** (Application menu), click **Save as** command .
- In the **Save as** dialog box specify the following data:
  - file name – **Example11_M3**;
  - location where you want to save this file (**Data** folder is displayed by default).
- Click **Save**.

## Step 18. Editing moduli of subgrade reaction (subgrade moduli) for the third variant of design model

*For this variant it is considered that there are no subgrade moduli in the first span of base slab. But in METEOR system it is required that number of forces in sections of identically placed elements of*

*integrated models should be the same. That's why, for this element we will define subgrade modulus as equal to 0.01 t/m³. In this case, we will obtain soil pressure Rz and its value will have little or no effect on analysis.*
*The same values should be defined for the fourth variant of the model.*

- To present design model in projection on the XOZ-plane, on the **Projection** toolbar, click **Projection on XOZ-plane** .
- On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .
- Select with the pointer only elements of projection of the first (left) span 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, 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=0.01 t/m³.
- Click **Apply** .
- To unselect nodes, on the **Select** toolbar, click **Unselect all** button .
- To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection** .
- Then carry out analysis procedure and evaluate analysis results in a similar way to the first variant of the problem.
- Save and close the current problem.

## Step 19. Generating the fourth variant of the problem

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

- To open problem **Example11_M**, click it on the **LIRA-SAPR menu** (Application menu) in the list of recent files.
- Save the problem under another name **Example11_M4** .

## Step 20. Editing moduli of subgrade reaction (subgrade moduli) for the fourth variant of design model

- To present design model in projection on the XOZ-plane, on the **Projection** toolbar, click **Projection on XOZ-plane** .
- On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .
- Select with the pointer only elements of projection of the second (right) span 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, 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=0.01 t/m³.
- Click **Apply** .
- To unselect nodes, on the **Select** toolbar, click **Unselect all** button .
- To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection** .

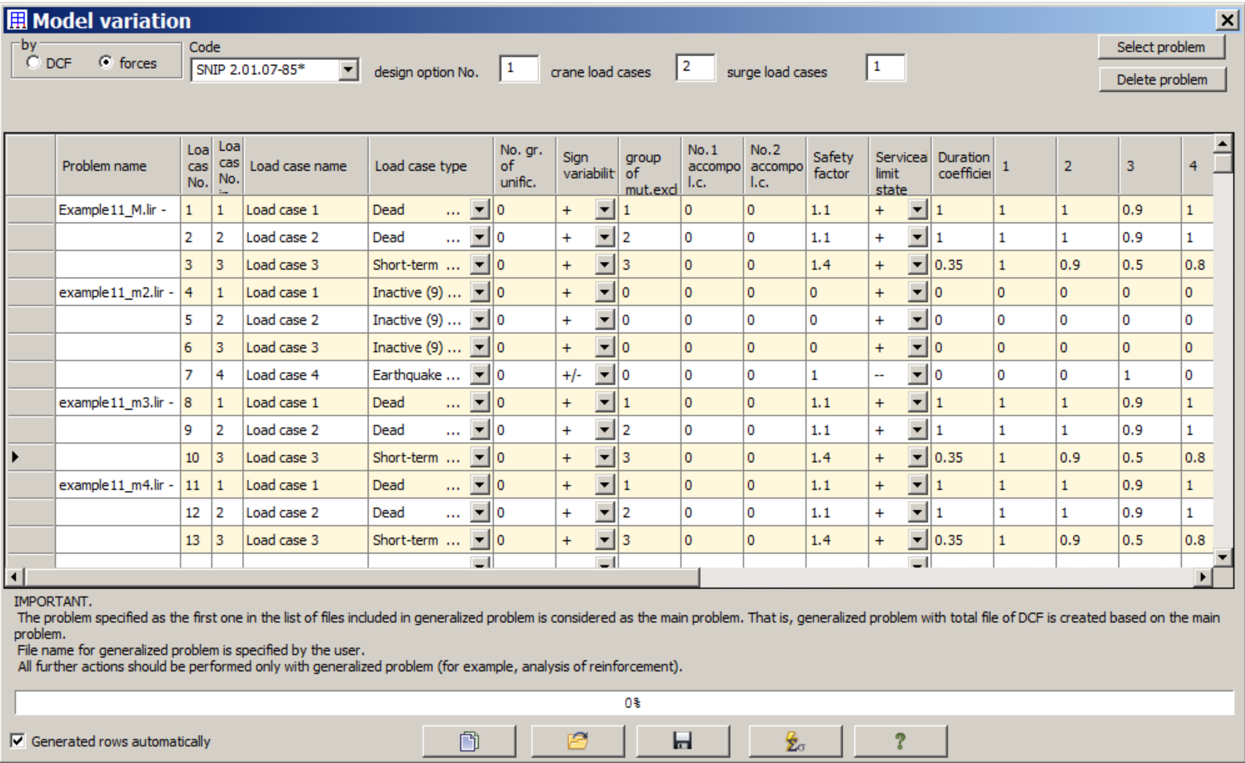
- Then carry out analysis procedure and evaluate analysis results in a similar way to the first variant of the problem.
- Save and close the current problem.

