

Example 10. Analysis of sheet piling (strengthened with anchors) together with soil of foundation pit [with nonlinear elements of soil, simulation of anchor pretension, simulation of excavation of a pit]

In this lesson you will learn how to:

- simulate multi-layer foundation using Coulomb-Mohr criterion;
- generate design model of the sheet piling of the foundation pit during assemblage;
- perform nonlinear analysis of the system 'nonlinear foundation - linear structures of sheet piling' with account of assemblage process and excavation of a pit;
- perform analysis with account of modulus of elasticity of soil along the unloading path (account of unloading of soil model).

Description:

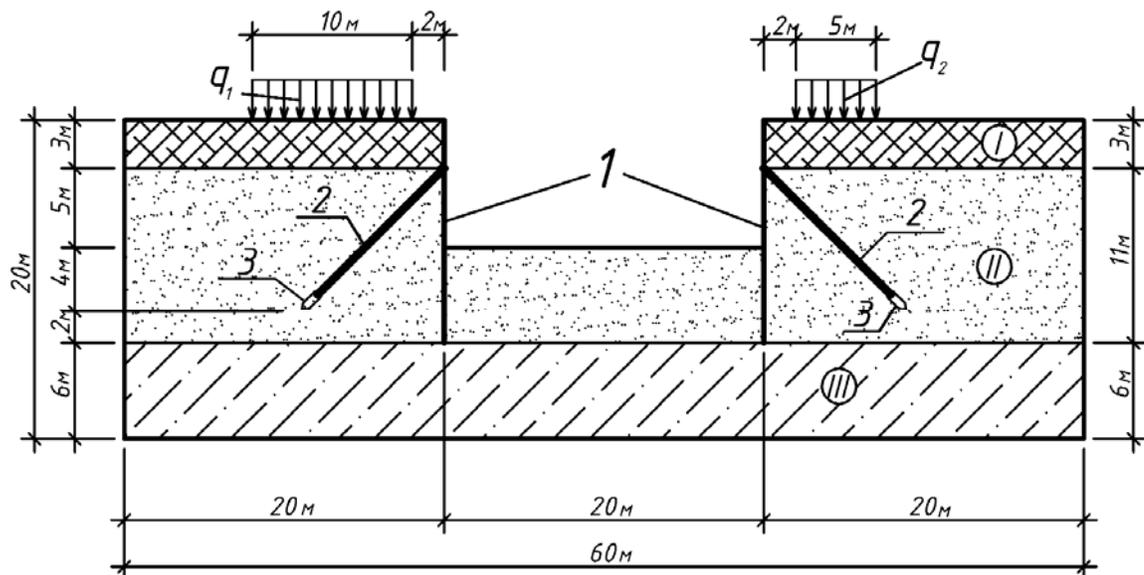
Three-layer foundation with dimensions 60x20m, thickness 1.0m (see Figure 10.1).

Pit of the following size: 20x8m (see Figure 10.1).

Vertical sheet piling with height 14m (see Figure 10.1).

Supports for anchors of length 3m, anchors of length 10m located at the angle 45 (see Figure 10.1).

Analysis of foundation is performed with FE of dimensions 1.0x1.0m.



- 1 - sheet piling
- 2 - anchors
- 3 - supports for anchors
- I - fill-up soil
- II - sand
- III - clay loam

Figure 10.1 Model of sheet piling of a pit and loads on multi-layer foundation

Loads:

load case 1 – dead weight of soil of three-layer foundation;

load case 2

- dead uniformly distributed load $g_1 = 1.0$ t/m and dead uniformly distributed load $g_2 = 0.5$ t/m applied to the surface of foundation (see Figure 10.1),
- dead weight of sheet piling;

load case 4 – pretension of anchors $F = 5t$.

- ⇒ 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.10.2) specify the following data:
 - problem name – **Example10**;
 - model type – **2 – Three degrees of freedom per node** (translations X, Z and rotation UY) X0Z.
- ⇒ Click **OK** .

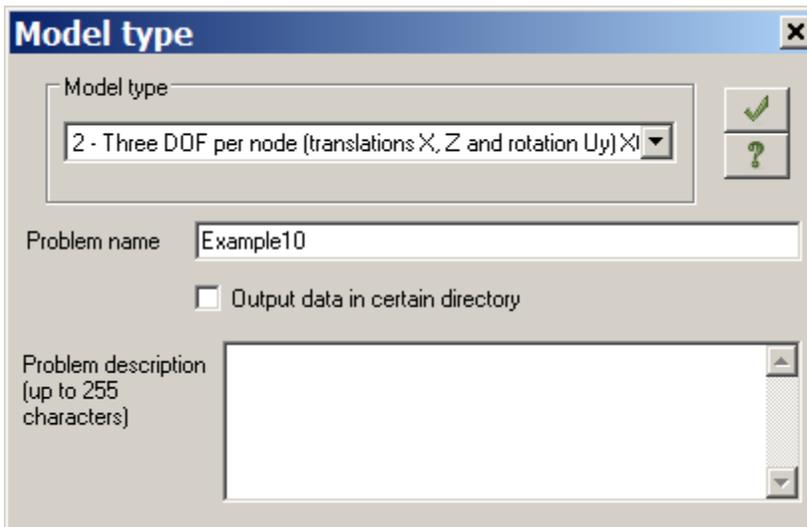


Figure 10.2 **Model type** dialog box



*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 2 (Three 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 2 (Three 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** drop-down list and click the **Create wall-beam**  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
1 60	1 20

- other parameters remain by default (see Fig.10.3).

⇒ Click **Apply**  .

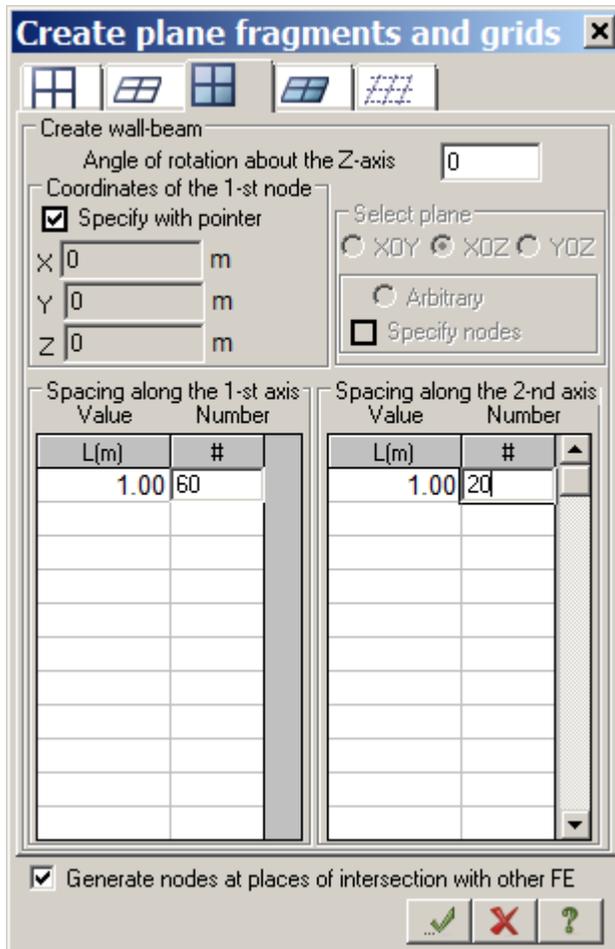
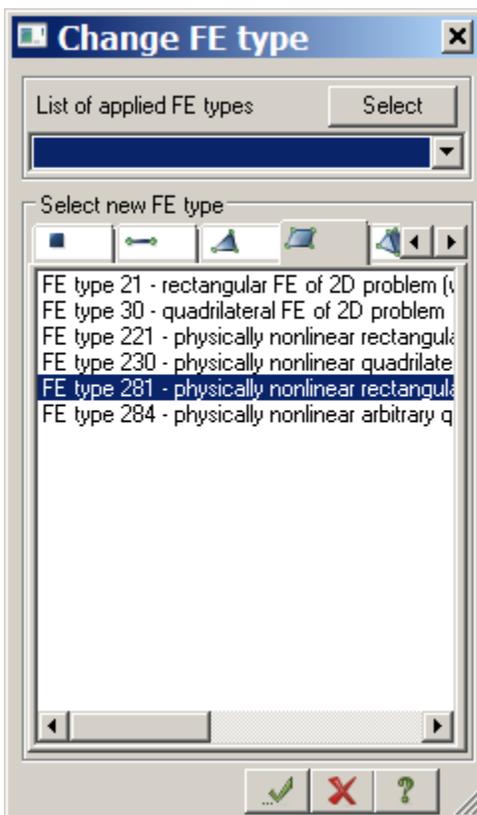


Figure 10.3 **Create plane fragments and grids** dialog box

To change FE type for elements of foundation:

- ⇒ 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 (nodes and elements will be coloured red).
- ⇒ On the **Advanced edit options** ribbon tab, on the **Model** panel, click **Change FE type**  .
- ⇒ In the **Change FE type** dialog box (see Fig.10.4), in the list of FE types, select **FE type 281 - physically nonlinear rectangular FE of 2D problem (soil)**.
- ⇒ Click **Apply**  .

