Example 22. Analysis of structure on piled foundation with calculation of pile stiffness in the SOIL system (new FE 57)

In this lesson you will learn how to:
– generate design model;
– define piled foundation;
– compute pile stiffness in the SOIL system.

Description:
One span, one storey building. Span length – 6 m, column step – 6 m, height of storey – 4 m.
Side overhangs of grillage – 1 м

Sections for elements:
– beam – T-section of height 500 mm (flange width – 500 mm, flange thickness – 200 mm, web thickness – 300 mm);
– column – rectangular section with dimensions 400 x 400 mm;
– floor slab – thickness 200 mm;
– grillage – thickness 500 mm.

Material for elements of the model – reinforced concrete B25.

Loads:
– load case 1 – dead weight;
– load case 2 – dead uniformly distributed load p = 1.5 t/m^2 applied to floor slab;
dead uniformly distributed load p = 2 t/m^2 applied to grillage;
– load case 3 – wind load p = 0.35 t/m^2.

➪ 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.22.1) specify the following data:

- problem name – **Example22**;
- model type – **5 – Six degrees of freedom per node** (translations X, Y, Z and rotations Ux, Uy, Uz).

➪ Click **OK** .

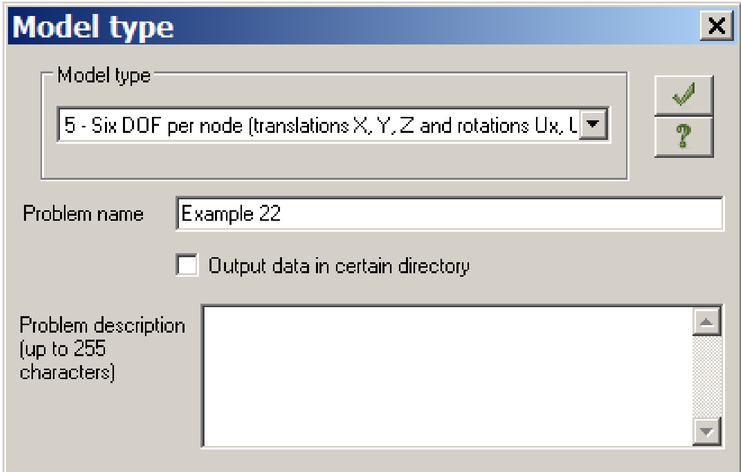


Figure 22.1 **Model type** dialog box

*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

- 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.22.2), specify the following data:
  - spacing along X: **L(m) N M** 6 1 12; spacing along Y: **L(m) N M** 6 1 12; spacing along Z: **L(m) N M** 4 1 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**.

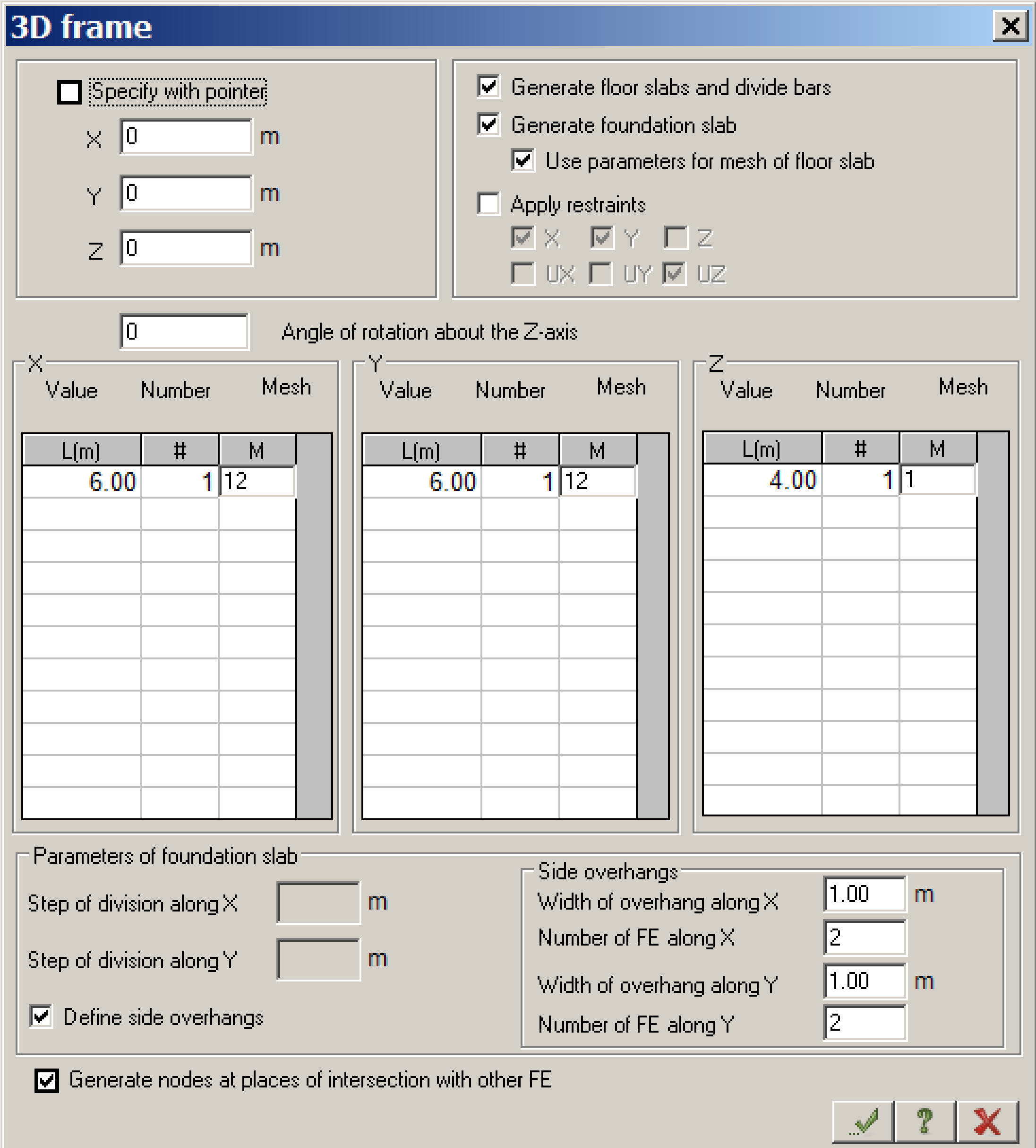


Figure 22.2 **3D frame** dialog box

<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 – **Example22**;
- location where you want to save this file (**Data** folder is displayed by default).