## Step 21. Generating and carrying out analysis of integrated DCF

*Data for integrated DCF problem is generated in the **Model variation (METEOR)** dialog box (see Fig.11.28). To open this dialog box, in the initial mode of the program, on the **Analysis** ribbon tab, on the **Analysis** panel, click **Model Variation (METEOR)** button . This command is active only in initial mode of the program (if no file with input data is presented in the window).*
*Analysis of integrated DCF may be carried out **by DCF** or **forces** of all problems included in a set.*
*If analysis is carried out **by DCF**, you could unite analysis results of topologically identical design models. In analysis of integrated problem it is supposed that DCF calculated for all problems in the list are mutually exclusive ones. Mutual exclusion is made separately for every DCF criterion.*
*If analysis is carried out **by forces**, DCF are calculated for the whole set of problems. In this case, DCF are calculated according to numbers of load cases in input table of METEOR system. DCF for integrated problem are calculated as for the standard problem. You could edit input table in METEOR system.*

To generate the list of topologically identical design models:

- In the initial mode of the program, on the **Analysis** ribbon tab, on the **Analysis** panel, click **Model variation (METEOR)** button .
- To generate the list (set) of topologically identical design models, in the **Model variation (METEOR)** dialog box (see Fig.11.28), click **Select problem** button.




| Problem name | Loa cas No. | Loa cas No. | Load case name | Load case type | No. gr. of unific. | Sign variabilit | group of mut.excl | No.1 accompo l.c. | No.2 accompo l.c. | Safety factor | Servicea limit state | Duration coefficie | 1 | 2 | 3 | 4 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Example11_M.lir - | 1 | 1 | Load case 1 | Dead ... | 0 | + | 1 | 0 | 0 | 1.1 | + | 1 | 1 | 1 | 0.9 | 1 |
| | 2 | 2 | Load case 2 | Dead ... | 0 | + | 2 | 0 | 0 | 1.1 | + | 1 | 1 | 1 | 0.9 | 1 |
| | 3 | 3 | Load case 3 | Short-term ... | 0 | + | 3 | 0 | 0 | 1.4 | + | 0.35 | 1 | 0.9 | 0.5 | 0.8 |
| example11_m2.lir - | 4 | 1 | Load case 1 | Inactive (9) ... | 0 | + | 0 | 0 | 0 | 0 | + | 0 | 0 | 0 | 0 | 0 |
| | 5 | 2 | Load case 2 | Inactive (9) ... | 0 | + | 0 | 0 | 0 | 0 | + | 0 | 0 | 0 | 0 | 0 |
| | 6 | 3 | Load case 3 | Inactive (9) ... | 0 | + | 0 | 0 | 0 | 0 | + | 0 | 0 | 0 | 0 | 0 |
| | 7 | 4 | Load case 4 | Earthquake ... | 0 | +/- | 0 | 0 | 0 | 1 | -- | 0 | 0 | 0 | 1 | 0 |
| example11_m3.lir - | 8 | 1 | Load case 1 | Dead ... | 0 | + | 1 | 0 | 0 | 1.1 | + | 1 | 1 | 1 | 0.9 | 1 |
| | 9 | 2 | Load case 2 | Dead ... | 0 | + | 2 | 0 | 0 | 1.1 | + | 1 | 1 | 1 | 0.9 | 1 |
| | 10 | 3 | Load case 3 | Short-term ... | 0 | + | 3 | 0 | 0 | 1.4 | + | 0.35 | 1 | 0.9 | 0.5 | 0.8 |
| example11_m4.lir - | 11 | 1 | Load case 1 | Dead ... | 0 | + | 1 | 0 | 0 | 1.1 | + | 1 | 1 | 1 | 0.9 | 1 |
| | 12 | 2 | Load case 2 | Dead ... | 0 | + | 2 | 0 | 0 | 1.1 | + | 1 | 1 | 1 | 0.9 | 1 |
| | 13 | 3 | Load case 3 | Short-term ... | 0 | + | 3 | 0 | 0 | 1.4 | + | 0.35 | 1 | 0.9 | 0.5 | 0.8 |