Figure 10.4 **Change FE type** dialog box

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

To present numbers of nodes on the screen:

- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **Flags of drawing** button .
- ⇒ In the **Display** dialog box (see Fig.10.5), select the **Node numbers** check box on the **Nodes** tab.
- ⇒ Click **Redraw** .

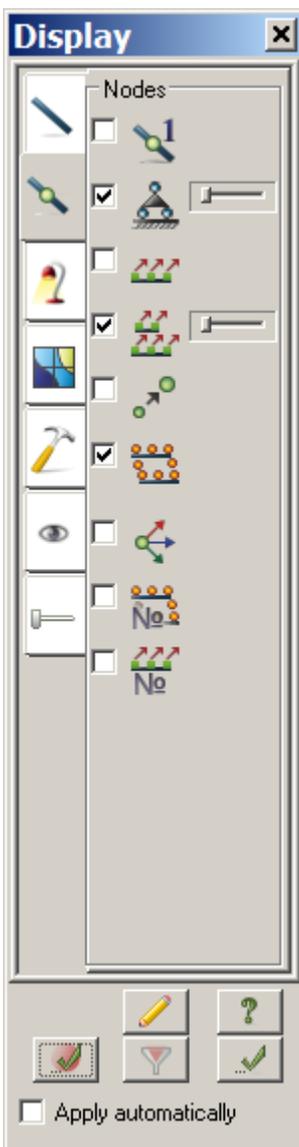


Figure 10.5 **Display** dialog box

To add sheet piling of a pit, anchors and supports for anchors:



To reduce or enlarge the model, rotate the mouse wheel.

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click **Add bar** .
- ⇒ The **Add element** dialog box is presented with the **Add bar** tab open (see Fig.10.6).
- ⇒ In the **Add element** dialog box, clear the **Generate nodes at places of intersection with other FE** check box.

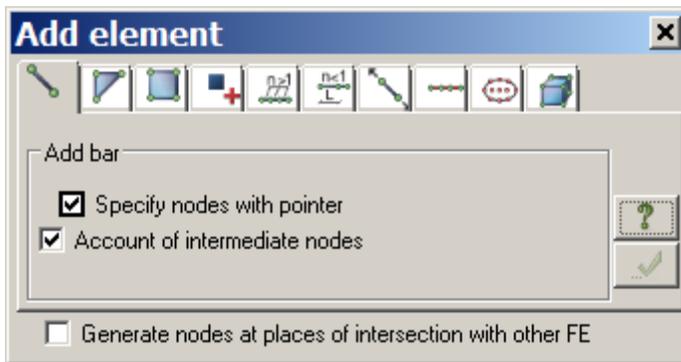


Figure 10.6 Add element dialog box

- ⇒ To add sheet piling of a pit between nodes No.387 and 1241, 407 and 1261 (along the vertical), make sure that **Specify nodes with pointer** and **Account for intermediate nodes** check boxes are selected and then specify with the pointer these pairs of nodes in sequence (in this case the rubber-band line is automatically stretched between the nodes that you select).
- ⇒ To add supports for anchors between nodes No.500 and 624, 538 and 658 (along oblique line), specify with the pointer these pairs of nodes in sequence.
- ⇒ To add anchors between nodes No.624 and 1058, 658 and 1078 (along oblique line), in the **Add element** dialog box, clear the **Account for intermediate nodes** check box and then specify with the pointer these pairs of nodes in sequence.

To change FE type for anchors and supports for anchors:

- ⇒ On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** command .
- ⇒ On the **Select** toolbar, click **Select by contour** command .
- ⇒ With left mouse button specify closed contour around oblique elements of anchor supports beginning from the left side of sheet piling of a pit between nodes No.500 and 624, and then click **Select by contour** (button  on the toolbar) command once more - from the right side of a pit between nodes No.538 and 658. You could also specify elements on the model with the pointer.
- ⇒ On the **Advanced edit options** ribbon tab, on the **Model** panel, click **Change FE type** .
- ⇒ In the **Change FE type** dialog box, in the list of FE types, select **FE type 1 - FE of 2D truss**.
- ⇒ Click **Apply** .
- ⇒ On the design model, select elements of anchors located between nodes No.624 and 1058, 658 and 1078 with the **Select by contour** command.
- ⇒ In the **Change FE type** dialog box, in the list of FE types, select **FE type 208 - physically nonlinear special 2-node FE of pretension**.
- ⇒ Click **Apply** .
- ⇒ On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button  once again in order to make this command not active.

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 – **Example10**;
 - location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

Step 3. Defining boundary conditions

- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button  .
- ⇒ Specify with the pointer nodes of the lower edge of foundation - nodes No.1-61 (they will be coloured red).

To define boundary conditions at nodes of lower edge of foundation:

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Restraints** button  .
- ⇒ In the **Restraints on nodes** dialog box (see Fig.10.7) specify directions along which displacements of nodes are not allowed (X, Z). To do this, select appropriate check boxes.
- ⇒ Click **Apply**  (the nodes will be coloured blue).

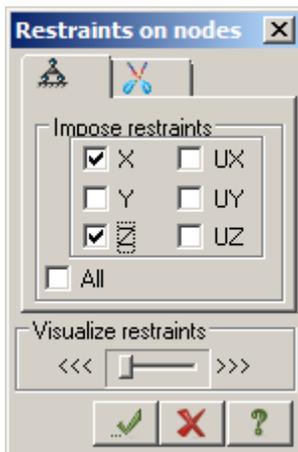


Figure 10.7 **Restraints on nodes** dialog box

To define boundary conditions at nodes of side edge of foundation:

- ⇒ Specify with the pointer nodes of the farthest to the left and farthest to the right side edges of foundation.
- ⇒ In the **Restraints on nodes** dialog box, specify directions along which displacements of nodes are not allowed (X). To do this, clear the Z check box.
- ⇒ Click **Apply**  .

- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button  once again in order to make this command not active.

Step 4. Defining material properties to elements of design model

To create material data sets:

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Material properties** button  .

- ⇒ In the **Stiffness and materials** dialog box (see Fig.10.8a), click **Add**. The dialog box expands to display the library of stiffness parameters.
- ⇒ In the **Add stiffness** dialog box (see Fig.10.8b), select the **Plates, solids, numerical** tab (the third tab) and double-click the **Numerical for FE 281-284** icon in the list.