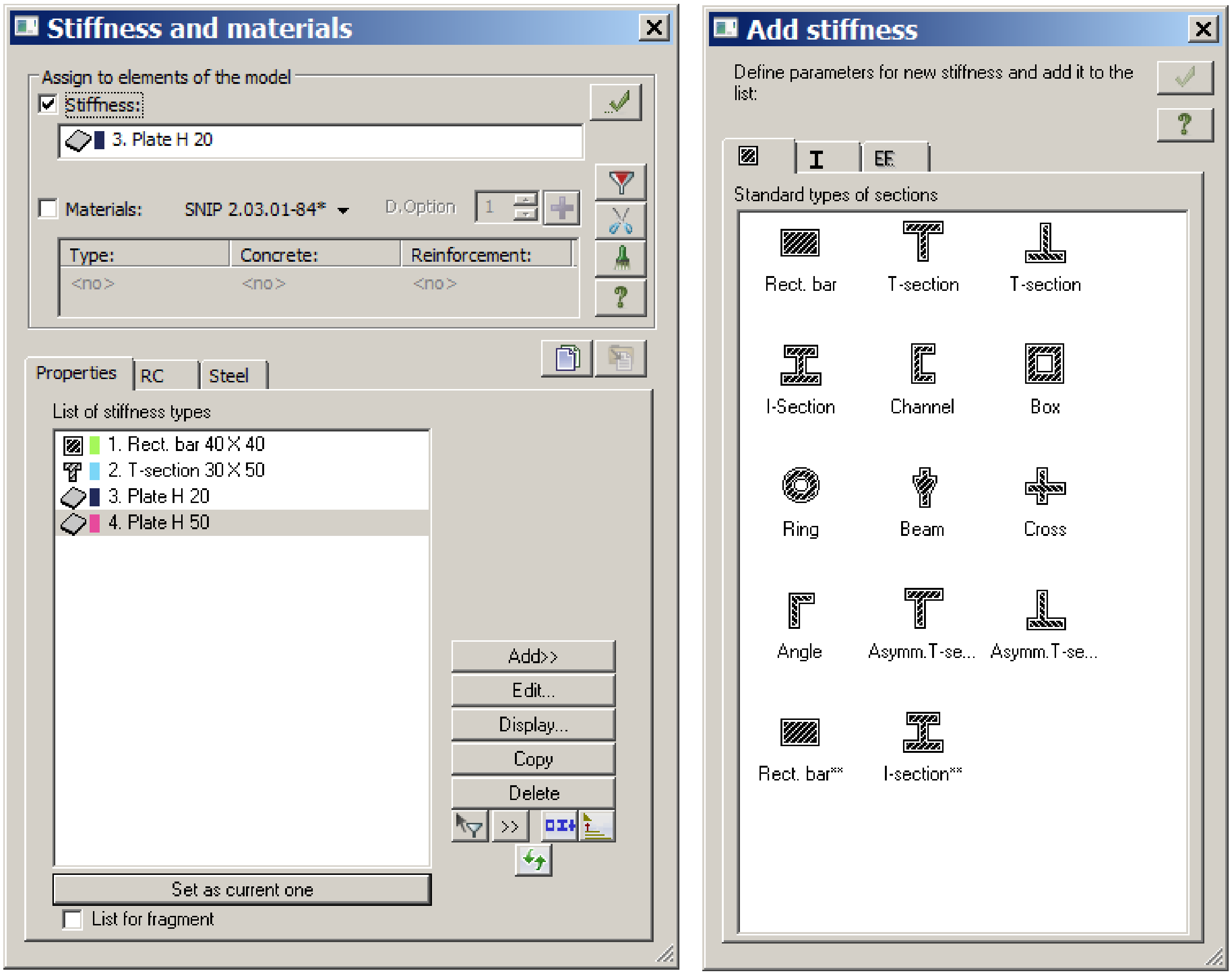
➪ Click **Save**.

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.22.3a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.22.3b).



a b

Figure 22.3 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.22.4):
  - 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 – R_o = 2.75 t/m^3.
- To preview schematic presentation, click **Draw**.
- To confirm the specified data, click **OK** .

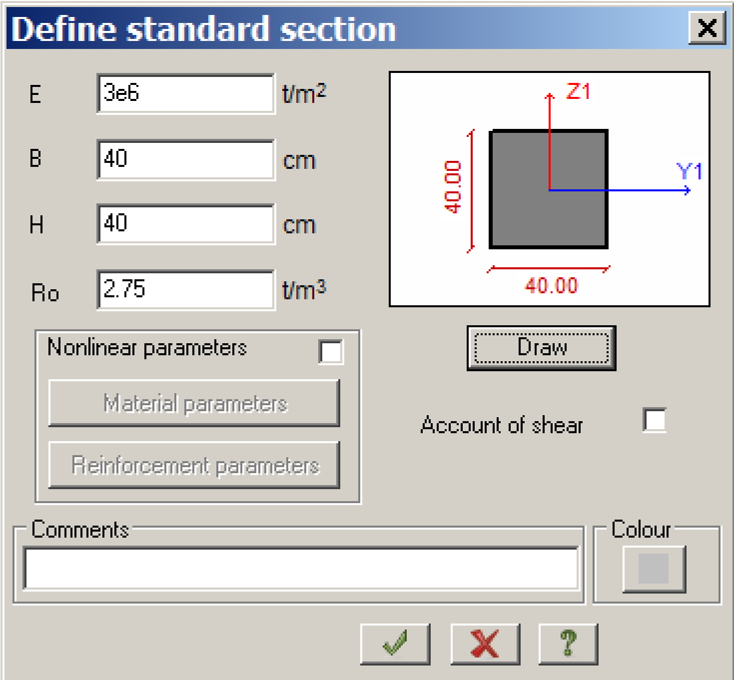


Figure 22.4 **Define standard section** dialog box

➪ 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 = 50 cm; B1 = 50 cm; H1 = 20 cm;
- unit weight of material – R_o = 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.22.5), 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** .