Figure 11.28 **Model variation (METEOR)** dialog box

- In the **Open** dialog box (see Fig.11.29), open folder where all analysed problems are saved (by default, *Data* folder), select the *Example11_M.lir* file in the list and click **Open**.

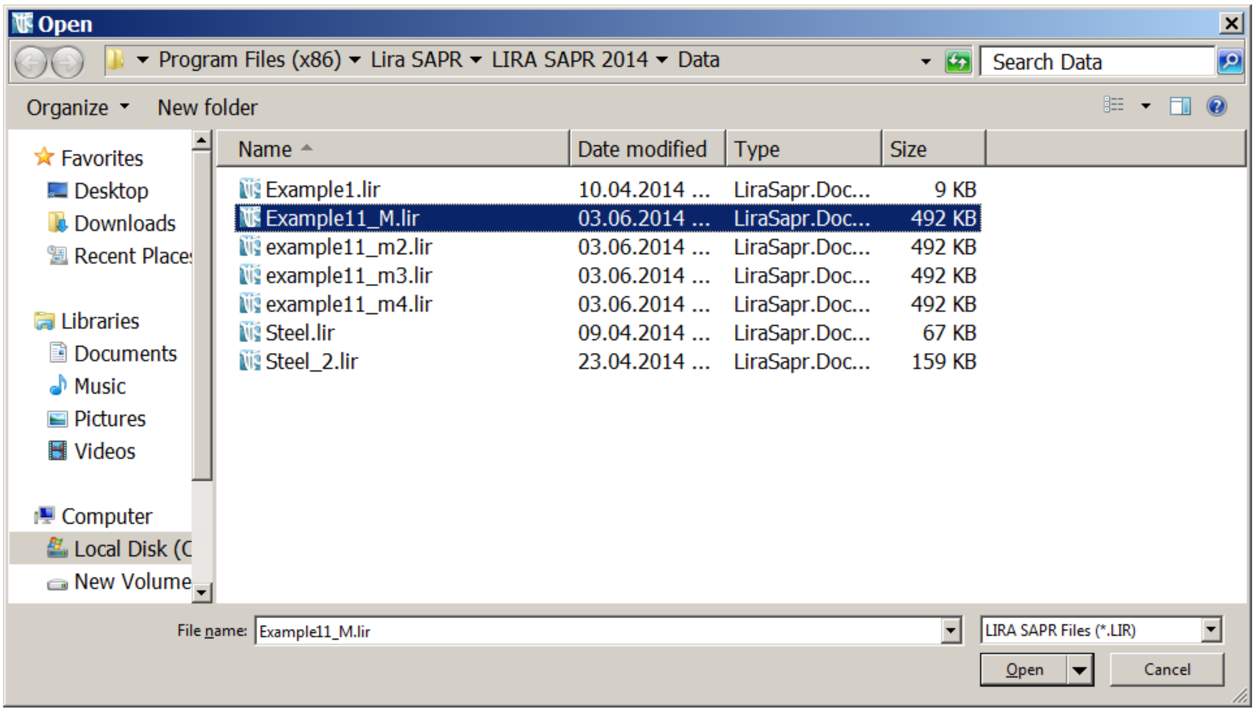



Figure 11.29 **Open** dialog box

- To add the second problem to the list of topologically identical design models, in the **Model variation (METEOR)** dialog box, click **Select problem** button.
- In the **Open** dialog box, open the *Example11_M2.lir* file.
- To add the third problem to the list of topologically identical design models, in the **Model variation (METEOR)** dialog box, click **Select problem** button.
- In the **Open** dialog box, open the *Example11_M3.lir* file.
- To add the fourth problem to the list of topologically identical design models, in the **Model variation (METEOR)** dialog box, click **Select problem** button.
- In the **Open** dialog box, open the *Example11_M4.lir* file.