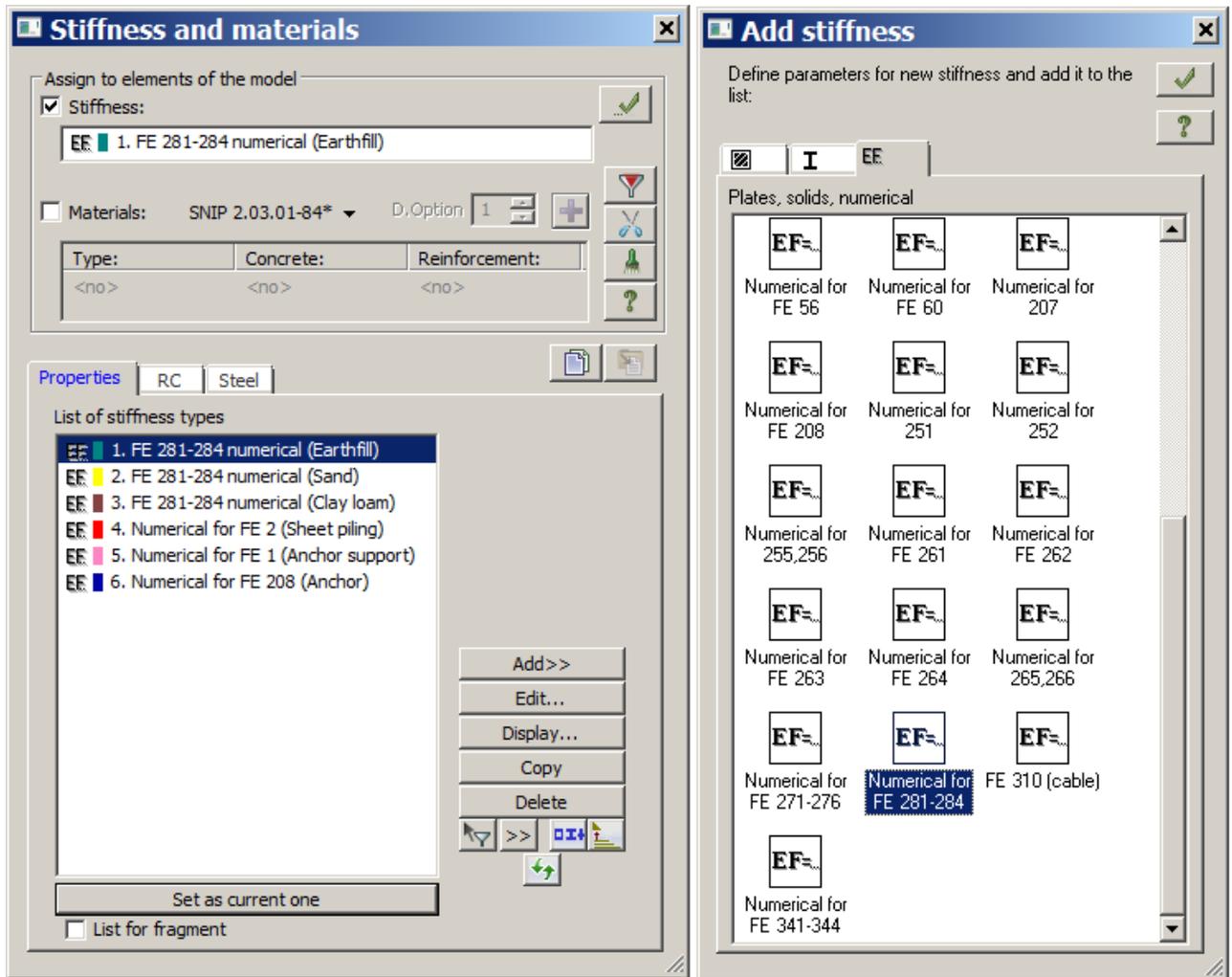


Figure 10.8 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ In the **Numerical description for FE 281-284** dialog box, specify the following parameters for the first layer of soil (earthfill) (see Fig.10.9):
 - modulus of elasticity for soil along the loading path – $E = 800 \text{ t/m}^2$;
 - Poisson's ratio – $\nu = 0.3$;
 - thickness – $H = 100 \text{ cm}$;
 - unit weight of material – $R_o = 1.6 \text{ t/m}^3$;
 - cohesion - $C = 0.1 \text{ t/m}^2$;
 - ultimate stress in tension - $R_t = 0.01 \text{ t/m}^2$;
 - angle of internal friction - $F_i = 30 \text{ degrees}$;
 - conversion coefficient to modulus of elasticity for soil along the unloading path - $K_e = 3$;
 - in the **Comments** box, type **Earthfill** and select colour for this stiffness (green).

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

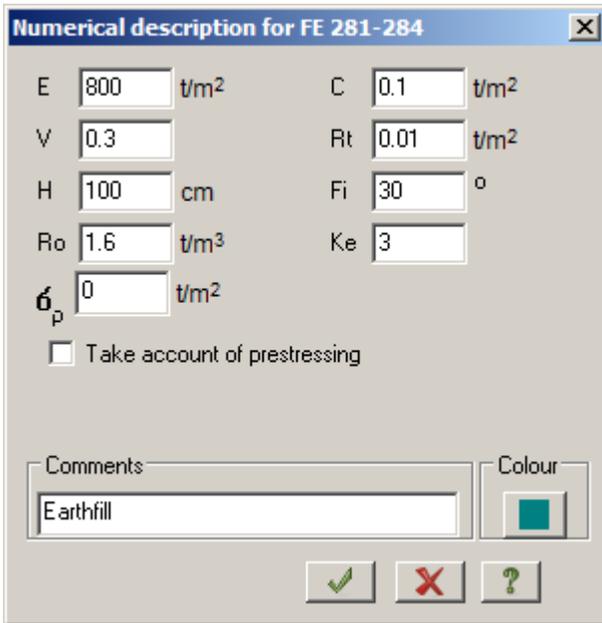


Figure 10.9 Numerical description for FE 281-284 dialog box

⇒ In the **Stiffness of elements** dialog box, in the **List of stiffness types**, select '1. FE 281-284 numerical' and click **Copy** two times.

⇒ In the **List of stiffness types**, select '2. FE 281-284 numerical' and click **Edit**.

⇒ In the **Numerical description for FE 281-284** dialog box, specify properties of the second layer of soil (sand):

- modulus of elasticity of soil along the loading path – $E = 3000 \text{ t/m}^2$;
- Poisson's ratio – $V = 0.3$;
- thickness – $H = 100 \text{ cm}$;
- unit weight of material – $R_o = 1.7 \text{ t/m}^3$;
- cohesion $C = 0.1 \text{ t/m}^2$;
- ultimate stress in tension $R_t = 0.01 \text{ t/m}^2$;
- angle of internal friction $F_i = 34 \text{ degrees}$;
- conversion coefficient to modulus of elasticity for soil along the unloading path $K_e = 3$;
- in the **Comments** box, type **Sand** and select the colour for this stiffness (yellow).

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

⇒ In the **Stiffness of elements** dialog box, in the **List of stiffness types**, select '3. FE 281-284 numerical' and click **Edit**.

⇒ In the **Numerical description for FE 281-284** dialog box, specify properties of the third layer of soil (clay loam):

- modulus of elasticity of soil along the loading path – $E = 2000 \text{ t/m}^2$;
- Poisson's ratio – $V = 0.33$;
- thickness – $H = 100 \text{ cm}$;

- unit weight of material – $R_o = 1.7 \text{ t/m}^3$;
- cohesion $C = 0.8 \text{ t/m}^2$;
- ultimate stress in tension $R_t = 0.08 \text{ t/m}^2$;
- angle of internal friction $F_i = 29$ degrees;
- conversion coefficient to modulus of elasticity for soil along the unloading path $K_e = 3$;
- in the **Comments** field, type **Clay loam** and select the colour for this stiffness (brown).

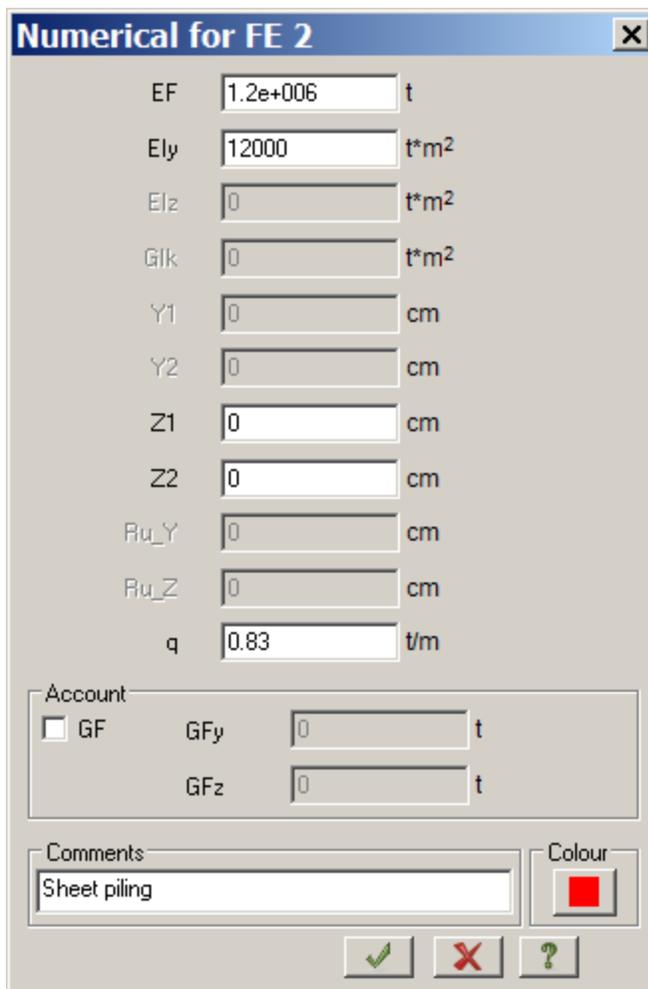
⇒ To confirm the specified data, click **OK** .