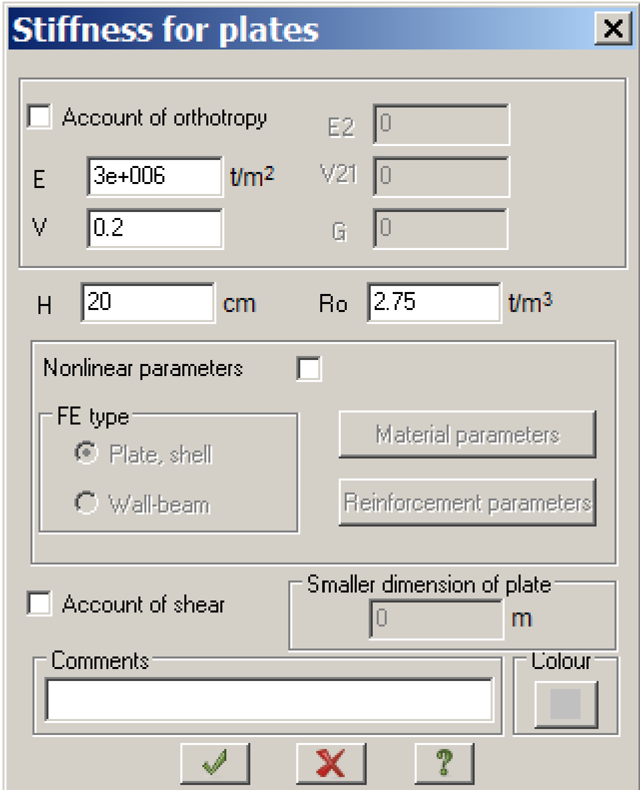


Figure 22.5 **Stiffness for plates** dialog box

- In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select ‘3.Plate H 20’.
- Click **Copy**.
- 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 grillage):
  - thickness - H = 50 cm.
- Click **OK** .

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

To assign stiffness to elements of the model:

- To select elements of the model, on the **Select** toolbar, click **Select blocks** button .
- Specify with the pointer any node or element of floor slab (nodes and elements of floor slab as well as beams and columns will be coloured red).

- In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select stiffness type '3.Plate H 20'.
- 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** .

- The **Warning** box is displayed (see Fig.22.6). Click **OK**.

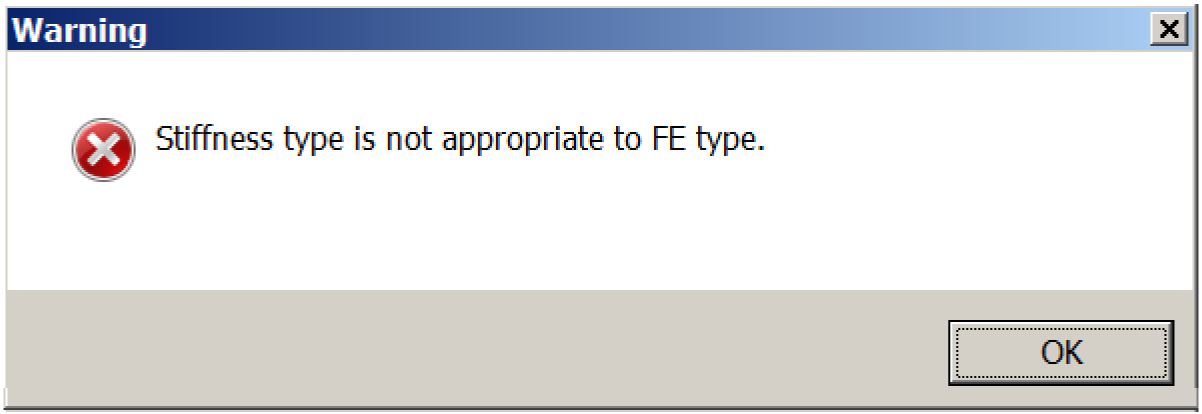


Figure 22.6 **Warning** dialog box

This message is displayed because bar and plate elements were selected on design model as well as floor slab. But this type of stiffness is not allowed for bar elements.

- In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select stiffness type '2.T-section 30x50'.
- 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.

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

- In the **Stiffness and materials** dialog box, click **Apply** .
- In the **Warning** box, click **OK**.

- In the **Stiffness and materials** dialog box, in the list of stiffness types, select ‘4.Plate H 50’.
- Click **Set as current type**.
- On the **Select** toolbar, click **Select blocks** button .
- Specify with the pointer any node or element of grillage.
- 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** .

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

Step 4. Defining parameters of piled foundation

- On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **Flags of drawing** button  .
- In the **Display** dialog box (see Fig.22.7), select the **Node numbers** check box on the **Nodes** tab.
- Click **Redraw**  .

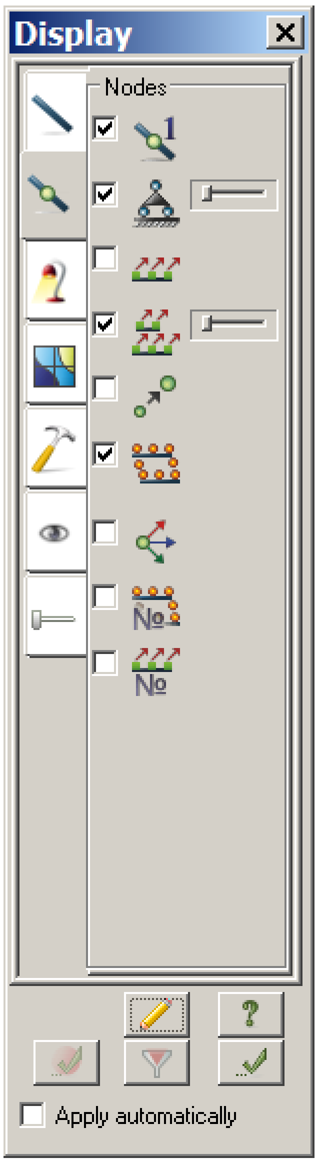


Figure 22.7 **Display** dialog box

To add FE 57:

- On the **Select** toolbar, click **Select block** button  .
- Select with the pointer any node or element of the grillage.
- On the **Select** toolbar, click **Select vertical bars** button  .

