Example 17. SOIL system for generation of 2D and 3D soil masses

In this lesson you will learn how to:
– generate 2D problem based on 3D model of soil, perform automatic triangulation, assign physical and mechanical properties for FE of soil;
– generate 3D soil model, perform automatic triangulation, assign physical and mechanical properties for FE of soil.

Description:
Soil model from example No.9 is used for generation of 2D and 3D soil masses.
Two variants of 3D soil mass – RC foundation slab 6 x 6 m, thickness 600 mm.

Loads (variant of 3D soil mass):
load case 1 – dead weight.

Edit soil model

Step 1. Editing soil model

For generation of 2D and 3D soil masses, the soil model from example 9 will be applied (see the Samples\Eng folder with examples).

- Open the *Example9.sld* file from the above-mentioned folder.
- To save the soil model under another name, on the FILE menu, click **Save as**.
- In the **Save as** dialog box (see Fig.17.1) define the following data:
  - file name – Example17;
  - location where you want to save this file (**Data** folder is displayed by default).
- Click **Save**.

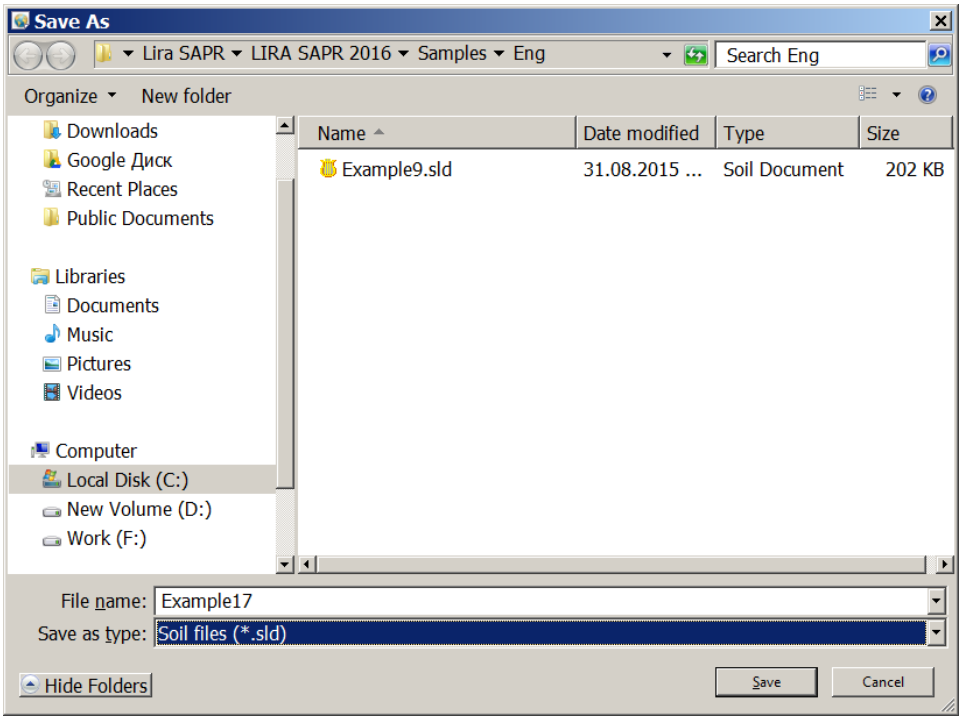

Figure 17.1. **Save as** dialog box

- To close the file with soil model, on the FILE menu, click **Close**.

Generate new problem (2D soil mass)

- On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2016** and then click **LIRA-SAPR 2016**.

Step 2. Generating new problem

- On the FILE menu, click **New** (button on the toolbar).
- In the **Model type** dialog box (see Fig.17.2) specify the following data:
  - problem name – **Example17_1**;
  - model type – **2 – Three degrees of freedom per node** (translations X, Z and rotation Uy) X0Z.
- Click **OK** .