To edit input table:

*Analysis of integrated DCF will be carried out by forces. This type of analysis is stipulated by default when you open the **Model variation (METEOR)** dialog box.*

- In the **Model variation (METEOR)** dialog box, make sure that **by forces** option is selected for analysis, double-click the **Load case type** cell and for the load cases with sequential numbers 4, 5 and 6 (that is, the 1st, 2nd and 3rd load cases of the second problem), define the following types of load cases:
  - for the 4th load case – Inactive (9);
  - for the 5th load case – Inactive (9);
  - for the 6th load case – Inactive (9).
- Then for the load cases with sequential numbers 1, 8 and 11, in the **group of mutual exclusion** cell, define group number 1.
- For the load cases with sequential numbers 2, 9 and 12, in the **group of mutual exclusion** cell, define group number 2.
- For the load cases with sequential numbers 3, 10 and 13, in the **group of mutual exclusion** cell, define group number 3.
- To save the input table, click the **Save file of integrated problem** button .

- In the **Save as** dialog box, click **Save**.
- To carry out analysis of integrated DCF, in the **Model variation (METEOR)** dialog box, click **Calculate** button .
- Close the **Model variation (METEOR)** dialog box and review analysis results of integrated problem.

*To carry out analysis of integrated DCF by DCF of every problem included in a set, it is necessary to generate the list of topologically identical design models as described above (in this example you could generate the list from the 1st, 3rd and 4th problems). Then click **by DCF** option in the dialog box (this option may be selected before you generate the list as well). In this case, there will be no option to edit the input table. To carry out analysis, click the **Calculate** button in the dialog box.*

## Step 22. Review and evaluation of analysis results for the integrated problem

*After analysis of integrated DCF, the program generates new file of the problem with the name that was mentioned when you save the input table of METEOR system. For this file all options to edit design model will become unavailable; it is possible to edit only data for design of elements of design model.*
*When you carry out analysis of integrated DCF **by forces**, in the file generated according to analysis there will be analysis results (displacements, forces and stresses in elements, results of dynamic analysis) by load cases for all problems included into integrated problem and results of the integrated DCF.*
*When you carry out analysis of integrated DCF **by DCF** of every problem included into the set, in the file generated according to analysis there will be only results of integrated DCF.*

Output of displacements, forces and stresses in elements, presentation of dynamic analysis results and generation of DCF table for integrated problem - all these commands are similar to the commands in the standard problem.

## Step 23. Defining design options

*To define design options, material properties and other data for analysis of reinforcement and analysis of steel structures, use buttons presented on the **Design** panel on the **Create and edit** and **Design** ribbon tabs (for the standard ribbon interface).*
*For this example we will define all data necessary for design procedure with the buttons presented on the **Design** panel on the **Design** ribbon tab.*

- On the **Design** ribbon tab, on the **Design** panel, click **Design options for main model** command .
- In the **Design options** dialog box (see Fig.11.30), define parameters for the first design option:
  - in the **Analysis of sections by** list, select **DCF**;
  - other parameters remain by default.
- Click **Apply** .