- Select all vertical elements of the model with the pointer. The elements will be coloured red.

- To present on the screen only selected nodes and elements of the model, on the **Select** toolbar, click **Fragmentation**  .
- On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button  .
- Specify with the pointer nodes No. 1, 4, 7, 10, 13, 28, 31, 34, 37, 55, 58, 61, 64, 79, 82, 85, 88, 91, 106, 109, 112, 115, 133, 136, 139, 142, 157, 160, 163, 166 и 169.

- 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.22.8).
- In this dialog box, specify with the pointer **FE '57' (pile)** option.
- Click **Apply**  .

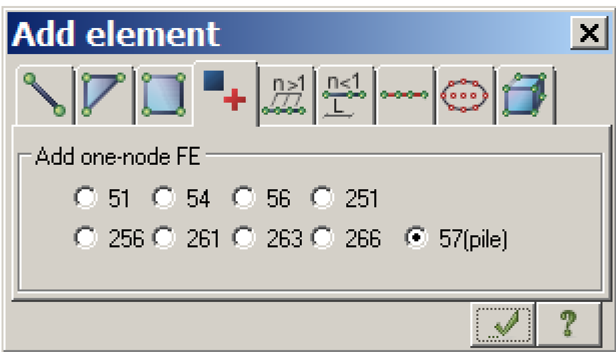


Figure 22.8 **Add element** dialog box

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

<u>To hide numbers of nodes on design model:</u>

- 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**  .

<u>To define stiffness parameters for piles:</u>

- On the **Create and edit** ribbon tab, on the **Stiffness and restraint** panel, click **Copy Pile stiffness**  .
- In the **Define stiffness parameters for piles** dialog box (see Fig.22.9), click **Input data for calculation**.

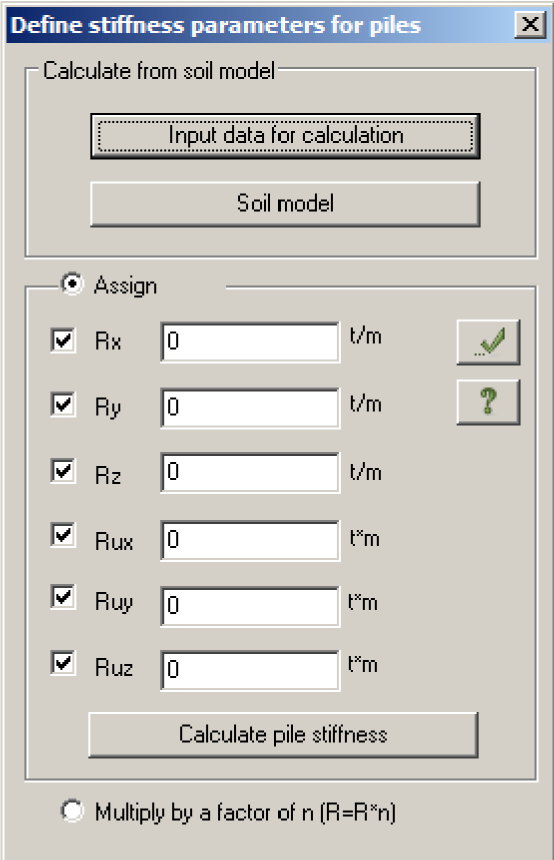
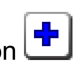


Figure 22.9 **Define stiffness parameters for piles** dialog box

- In the **Pile groups** dialog box (see Fig.22.10), click the **Pile of circular cross-section** button and define the following parameters for piles:
  - pile length – L = 15 m;
  - outside diameter of pile – D = 80 cm;
  - partial safety factor – γ_c = 1;
  - coefficient for depth below the pile toe – k = 0.5;
  - modulus of elasticity for pile material – Ec = 3e6 t/m^2 (for the U.S. keyboard layout);
  - number of division intervals – Lv = 20.
- To input defined data to the table in the left part of the dialog box, click the **Add new row** button .

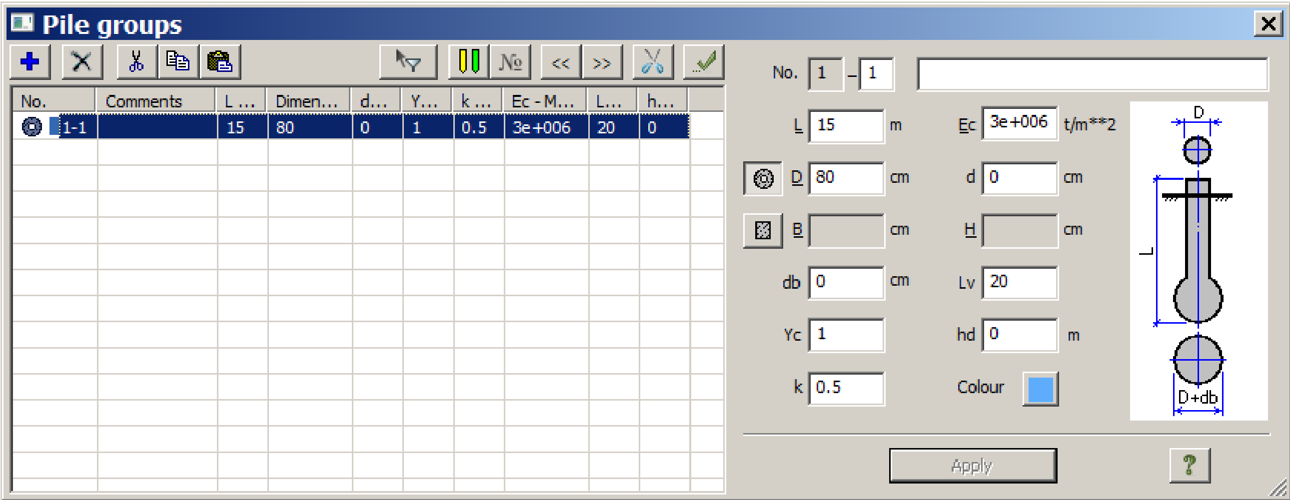


Figure 22.10 **Pile groups** dialog box

To assign stiffness parameters to piles:

- To select elements of piles, 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.22.11), select **By FE type** check box and select **FE type 57(pile)** option in the list.
- In the **PolyFilter** dialog box, click **Apply**.