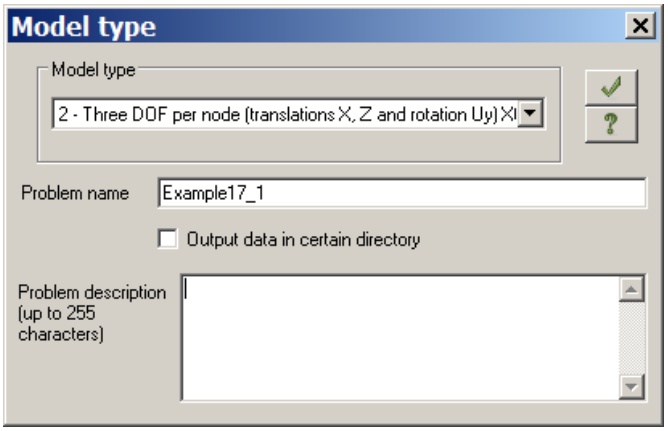

Figure 17.2. **Model type** dialog box

*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.

Generate 2D soil model

Step 3. Generating 2D soil model

It is possible to generate 2D soil model even when VISOR-SAPR working area is empty.

To start SOIL system:

- On the **Advanced edit options** ribbon tab, on the **SOIL** panel, click **2D soil mass (FE model)**.
- In the **Create plane FE of soil** dialog box (see Fig. 17.3), click **Attach soil model**.

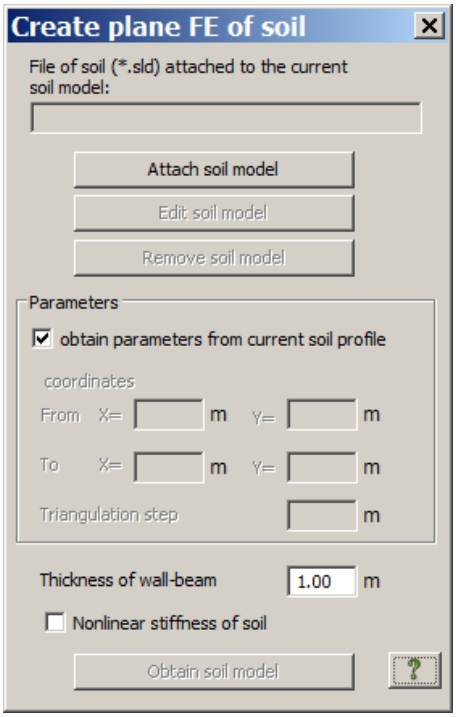

Figure17.3. **Create plane FE of soil** dialog box

- In the **Open file with soil model** dialog box (see Fig. 17.4), select the row *Example17.sld*.
- Click **Open**.