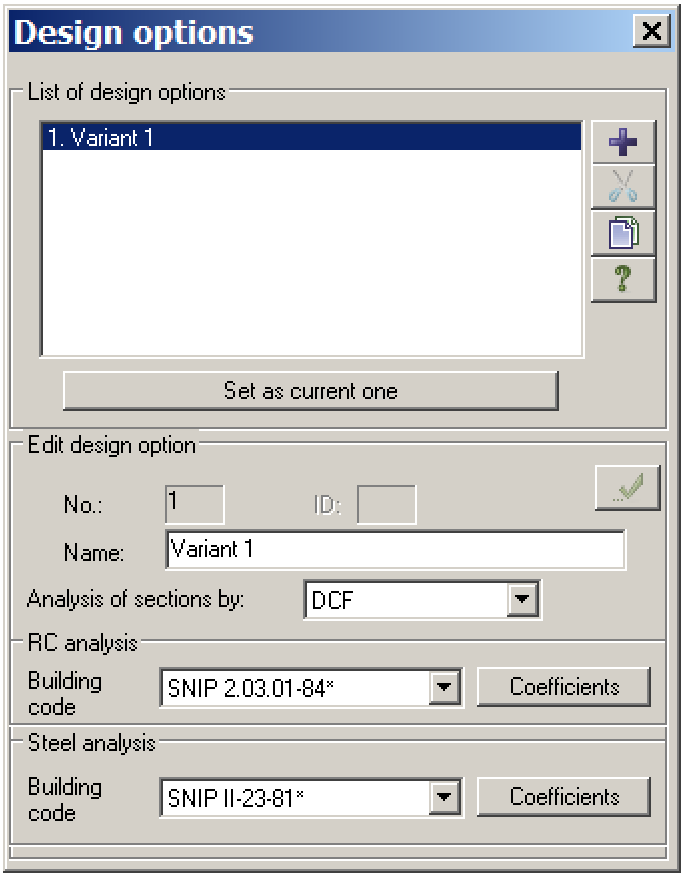
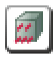



Figure 11.30 **Design options** dialog box

- To close the **Design options** dialog box, click the **Close** button.

## Step 24. Defining material properties to elements of the model

To create material data sets for RC structures:

- On the **Design** ribbon tab, on the **Design** panel, click **Material properties (RC)** button  .
- The **Stiffness and materials** dialog box (see Fig.11.31) is presented with the second tab **Define parameters for RC structures** open.