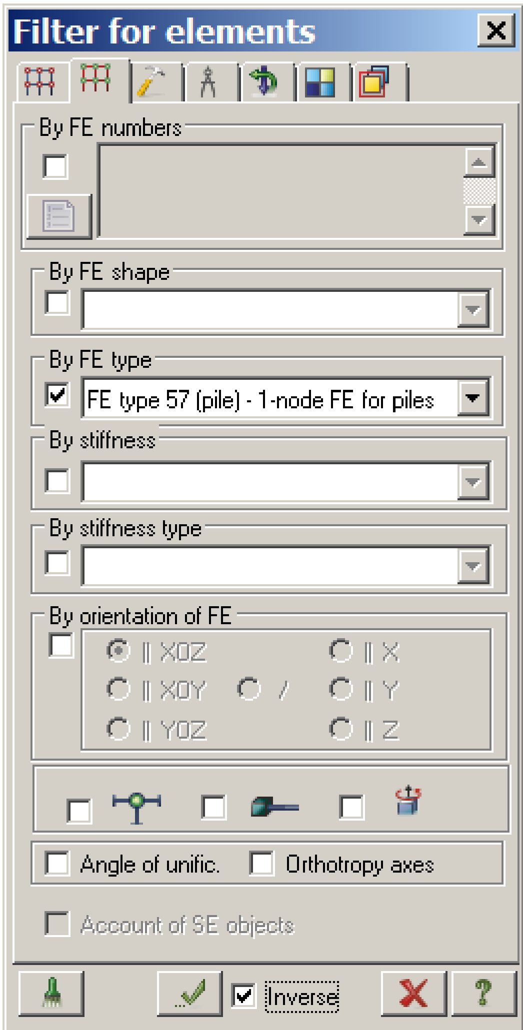


Figure 22.11 **Filter for elements** dialog box

- In the **Pile groups** dialog box, click the **Apply highlighted parameters to the piles selected on the model** button  .
- Then close the dialog box.

<u>To activate SOIL system:</u>

- To activate the SOIL system, in the **Define stiffness parameters for piles** dialog box (see Fig.22.9), click **Soil model** button.
- In the **Soil model** dialog box (see Fig.22.12), click **Position** tab and define the following data:
  - location on plan for point 1 on design model – X = -1 m;
  - location on plan for point 1 on design model – Y = -1 m;
  - location on plan for point 2 on design model – X = 0 m;
  - location on plan for point 2 on design model – Y = 0 m;
  - location in height for min Z-coordinate of design model – Z = 0 m;
  - location on plan for point 1 on soil model – X = 0 m;
  - location on plan for point 1 on soil model – Y = 0 m;
  - location on plan for point 2 on soil model – X = 1 m;

- location on plan for point 2 on soil model – Y = 1 m;
- location in height for appropriate Z-coordinate of soil model – Z = 96 m.

➪ Then in the **Soil model** dialog box, click **Attach soil model** button.

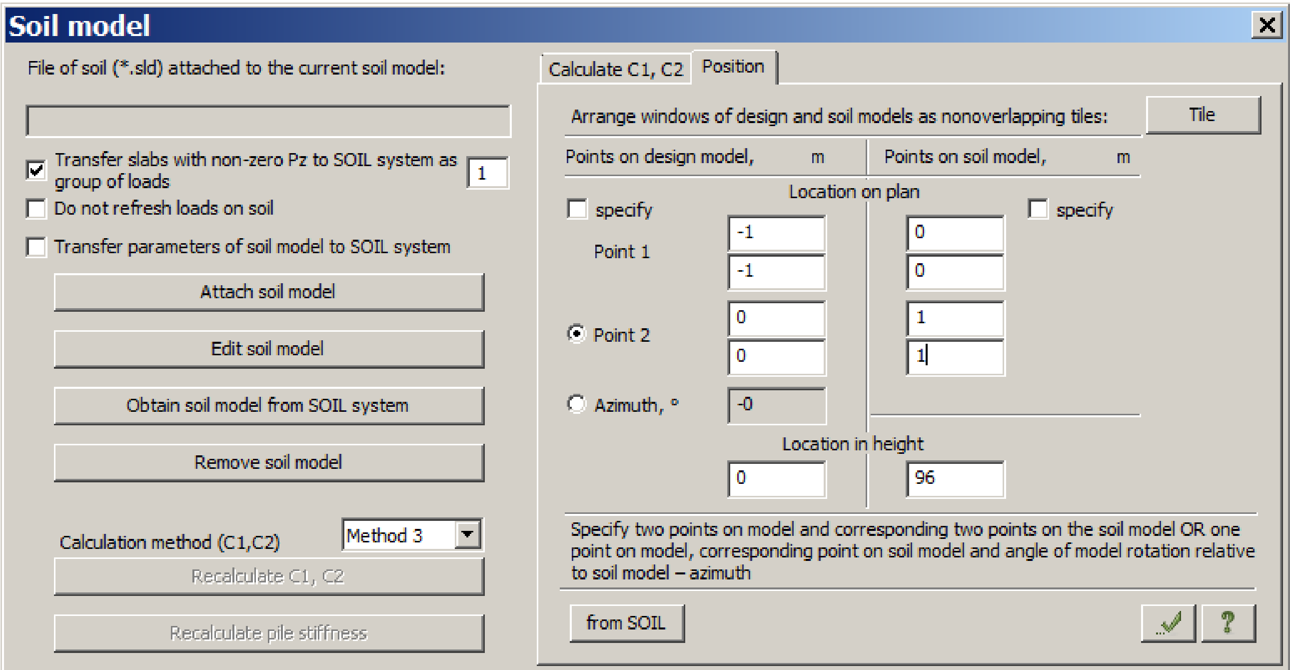


Figure 22.12 **Soil model** dialog box

*For calculation of pile stiffness, we will use the soil model from **Example 9** where the generation of soil model is described in detail.*

➪ In the **Open file with soil model** dialog box (see Fig.22.13), select **Example9** row and click **Open**.