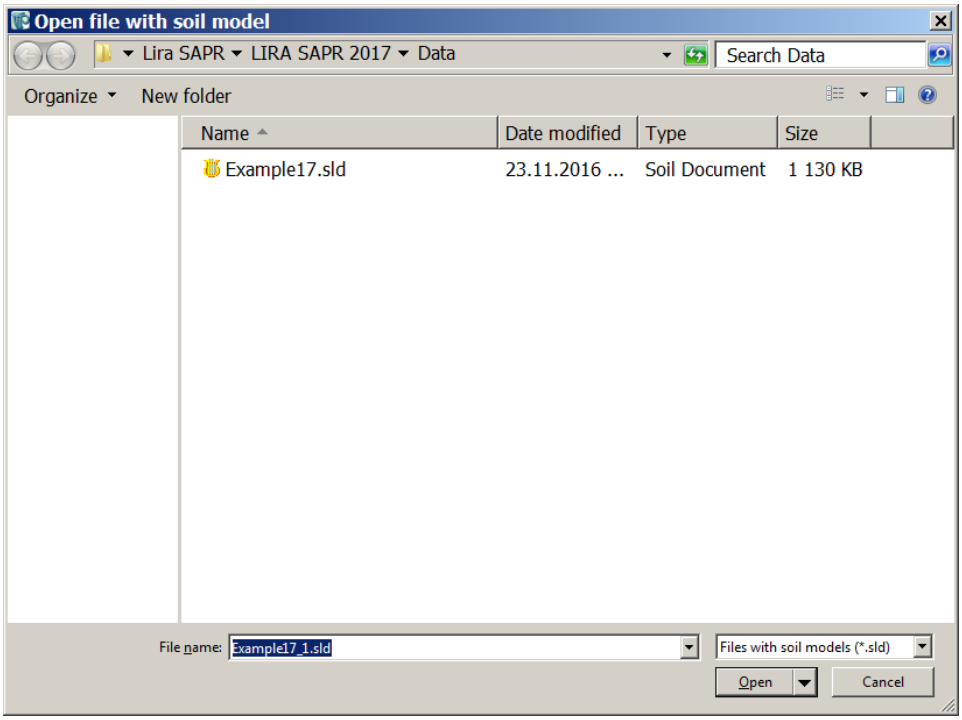

Figure 17.4. **Open file with soil model** dialog box

*Working window of the SOIL system activated with the **2D soil (FE model)** button (**Advanced edit options** ribbon tab, **SOIL** panel) and working window of the SOIL system activated with the **Soil model** button (**Advanced edit options** ribbon tab, **SOIL** panel) are not the same.*

To define and triangulate 2D vertical soil profile:

*In the **Arbitrary soil profile** floating box, 2D vertical soil profile and the load are displayed. By default, the soil profile is passed through coordinates of the first two boreholes.*

- To define arbitrary soil profile, in the **Arbitrary soil profile** floating box, click **Specify points on plan** button.
- Specify (in sequence) on model the beginning and the end of the soil profile line (see Fig. 17.5).

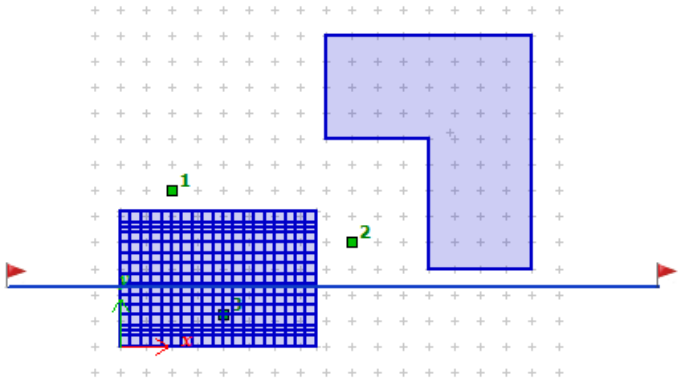

Figure 17.5. Defining the soil profile line

- To increase the floating dialog box, drag the top border of **Arbitrary soil profile** box upward to appropriate size.
- In the same floating box, define the **Step** (triangulation step for this soil profile) as equal to 0.8 (m).
- Click **Triangulate** .
- To save the data, on the FILE menu, click **Save** (button on the toolbar).

To import data to VISOR-SAPR:

- To return to the model of design model generation, click LIRA-SAPR button on the taskbar (by default, it is located at the bottom of the screen).
- To automatically generate stiffness parameters for soil FE as nonlinear ones, in the **Create plane FE of soil** dialog box (see Fig. 17.6), select the **Nonlinear stiffness of soil** check box (by default, thickness of such plane FE is taken as equal to 1m).
- To display design model that contains only triangular FE of soil with stiffness types that correspond to defined soil properties, click **Obtain soil model**.
- Close the **Create plane FE of soil** dialog box.

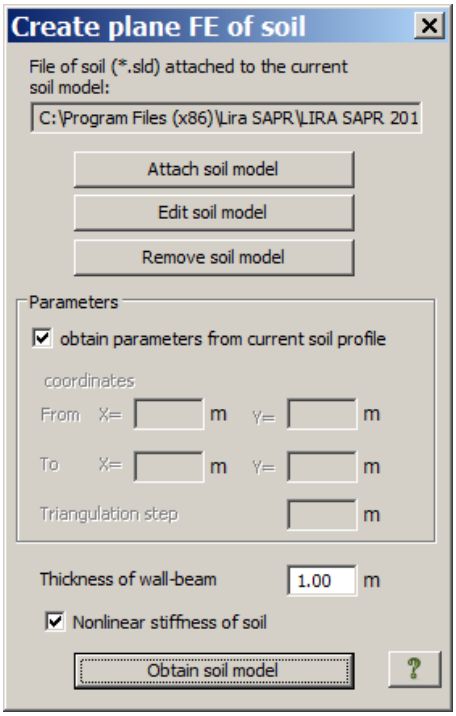

Figure 17.6. **Create plane FE of soil** dialog box

Two blocks are automatically generated in the obtained design model. One block includes nodes and elements located at the place where soil is excavated (foundation pit), the second block includes nodes and elements located outside the limits of foundation pit.