⇒ In the **Stiffness of elements** dialog box, double-click **Numerical for FE 2** icon.

⇒ In the **Numerical description for FE 2** dialog box (see Figure 10.10), specify parameters of sheet piling section:

- stiffness of the element in axial compression (tension) – $E_F = 1.2 \times 10^6 \text{ t}$ (for the U.S. keyboard layout);
- stiffness of the element in bending about the Y1-axis – $E_{ly} = 12000 \text{ t} \cdot \text{m}^2$;
- weight per unit length $q = 0.83 \text{ t/m}$;
- in the **Comments** field, type **Sheet piling** and select the colour for this stiffness (red).

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



Numerical for FE 2

EF t

Ely t*m²

Elz t*m²

Gik t*m²

Y1 cm

Y2 cm

Z1 cm

Z2 cm

Ru_Y cm

Ru_Z cm

q t/m

Account

GF GFy t

GFz t

Comments

Sheet piling

Colour 

Figure 10.10 **Numerical description for FE 2** dialog box

- ⇒ In the **Stiffness of elements** dialog box, double-click **Numerical for FE 1** icon.
- ⇒ In the **Numerical description for FE 1** dialog box (see Figure 10.11), specify parameters of supports for anchors:
 - stiffness of the element in axial compression (tension) – $EF = 10000$ t;
 - in the **Comments** field, type **Support for anchor** and select the colour for this stiffness (pink).
- ⇒ To confirm the specified data, click **OK** .

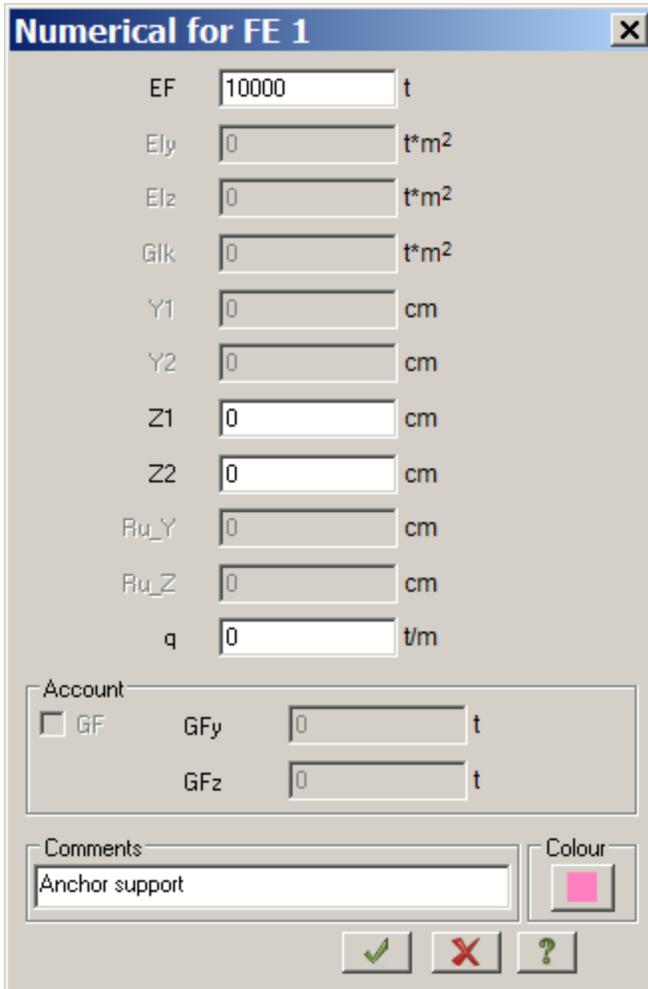


Figure 10.11 **Numerical description for FE 1** dialog box

- ⇒ In the **Stiffness of elements** dialog box, double-click **Numerical for FE 208** icon.
- ⇒ In the **Numerical description for FE 208** dialog box (see Figure 10.12), under **Method of description**, click **Numerical** and specify parameters of anchors:
 - stiffness of the element in tension – $EF = 8000$ t;
 - maximum tensile force – $N_{max} = 1e9$ t (for the U.S. keyboard layout);
 - in the **Comments** field, type **Anchor** and select the colour for this stiffness (blue).
- ⇒ To confirm the specified data, click **OK** .

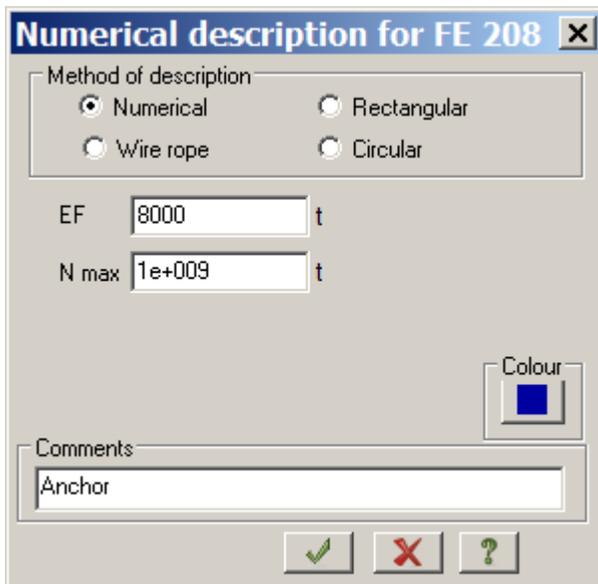


Figure 10.12 Numerical description for FE 208 dialog box

⇒ To hide library of stiffness properties, in the **Stiffness of elements** dialog box, click **Add** unfold button.

To assign material properties to elements of design model:

- ⇒ In the **Stiffness of elements** dialog box, under **List of stiffness types**, click the stiffness type '1.FE 281-284 numerical'.
- ⇒ Click **Set as current type**. Selected type will be displayed in the **Stiffness** box under **Assign to elements of the model**. You can also specify the current type by double-clicking the necessary type in the **List of stiffness types**.
- ⇒ On the SELECT menu, click **Select elements** (button  on the toolbar).
- ⇒ Select with the pointer three upper rows of finite elements of foundation (thickness of layer 3m).



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

- ⇒ In the **Stiffness of elements** dialog box, click **Apply** .
- ⇒ The **Warning** box (see Figure 10.13) is displayed. Click **OK**. (This message is displayed because bar elements were selected on design model as well as plate elements. But this type of stiffness is not allowed for bar elements.)



Figure 10.13 Warning message box

- ⇒ In the **Stiffness of elements** dialog box, under **List of stiffness types**, click the stiffness type '2.FE 281-284 numerical'.
- ⇒ Click **Set as current type**.
- ⇒ With the pointer select the range from the 4th up to 14th inclusive (from the top of anchors up to bottom of sheet piling elements) rows of finite elements of foundation (thickness of layer 11m).
- ⇒ In the **Stiffness of elements** dialog box, click **Apply** .
- ⇒ The **Warning** box is displayed. Click **OK**.
- ⇒ To unselect elements, on the SELECT menu, click **Unselect all** (button  on the toolbar).

- ⇒ In the **Stiffness of elements** dialog box, under **List of stiffness types**, click the stiffness type '3.FE 281-284 numerical'.
- ⇒ Click **Set as current type**.
- ⇒ With the pointer select the six rows of finite elements of foundation that remain (thickness of layer 6m).
- ⇒ In the **Stiffness of elements** dialog box, click **Apply** .

- ⇒ In the **Stiffness of elements** dialog box, set the stiffness type '4.FE 2 numerical' as current.
- ⇒ Click **Set as current type**.
- ⇒ On the **Select** menu, click **Select vertical elements** (button  on the toolbar).
- ⇒ Select all vertical elements of the model with the pointer.
- ⇒ In the **Stiffness of elements** dialog box, click **Apply** .

- ⇒ In the **Stiffness of elements** dialog box, set the stiffness type '6.FE 208 numerical' as current.
- ⇒ To select elements of anchors, on the SELECT menu, click **PolyFilter** (button  on the toolbar).
- ⇒ In the **PolyFilter** dialog box (see Figure 10.14), click the **Filter for elements** tab.
- ⇒ Select **By FE type** check box and specify **FE type 208 - physically nonlinear special 2-node FE for simulation of pretension**.
- ⇒ Click **Apply** .