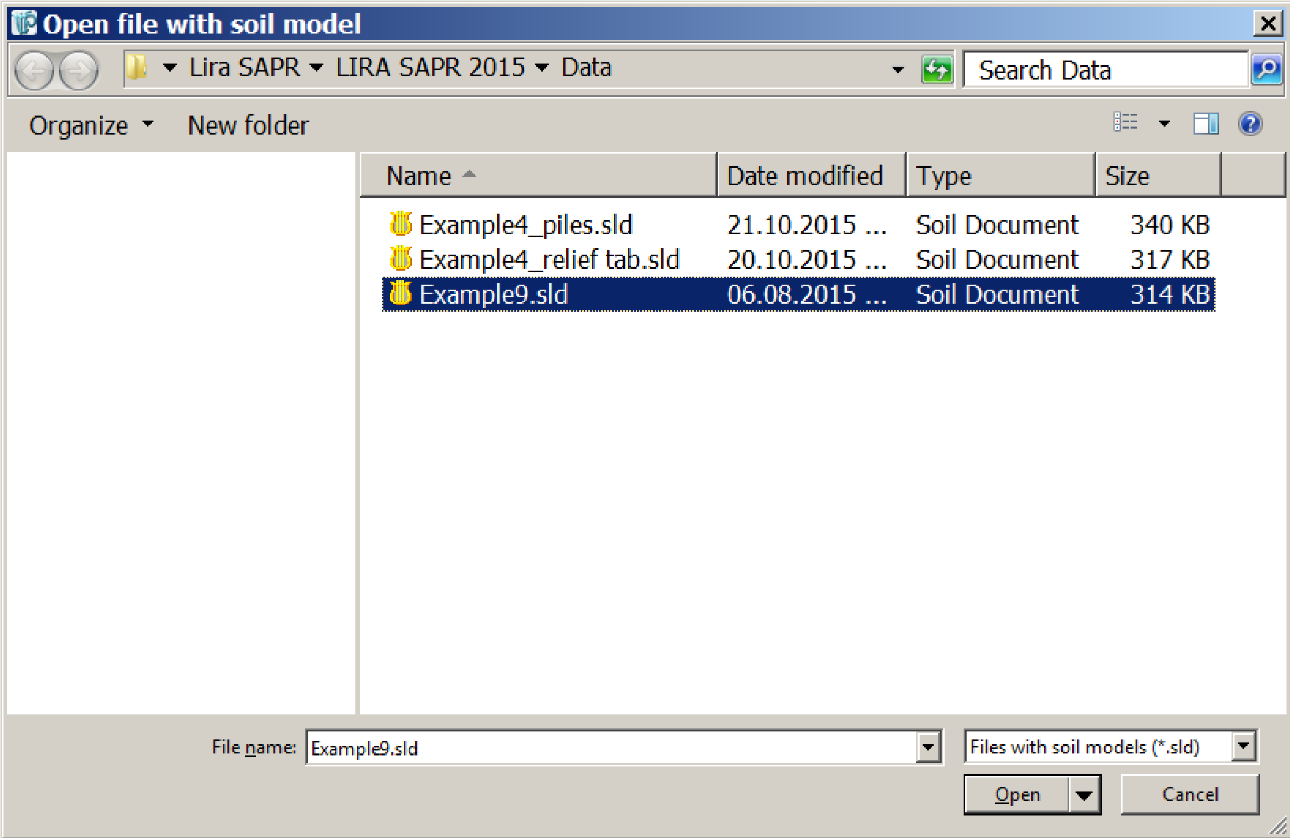


Figure 22.13 **Open file with soil model** dialog box

Step 5. Applying loads

- Return to LIRA-SAPR program to the mode of design model generation. To do this, just click appropriate taskbar button.

To create load case No.1:

- To define load from dead weight, 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.22.14), click **All elements** and specify **Load factor** as equal to **1**. Then click **Apply** (dead weight is automatically applied to elements).

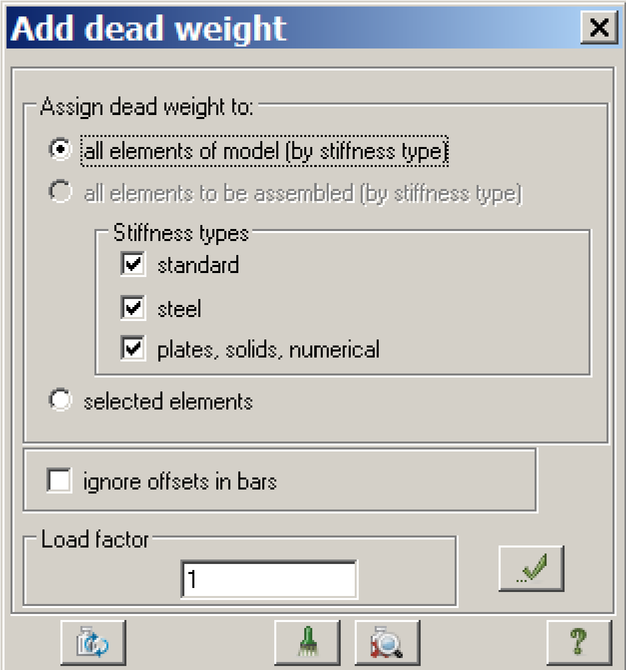


Figure 22.14 **Add dead weight** dialog box

<u>To create load case No.2:</u>

- To change the number of the current load case, click the **Next load case** button  located on the Status bar.
- On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **Select block**  .
- Specify with the pointer any node or element of the floor slab.
- 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.22.15), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