To complete generation of design model in VISOR-SAPR module:

- For description how to complete generation of design model, see Example 10.

It is recommended that you should pack the model when design model is complete.
*To obtain proper analysis results for plate elements, it is necessary to unify their local axes. To do this, on the **Edit plates** context tab, on the **Plates** panel, click **Local axes of plates** button. Then unify local axes of plates in the dialog box.*
Load defined in the SOIL system is not transferred to design model because location of soil profile is not known.

- When design model is generated, you could carry out physically nonlinear analysis of the model and evaluate analysis results.

Generate new problem (3D soil mass)

Step 4. Generating new problem (3D soil mass)

- On the FILE menu, click **New** (button on the toolbar).
- In the **Model type** dialog box (see Fig.17.7) specify the following data:
  - problem name – **Example17_2**;
  - model type – **5 – Six degrees of freedom per node** (translations X, Y, Z and rotations Ux, Uy, Uz).
- Click **OK** .

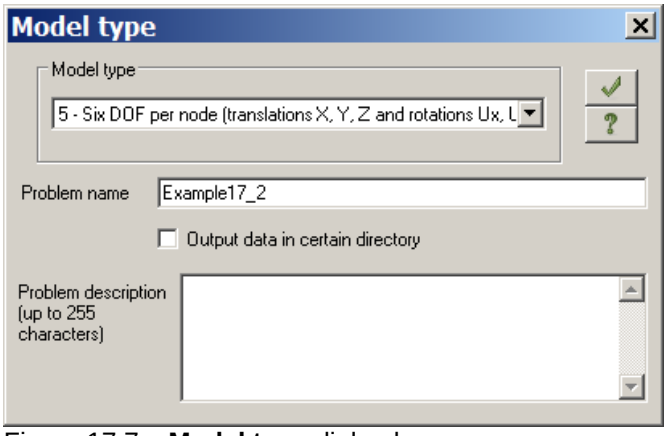

Figure 17.7. **Model type** dialog box

*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.

Generate model geometry for foundation slab

Step 5. Generating model geometry for foundation slab

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

L(m)	N	L(m)	N
0.5	12	0.5	12

  - other parameters remain by default (see Fig.17.8).
- Click **Apply** .

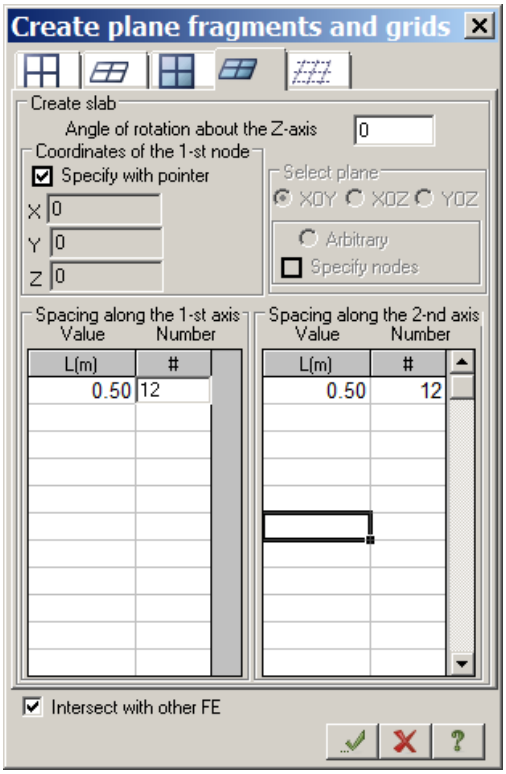

Figure 17.8. **Create plane fragments and grids** dialog box

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 – **Example17_2**;
  - location where you want to save this file (**Data** folder is displayed by default).
- Click **Save**.

Generate 3D soil model

Step 6. Generating 3D soil model

To activate SOIL system:

- 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.