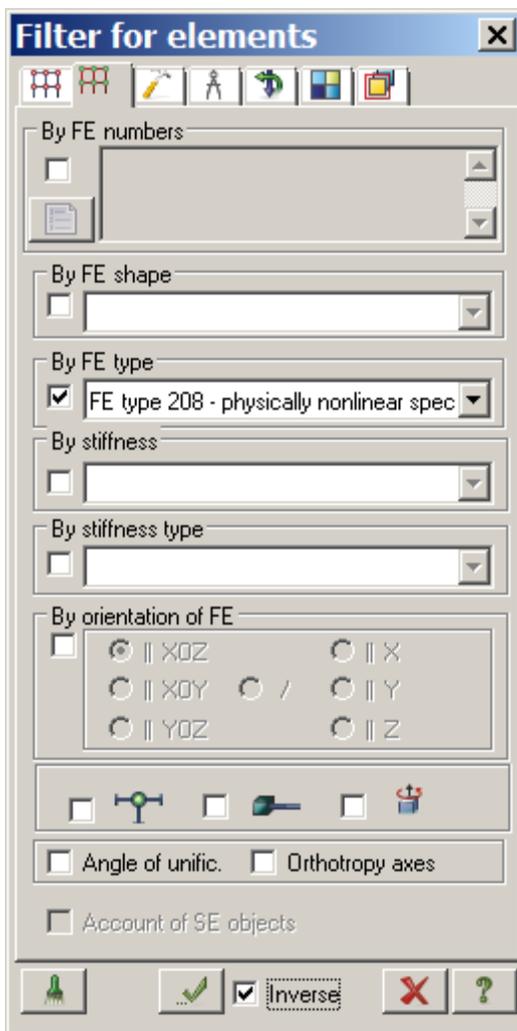


Figure 10.14 PolyFilter dialog box

- ⇒ In the **Stiffness of elements** dialog box, click **Apply** .
- ⇒ In the **Stiffness of elements** dialog box, set the stiffness type '5.FE 1 numerical' as current.
- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, select **By FE type** check box and specify **FE type 1 - FE of 2D truss**.
- ⇒ Click **Apply** .
- ⇒ In the **Stiffness of elements** dialog box, click **Apply** .

Step 5. Applying loads

To create load case No.1:

- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, select **By FE type** check box and specify **FE type 281 - physically nonlinear rectangular FE of 2D problem (soil)**.
- ⇒ Click **Apply** .
- ⇒ To apply load from dead weight of the elements of foundation, on the LOADS menu, click **Add dead weight**.

- ⇒ In the **Add dead weight** dialog box (see Figure 10.15), click **Selected elements** and click **Apply**  (dead weight of elements is added according to the specified unit weight Ro).

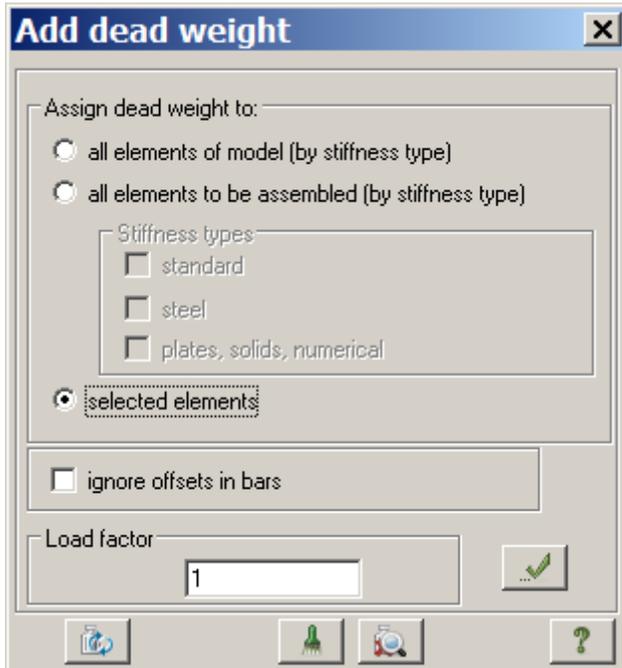


Figure 10.15 **Add dead weight** dialog box

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.
- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, select **By FE type** check box and specify **FE type 10 - arbitrary 3D bar**.
- ⇒ Click **Apply** .
- ⇒ In the **Add dead weight** dialog box, click **Selected elements** and click **Apply** .
- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Specify with the pointer nodes of the upper edge of foundation - nodes No.1230-1238.
- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel, then select **Load at nodes** command  from the **Loads on nodes and elements** drop-down list.
- ⇒ In the **Define loads** dialog box (see Fig.10.16), specify **Global** coordinate system and direction along the **Z-axis** (default parameters).

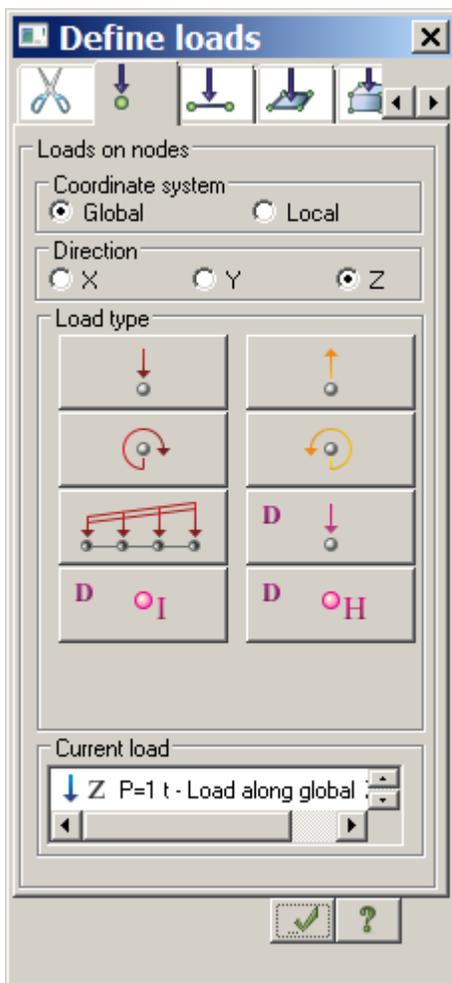


Figure 10.16 Define loads dialog box

- ⇒ In the **Load type** area, click the **Concentrated load** button .
- ⇒ In the **Load parameters** dialog box (see Fig.10.17), specify $P = 1$ t.
- ⇒ Click **OK**.

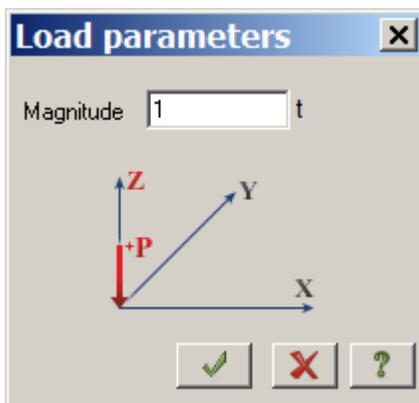


Figure 10.17 Load parameters dialog box

- ⇒ Specify with the pointer nodes of the upper edge of foundation - nodes No.1229 and 1238.

- ⇒ In the **Define loads** dialog box, in the **Load type** area, click the **Concentrated load** button



- ⇒ In the **Load parameters** dialog box specify $P = 0.5$ t.
- ⇒ Click **OK** .

- ⇒ Specify with the pointer nodes of the upper edge of foundation - nodes No.1264-1267.

- ⇒ In the **Define loads** dialog box, click **Apply** .

- ⇒ Specify with the pointer nodes of the upper edge of foundation - nodes No.1263 and 1268.

- ⇒ In the **Define loads** dialog box, in the **Load type** area, click the **Concentrated load** button



- ⇒ In the **Load parameters** dialog box specify $P = 0.25$ t.
- ⇒ Click **OK** .

To create load case No.4:

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



In this problem there will be five stages of assemblage and five nonlinear load cases. To do this, it is necessary to remain the load case No.3 as 'empty', in load case No.4 specify pretension of anchors and in load case No.5 specify fictitious load in one of the extreme nodes of the foundation along boundary conditions of this node.

- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, select **By FE type** check box and specify **FE type 208 - physically nonlinear special 2-node FE for simulation of pretension**.
- ⇒ Click **Apply** .
- ⇒ In the **Define loads** dialog box, click the **Loads on bars** tab.
- ⇒ Click the **Load on special element (turnbuckle)** button  and in the **Load parameters** dialog box specify $P = 5$ t (see Figure 10.18).
- ⇒ Click **OK** .

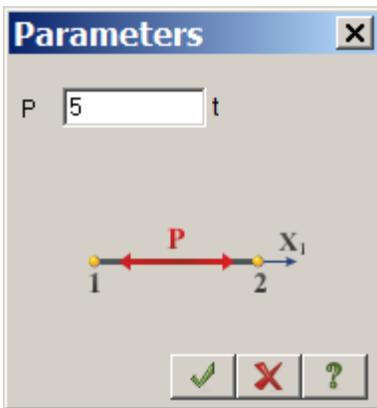
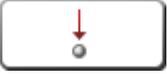


Figure 10.18 Load parameters dialog box

To create load case No.5:

- ⇒ To change the number of the current load case, click the **Next load case** button  located on the Status bar or on the toolbar.
- ⇒ Specify with the pointer node of the lower edge of foundation - node No.1.
- ⇒ In the **Define loads** dialog box, click the **Loads on nodes** tab.
- ⇒ In the **Load type** area, click the **Concentrated load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 0.001$ t.
- ⇒ Click **OK** .
- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button  once again in order to make this command not active.

To edit flags of drawing:

- ⇒ In the **Display** dialog box (see Figure 10.5), clear the **Node numbers** check box on the **Nodes** tab.
- ⇒ On the **Elements** tab, select the **Stiffness in colour** check box.
- ⇒ Click **Redraw** .

Obtained model is presented in Figure 10.19.

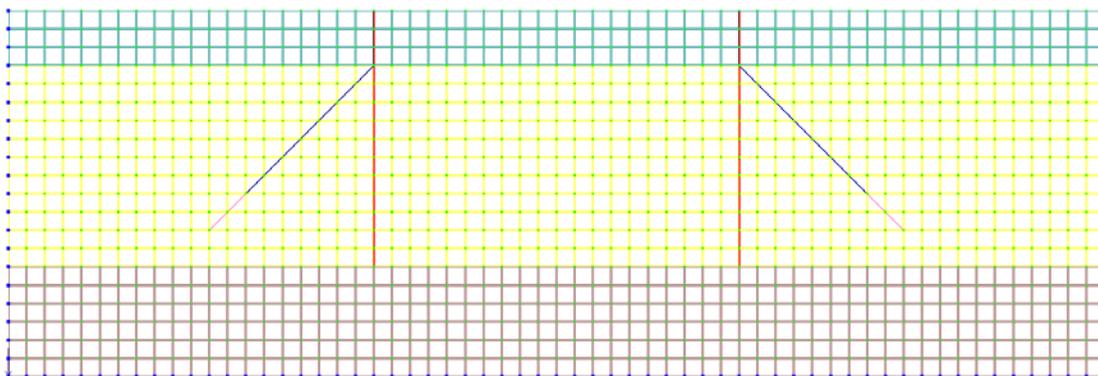


Figure 10.19 Design model of foundation

Step 6. Modelling stages of construction and nonlinear load cases

To model stages of construction:

- ⇒ On the **Analysis** ribbon tab, on the **Nonlinearity** panel, click **Assemblage** (button  on the toolbar).
- ⇒ In the **Model nonlinear load cases of structure** dialog box (see Figure 10.20), click the **Add** button  (in the left part of the dialog box, under **History**, the first load history is added and the row with load case indicated with question mark will become selected automatically).
- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, select **By FE type** check box and specify **FE type 281 - physically nonlinear rectangular FE of 2D problem (soil)**.
- ⇒ Click **Apply** .
- ⇒ When the elements are selected on the model, in the **Model nonlinear load cases of structure** dialog box, in the **Elements to be assembled** area, click **All selected** button. Numbers of elements selected on the model will be displayed in the list automatically.

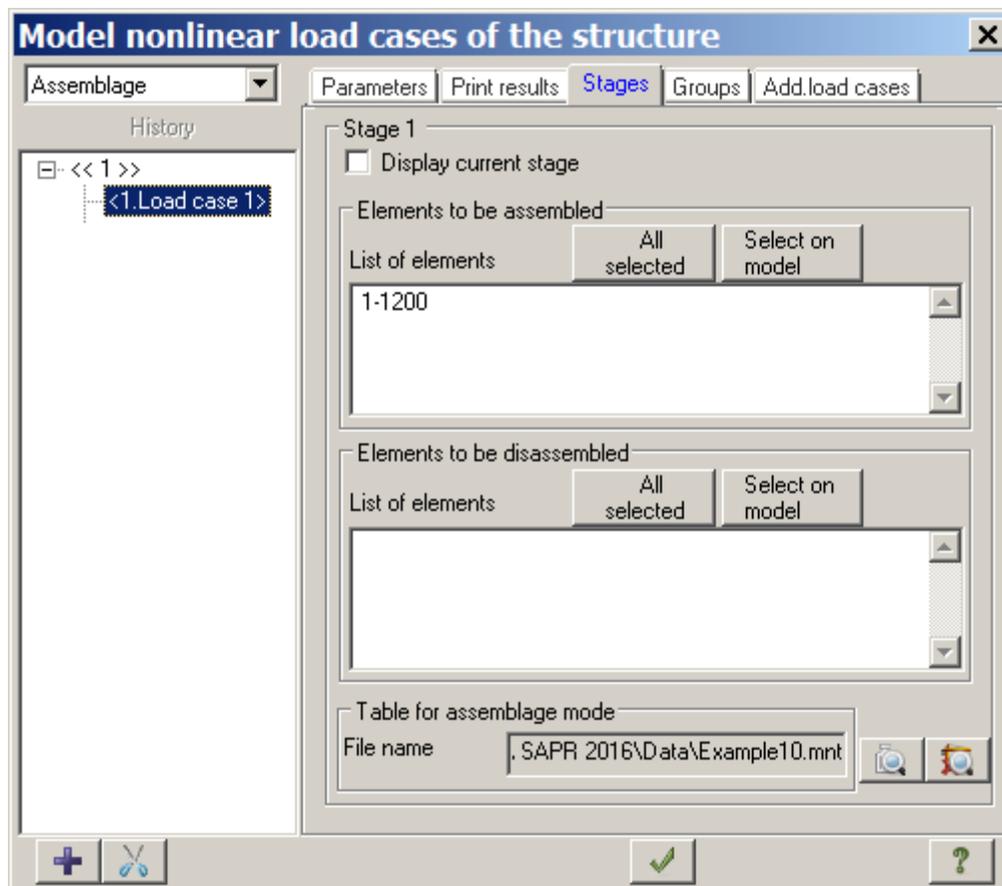


Figure 10.20 **Model nonlinear load cases of structure** dialog box

- ⇒ To unselect elements, on the **Select** toolbar, click **Unselect all** button .
- ⇒ To create the second stage of assemblage, in the **Model nonlinear load cases of structure** dialog box, click the **Add** button .
- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, clear **By FE type** check box and select **By stiffness** check box. Then specify **4. Numerical for FE 2**.

- ⇒ Click **Apply** .
- ⇒ When the elements are selected on the model, in the **Model nonlinear load cases of structure** dialog box, in the **Elements to be assembled** area, click **All selected** button.
- ⇒ To unselect elements, on the **Select** toolbar, click **Unselect all** button .

- ⇒ To create the third stage of assemblage, in the **Model nonlinear load cases of structure** dialog box, click the **Add** button .
- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, select **By stiffness** check box and specify **1. FE 281-284 Numerical**.
- ⇒ On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .
- ⇒ With the pointer, select elements of the 1st layer of foundation inside elements of sheet piling. To do this, just drag the pointer (selection window) from left to right as shown in Fig. 10.21.
- ⇒ When the elements are selected on the model, in the **Model nonlinear load cases of structure** dialog box, in the **Elements to be disassembled** area, click **All selected** button.
- ⇒ To unselect elements, on the **Select** toolbar, click **Unselect all** button .