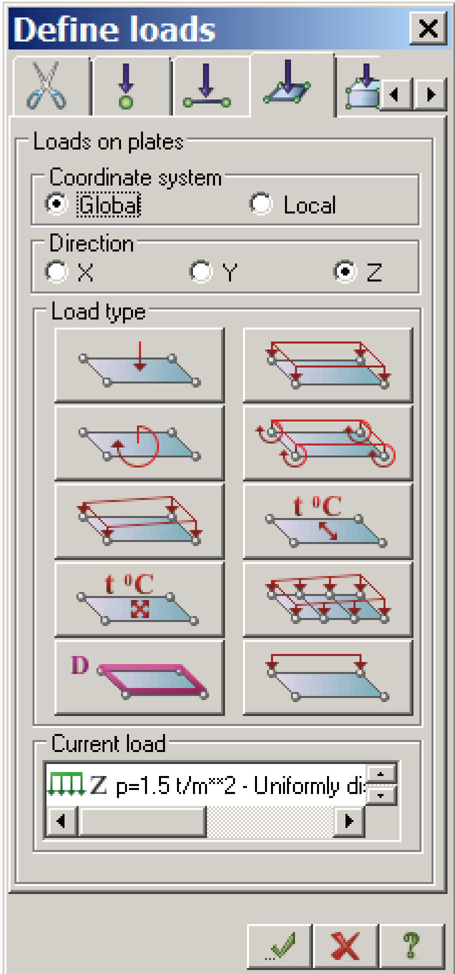
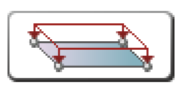


Figure 22.15 **Define loads** dialog box

- In the **Load type** area, click **Uniformly distributed load** button  .
- In the **Load parameters** dialog box specify P = 1.5 t/m^2 (see Fig.22.16).
- Click **OK**  .

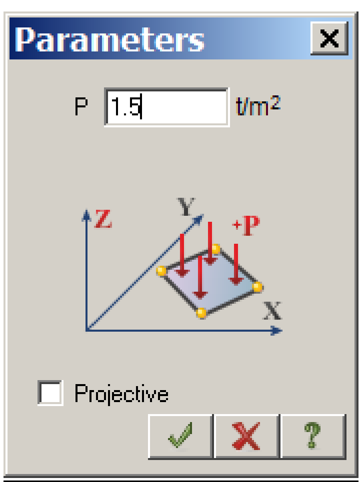


Figure 22.16 **Load parameters** dialog box

- The **Warning** message box is displayed (see Fig.22.17). Click **OK**.

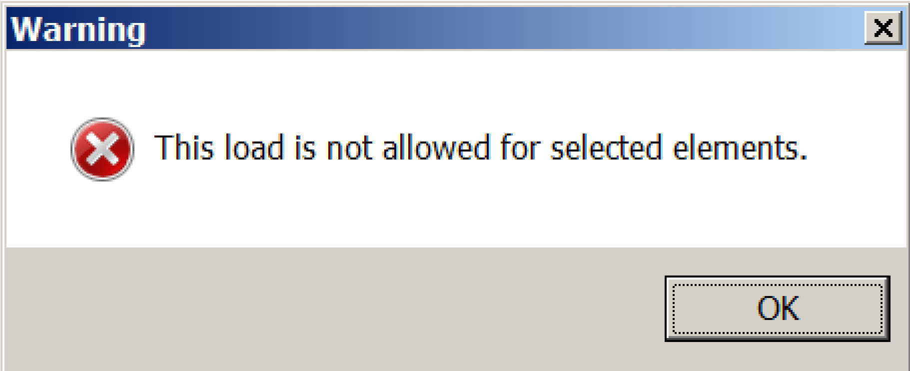


Figure 22.17 **Warning** message box

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

- To unselect nodes and elements, on the **Select** toolbar, click **Unselect all** button .
- When the **Select block** command is active, specify with the pointer any node or element of the grillage.
- 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 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 elements of the floor slab once again.
- 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 = 0.35 t/m^2.
- Click **OK** .
- In the **Define loads** dialog box, click **Apply** .
- To unselect nodes and elements, on the **Select** toolbar, click **Unselect all** button .

Step 6. Static 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** .
- In the **Warning** box (see Fig.22.18), select the **recalculate pile stiffness (FE 57) by soil model** check box and click **OK**.

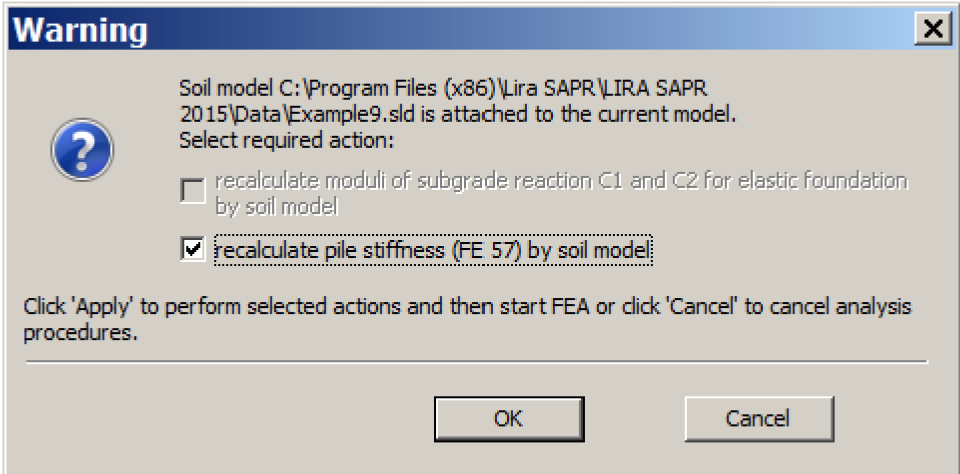


Figure 22.18. **Warning** box

Step 7. Review and evaluation of static analysis results

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** .