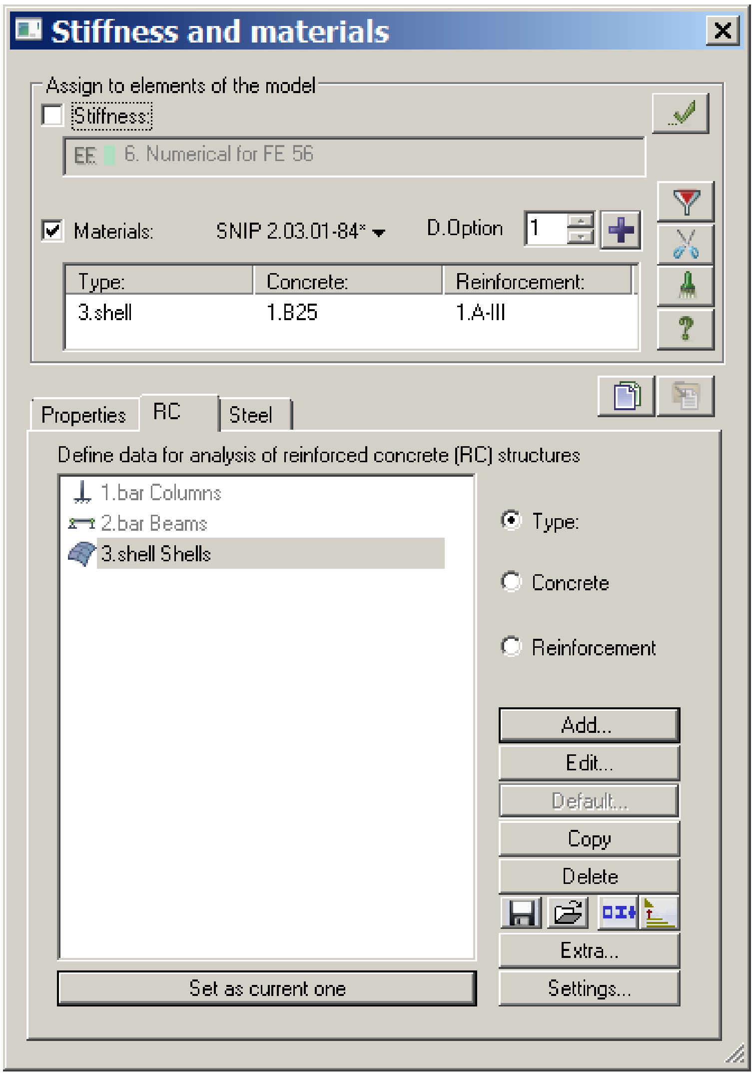



Figure 11.31 **Stiffness and materials** dialog box

- Select **Type** option and click **Add**.
- In the **General parameters** dialog box (see Figure 11.32), define the following parameters for columns:
  - in the **Module of reinforcement** list, select **Bar**;
  - in the **Reinforcement** area, select **Symmetric** type of reinforcement;
  - in the **Design requirements for bars** area, select **Column regular** option and clear the **Do not account design requirements** check box;
  - in the **Analysis according to serviceability limit states** area, select **Diameter** option;
  - in the drop-down list, select diameter of reinforcement 25mm;
  - in the **Length of element, Effective length**, select **Effective length factor** option;
  - define parameters LY = 0.7, LZ = 0.7;
  - in the **Comment** box, type comment - **Column**;
  - other parameters remain by default.
- Click **OK** .

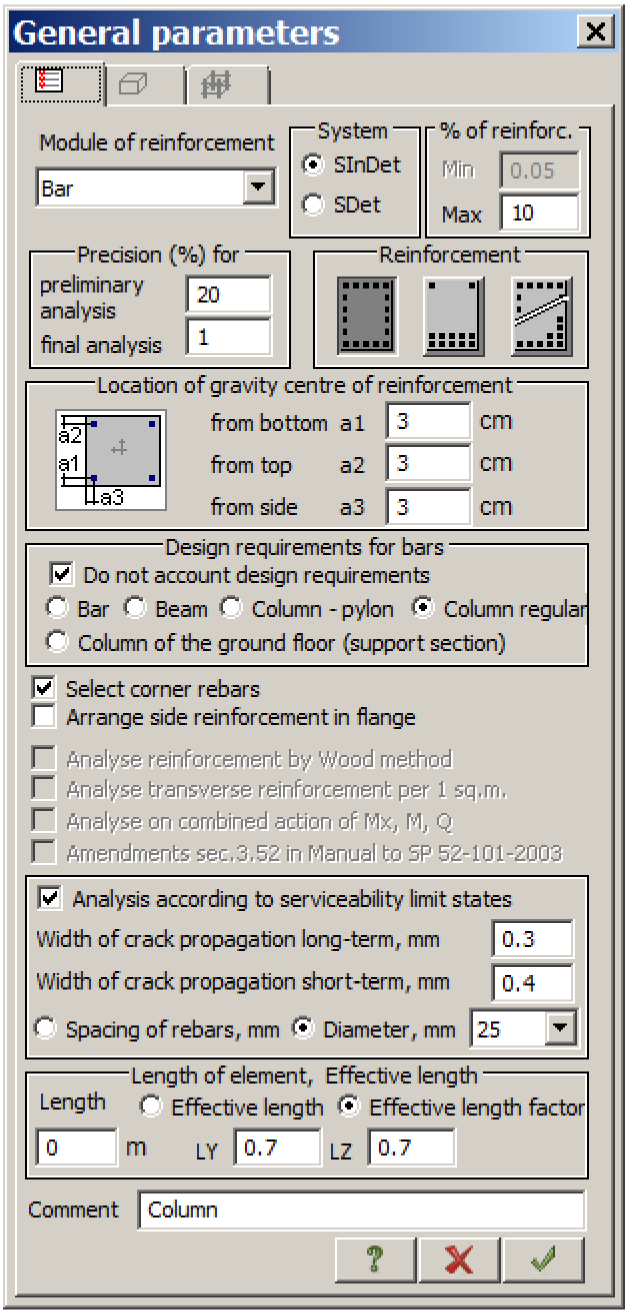



Figure 11.32 **General parameters** dialog box

- In the **Stiffness and materials** dialog box, click **Add** once again.
- In another **General parameters** dialog box, define parameters for beams:
  - in the **Reinforcement** area, select **Asymmetric** type of reinforcement;
  - in the **Design requirements for bars** area, select **Beam** option and clear the **Do not account design requirements** check box;
  - in the **Analysis according to serviceability limit states** area, select **Diameter** option;
  - in the drop-down list, select diameter of reinforcement 25mm;
  - in the **Length of element, Effective length**, select **Effective length factor** option;
  - define parameters LY = 0, LZ = 0;
  - in the **Comment** box, type comment - **Beam**;
  - other parameters remain by default.
- Click **OK** .