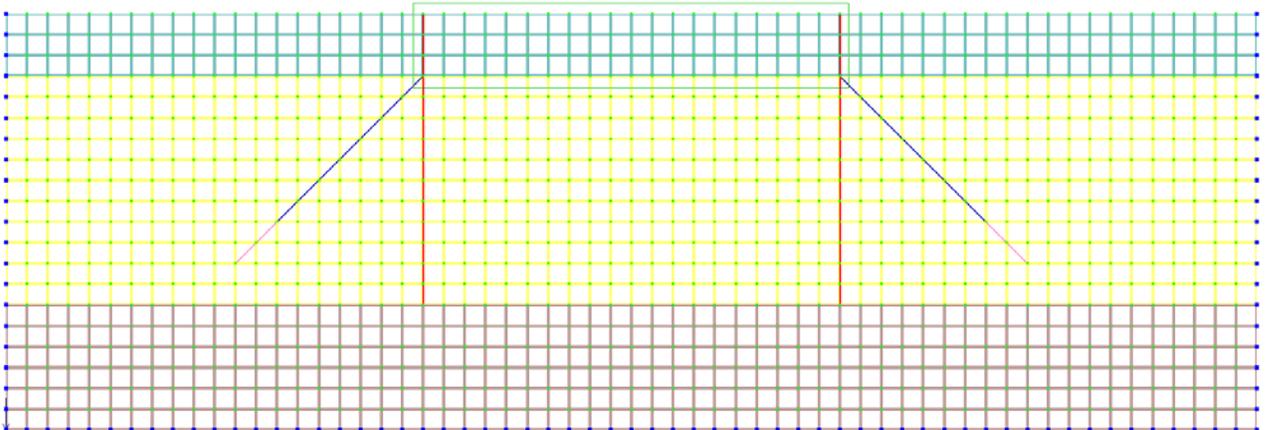


Figure 10.21 How to select elements of the 1st layer of foundation inside the sheet piling

- ⇒ To create the fourth stage of assemblage, in the **Model nonlinear load cases of structure** dialog box, click the **Add** button .
- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, clear **By stiffness** check box and select **By FE type**. Then specify **FE type 1 - FE of 2D truss**.
- ⇒ Click **Apply** .
- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, in the **By FE type** list, specify **FE type 208 - physically nonlinear special 2-node FE of pretension**.
- ⇒ Click **Apply** .
- ⇒ When the elements are selected on the model, in the **Model nonlinear load cases of structure** dialog box, in the **Elements to be assembled** area, click **All selected** button.
- ⇒ To unselect elements, on the **Select** toolbar, click **Unselect all** button .

- ⇒ To create the fifth stage of assemblage, in the **Model nonlinear load cases of structure** dialog box, click the **Add** button .
- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, clear **By FE type** check box and select **By stiffness**. Then specify **2. FE 281-284 Numerical**.
- ⇒ With the pointer, select elements 5 top rows of elements of the 2nd layer of foundation inside elements of sheet piling. To do this, just drag the pointer (selection window) from left to right as shown in Fig. 10.22.
- ⇒ When the elements are selected on the model, in the **Model nonlinear load cases of structure** dialog box, in the **Elements to be disassembled** area, click **All selected** button.
- ⇒ To unselect elements, on the **Select** toolbar, click **Unselect all** button .

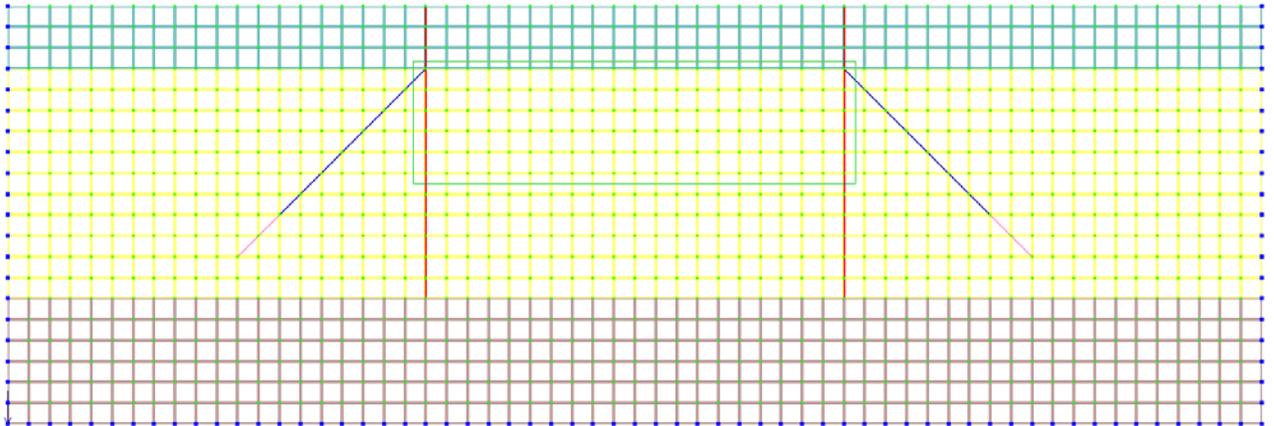


Figure 10.22 How to select elements of the 2nd layer of foundation inside the sheet piling

To model nonlinear load cases:

- ⇒ In the **Model nonlinear load cases of structure** dialog box, click **Parameters** tab and select the row that corresponds to the second stage of assemblage.
- ⇒ Select the **Clear displacements** check box (see Figure 10.23).
- ⇒ Other parameters remain by default.
- ⇒ Click **OK** .

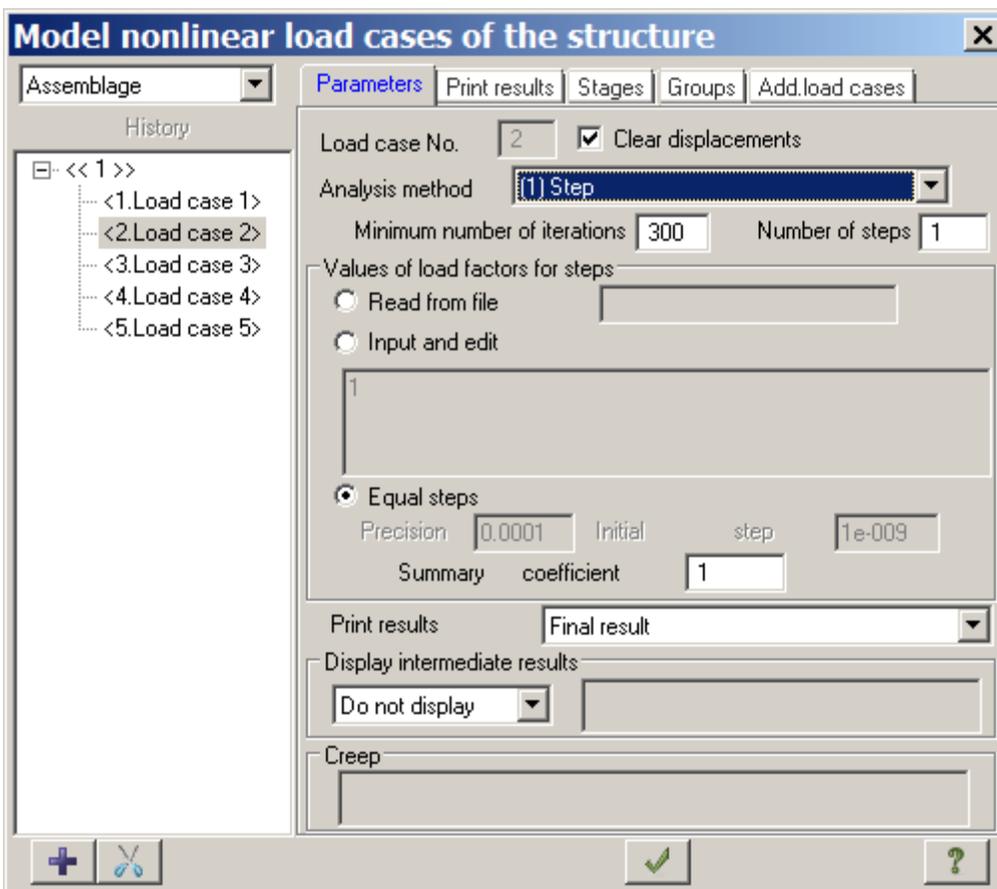


Figure 10.23 **Model nonlinear load cases of structure** dialog box for defining parameters of nonlinear load case



To ignore the strain of foundation from the dead weight (at the second stage of assemblage), for the second nonlinear load case, select the **Clear displacements** check box. Stresses will be summed up automatically in ASSEMBLAGE system.