To present mosaic plots of stiffness parameters for piles:

- To present mosaic plot of stiffness parameters for piles Rz, on the **Create and edit** ribbon tab, on the **Tools** panel, select the **Mosaic plot of stiffness parameters for piles Rz** command in the **Mosaic plot of stiffness parameters for piles** drop-down list .
- To present mosaic plot of stiffness parameters for piles Rx, on the **Create and edit** ribbon tab, on the **Tools** panel, select the **Mosaic plot of stiffness parameters for piles Rx** command in the **Mosaic plot of stiffness parameters for piles** drop-down list .
- To display information about stiffness parameters for a certain pile, on the **Select** toolbar, click the **Information about nodes and elements** button and specify with the pointer any 1-node element (e.g. below the left column).
- In the **Node or 1-node FE?** dialog box (see Fig.22.19), select the **Element** option and click **OK** .

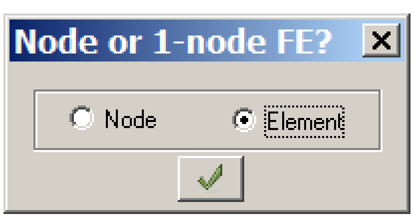


Figure 22.19 **Node or 1-node FE?** dialog box

- In the **Select element** dialog box (see Fig.22.20), select the row that corresponds to 1-node element and click **OK** .

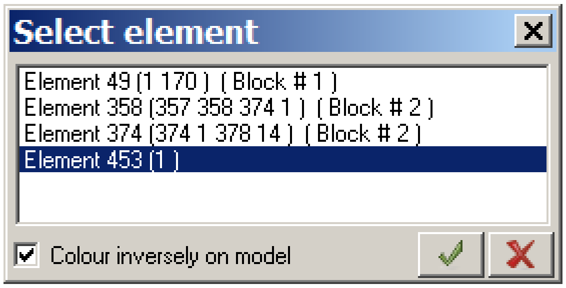


Figure 22.20 **Select element** dialog box

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

- In the mode of analysis results visualization, by default design model is presented with account of nodal displacements.

<u>To present displacement contour plots:</u>

- 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.

<u>To present diagrams of internal forces:</u>

- To display diagram $\mathbf{M_y}$, on the **Results** tab, select **Forces in bars** panel and click **Moment diagrams ($\mathbf{M_y}$)** button .
- To display diagram $\mathbf{Q_z}$, on the **Results** tab, select **Forces in bars** panel and click **Shear force diagrams ($\mathbf{Q_z}$)** 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.

<u>To change number of active load case:</u>

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

<u>To present stress mosaic plots:</u>

- 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 forces in piles:

- To present mosaic plots of forces Rz in 1-node elements, on the **Results** ribbon tab, on the **Forces in 1-node FE** panel, click **Rz (FE 51, 56, 57, 256, 266)** .
- To present mosaic plots of forces Rx in 1-node elements, on the **Results** ribbon tab, on the **Forces in 1-node FE** panel, click **Rx (FE 51, 56, 57, 256, 266)** .
- To display information about the forces in a certain 1-node FE, on the **Select** toolbar, click the **Information about nodes and elements** button (in the similar way as for stiffness parameters).

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.22.21), select **Forces** in the list.
- Make sure that **All load cases** option is defined in the **Choose load case No.** list and click **Apply** .

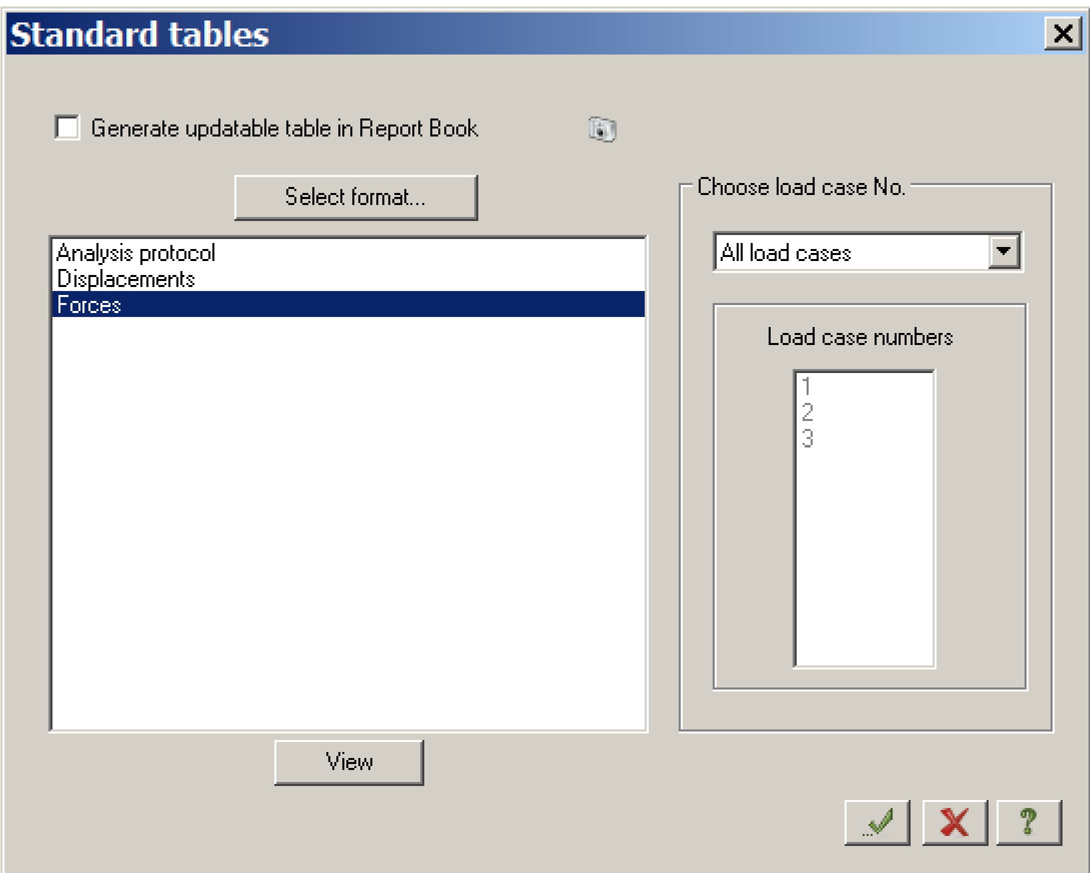


Figure 22.21 **Standard tables** dialog box

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