- On the **Advanced edit options** tab, on the **Soil** panel, click **3D soil mass (FE model)** button .
- In the **Create 3D FE of soil** dialog box (see Fig.17.9), in the **List of elements included into foundation** area, select **To list** button. (Elements of foundation slab under which it is necessary to generate 3D soil model will be included into list.)
- Click **Attach soil model**.

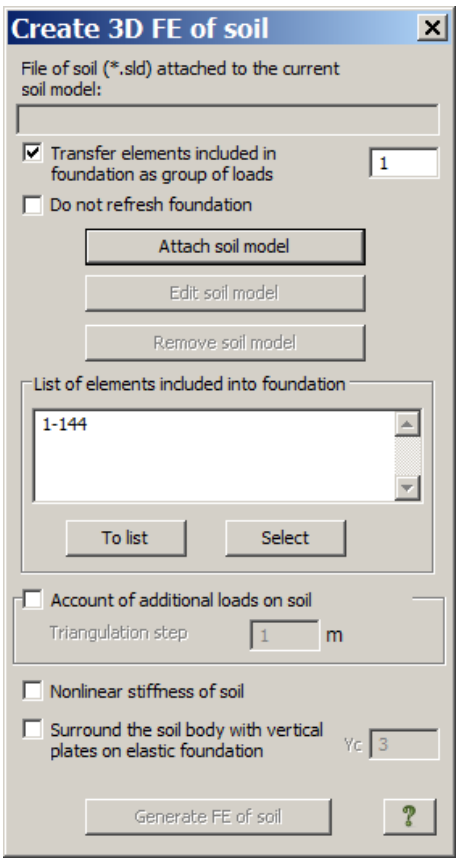

Figure 17.9. **Create 3D FE of soil** dialog box

- In the **Open file with soil model** dialog box (see Fig.17.4), select the file *Example17.sld*.
- Click **Open**.

*Working window of the SOIL system activated with the **3D soil (FE model)** button (**Advanced edit options** ribbon tab, **SOIL** panel) and working window of the SOIL system activated with the **Soil model** button (**Advanced edit options** ribbon tab, **SOIL** panel) are not the same.*

To generate 3D soil model:

- In the SOIL system, on the MODEL menu, click **Solid elements** (button on the toolbar).
- In the **Solid elements** dialog box (see Fig.17.10), according to schematic presentation define the following data for the plan area:
  - distance between footing and left margin of mesh region (X) – Xl = 5 m;
  - distance between footing and right margin of mesh region (X) – Xr = 15 m;
  - distance between footing and top margin of mesh region (X) – Yt = 10 m;
  - distance between footing and bottom margin of mesh region (X) – Yb = 5 m;
  - distance from footing elevation (Z) – H = 20 m.
- Then define the following parameters for triangulation of plan and division along the depth:
  - max (external) triangulation step – S = 1 m;
  - min wedge height – So = 0.5 m;
  - max wedge height – Sh = 1 m.
- Define elevation – Z = 96 m.
- Click **Apply** .