- In the **Stiffness and materials** dialog box, click **Add** once again.
- In another **General parameters** dialog box, define parameters for plate elements:
  - in the **Module of reinforcement** list, select **Shell**;
  - in the **Comment** box, type comment - **Shell**;
  - other parameters remain by default.
- Click **OK** .

- 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).

To assign stiffness and material properties to elements of the model:

- On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .
- Select all elements of the model with the pointer (selected elements will be coloured red).
- 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.
- The **Warning** box is displayed (see Fig.11.33). Click **OK**.

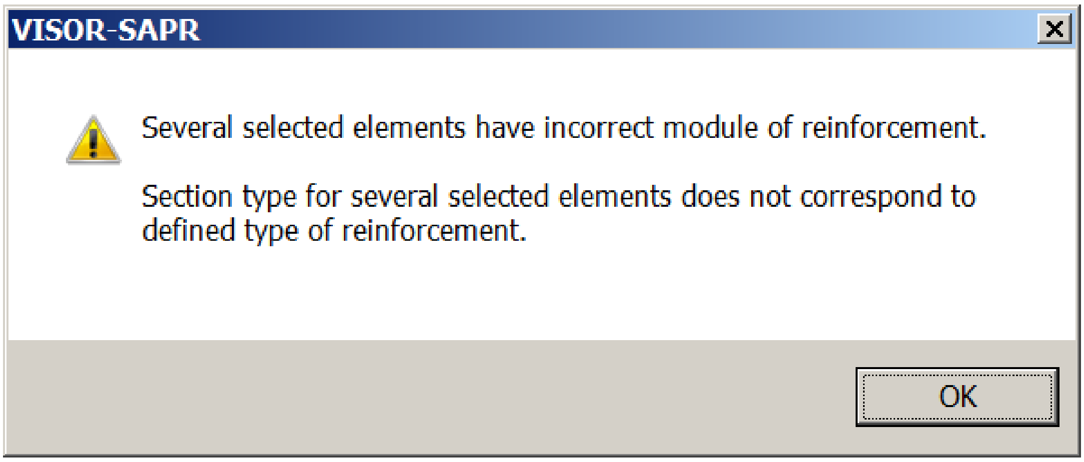



Figure 11.33 **Warning** dialog box

- In the **Stiffness and materials** dialog box, on the **RC** tab, select **Type** option and in the list of types of material properties for RC structures, select the line **2.bar Beam**.
- Click **Set as current type** (In this case selected type of material properties will be displayed in the **Materials** 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**.)

- In the **Stiffness and materials** dialog box, click **Apply** .
- In the first **Warning** box, click **OK**. In the second **Warning** box, click **No**.

- To unselect 1-node elements, on the **Select** toolbar, click **Unselect all** button .

- In the **Stiffness and materials** dialog box, on the **RC** tab, select **Type** option and in the list of types of material properties for RC structures, select the line **1.bar Column**.
- 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**  .

## Step 25. Carrying out analysis of reinforcement

➪ To carry out analysis of reinforcement, on the **Design** ribbon tab, on the **RC: Analysis** panel, click **Analyse reinforcement** button  .

➪ In the **RC analysis** dialog box (see Fig.11.34), click **Analyse**.

➪ When analysis is complete, close the dialog box.