Step 8. Nonlinear analysis of the model

- ⇒ To carry out complete analysis, 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 analysis results



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

- ⇒ In the mode of analysis results visualization, by default design model is presented with account of nodal displacements (see Fig.10.24). To display the model without nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button .

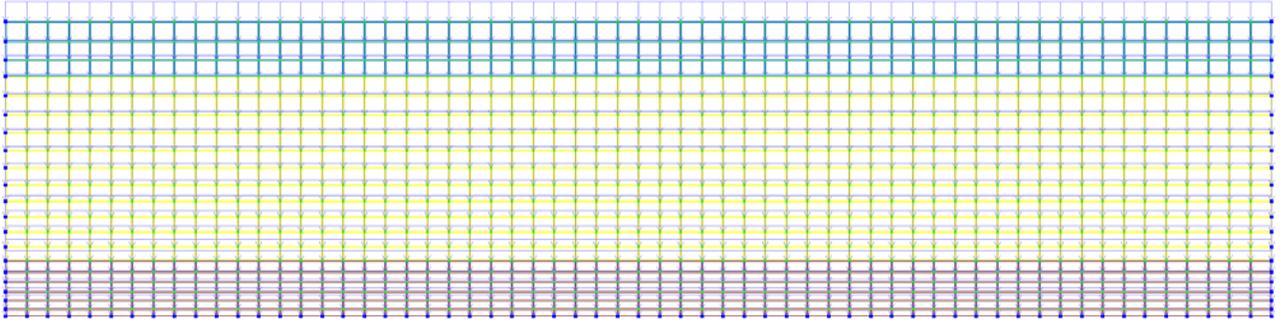


Figure 10.24 Design model with account of nodal displacements

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. **5** (that corresponds to the fifth stage of assemblage) and click **Apply**  .

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 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 contour plot of displacements along the X-axis, click **Displacements along X** button  on the same panel.

To present stress mosaic plots:

- ⇒ To present stress mosaic plot for N_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 N_x** button  on the same panel.
- ⇒ To present stress mosaic plot for N_z , click **Stress N_z** button  on the same panel.

To present diagrams of internal forces:

- ⇒ To display diagram M_y , on the **Results** tab, select **Forces in bars** panel and click **Moment diagrams (M_y)** button  .
- ⇒ To display diagram 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 display mosaic plots **My**, on the **Results** tab, select **Forces in bars** panel and click **Mosaic Plot My** 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.10.25), select **Forces** in the list.
- ⇒ Make sure that **All load cases** option is defined in the **Choose load case No.** list and 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.

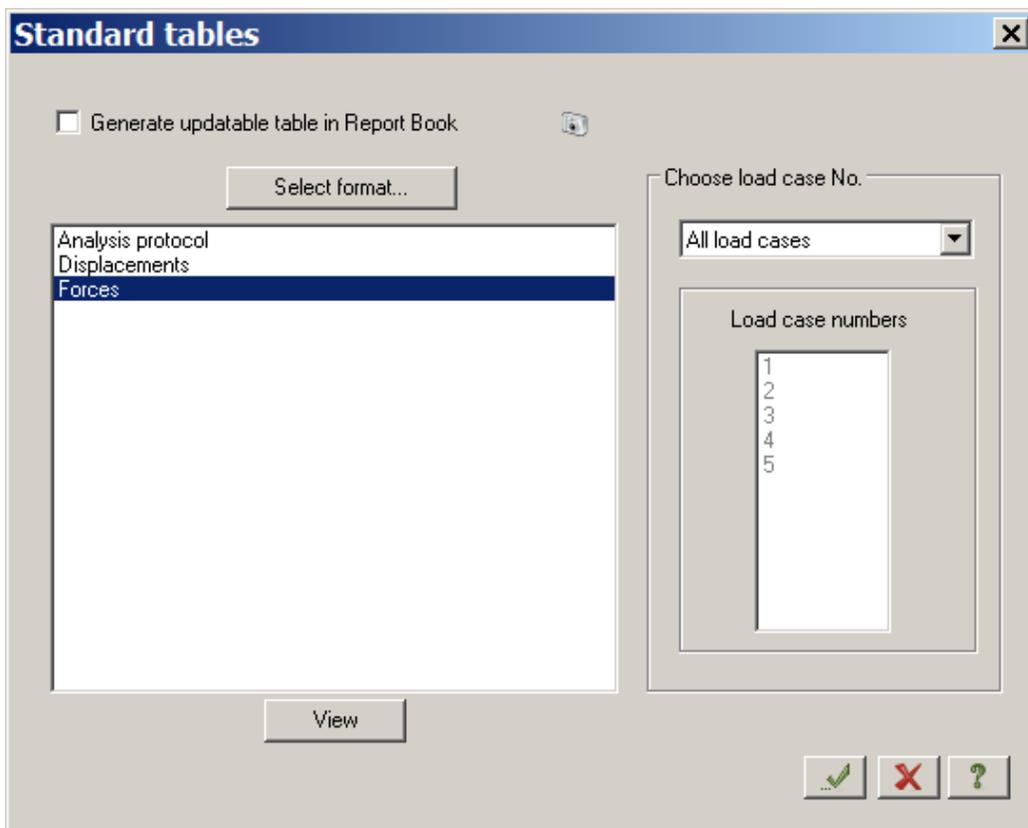
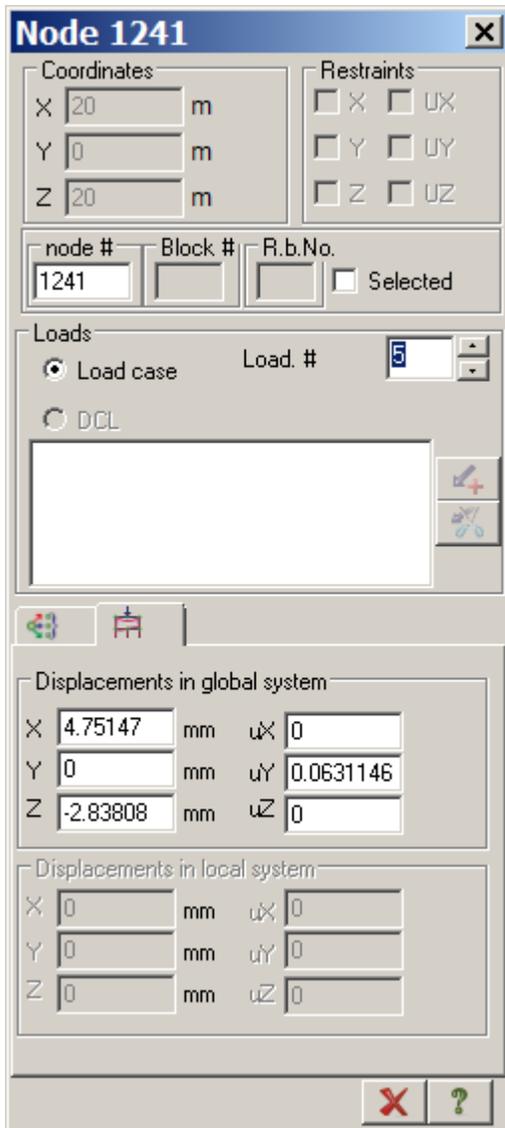


Figure 10.25 **Standard tables** dialog box

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

To present information about displacement of top of sheet piling:

- ⇒ To present information about displacement of top of sheet piling, on the **Select** menu, click **Information** (button  on the toolbar).
- ⇒ With the pointer specify one of the top nodes of sheet piling (for example, from the left side of a pit). In the dialog box (see Figure 10.26) you will see values of displacements in the specified node.



Node 1241			
Coordinates		Restrains	
X	20	m	<input type="checkbox"/> X <input type="checkbox"/> UX
Y	0	m	<input type="checkbox"/> Y <input type="checkbox"/> UY
Z	20	m	<input type="checkbox"/> Z <input type="checkbox"/> UZ
node #	Block #	R.b.No.	<input type="checkbox"/> Selected
1241			
Loads			
<input checked="" type="radio"/> Load case	Load. #	5	
<input type="radio"/> DCL			
Displacements in global system			
X	4.75147	mm	uX 0
Y	0	mm	uY 0.0631146
Z	-2.83808	mm	uZ 0
Displacements in local system			
X	0	mm	uX 0
Y	0	mm	uY 0
Z	0	mm	uZ 0

Figure 10.26 **Information about node No.** dialog box