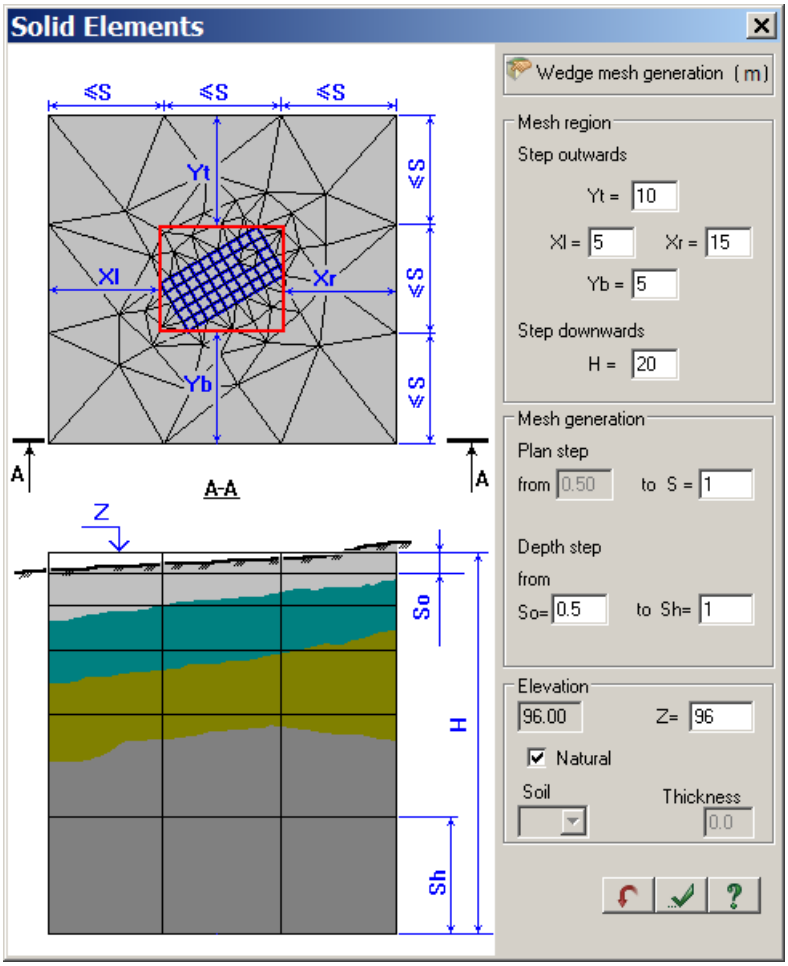

Figure 17.10. **Solid elements** dialog box

- To save the data, on the FILE menu, click **Save** (button on the toolbar).

To import data to VISOR-SAPR:

- To return to the model of design model generation, click LIRA-SAPR button on the taskbar (by default, it is located at the bottom of the screen).
- In the **Create 3D FE of soil** dialog box (see Fig.17.11), define the following parameters:
  - to consider the load from neighbouring structures defined in the SOIL system, select the **Account of additional loads on soil** check box and define triangulation step (for areas where these loads are applied) as equal to 0.5 m;
  - to automatically generate stiffness parameters for soil FE as nonlinear ones, select the **Nonlinear stiffness of soil** check box.
  - To display design model of soil body that corresponds to defined properties, click **Generate FE of soil**.

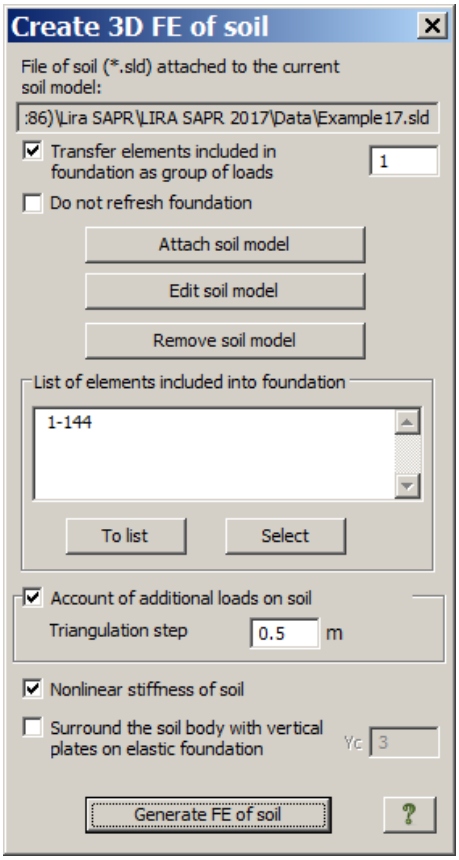

Figure 17.11. **Create 3D FE of soil** dialog box

To complete generation of design model in VISOR-SAPR module:

- After import to VISOR-SAPR, to complete generation of design model, according to description in previous examples, define the following data:
  - stiffness of foundation slab;
  - boundary conditions (bottom edge of soil should be fixed along the Z-axis, edges parallel to the X0Z-plane – along the Y-axis, edges parallel to the X0Y-plane – along the X-axis);
  - loads;
  - table for modelling of nonlinear load cases.

It is recommended that you should pack the model when design model is complete.
To obtain proper analysis results for solids, it is necessary to unify their local axes. To do this, on the ***Edit solids*** *context tab, on the* ***Solids*** *panel, click* ***Local axes of solids*** *button . Then unify local axes of plates in the dialog box.*

- When design model is generated, you could carry out physically nonlinear analysis of the model and evaluate analysis results.