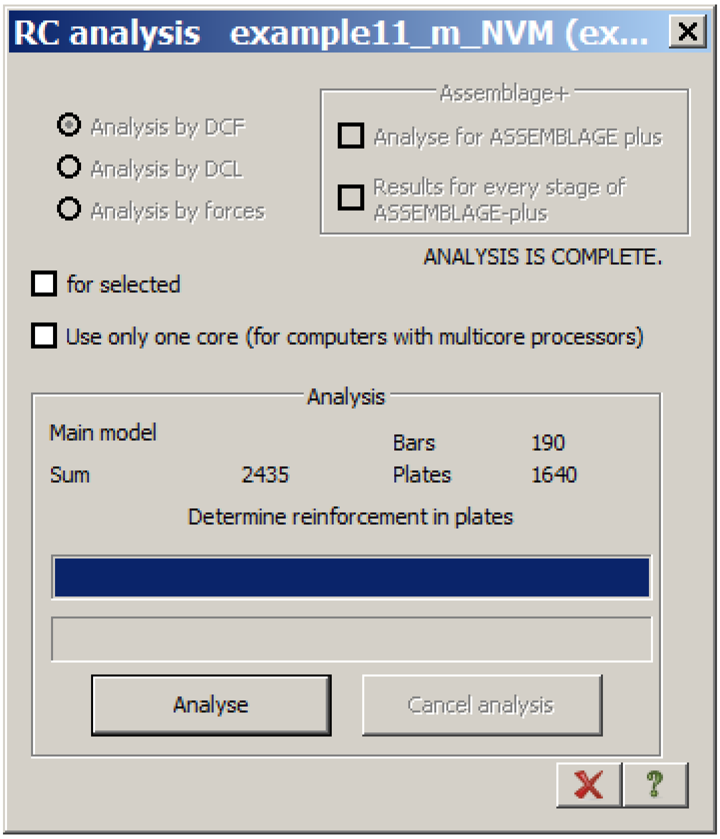



Figure 11.34 **RC analysis** dialog box

## Step 26. Review and evaluate results from analysis of reinforcement

To present results from analysis of reinforcement:

- To present information about determined reinforcement in a certain element, on the **Select** toolbar, click **Information about nodes and elements** button and specify with a pointer any bar or 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 switch to the mode for presentation of symmetric reinforcement in rebars, on the **Design** ribbon tab, select **RC: Bars** panel and click **Symmetric reinforcement** command in the **Reinforcement** drop-down list.
- To display mosaic plot for area of longitudinal reinforcement in the lower left corner of the section AU1, click the **Corner reinforcement AU1** button (on the **Design** ribbon tab, the **RC: Bars** panel).
- To display mosaic plot for area of longitudinal reinforcement in the lower right corner of the section AU2, click the **Corner reinforcement AU2** button (on the **Design** ribbon tab, the **RC: Bars** panel).
- To switch to the mode for presentation of asymmetric reinforcement in rebars, on the **Design** ribbon tab, select **RC: Bars** panel and click **Asymmetric reinforcement** command in the **Reinforcement** drop-down list.

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.11.35), the following data is mentioned by default: **Reinforcement in bars** option in the **Elements** area and **For all elements** option in the **Create table** area.
- Click **Table - on the screen** (It is also possible to add the table to the Report Book and select format for the table, in a similar manner to that used for standard tables).

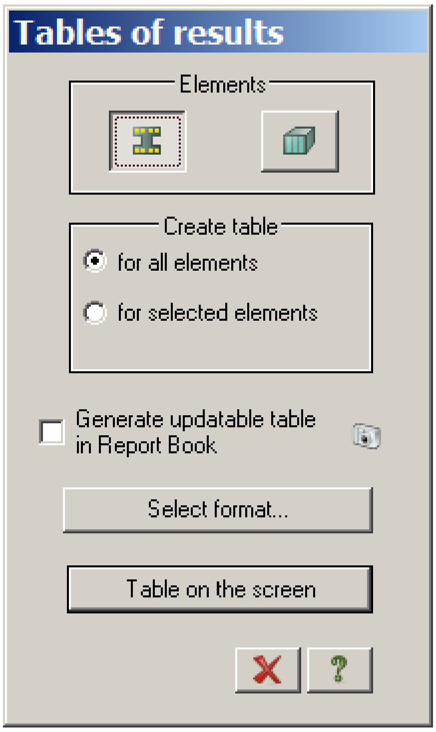


Figure 11.35 **Tables of analysis results** dialog box