Example 1

Analysis of a circular loaded area resting on different soil layers

Contents Page

1 Description of the problem

An example of a loaded area resting on different soil layers is selected to illustrate some features of *ELPLA* for analyzing shell elements using circular and annular elements.

2 Geometry and properties

A circular loaded area of a radius a=5 [m] acting on three different soil layers as shown in Figure 1.1.

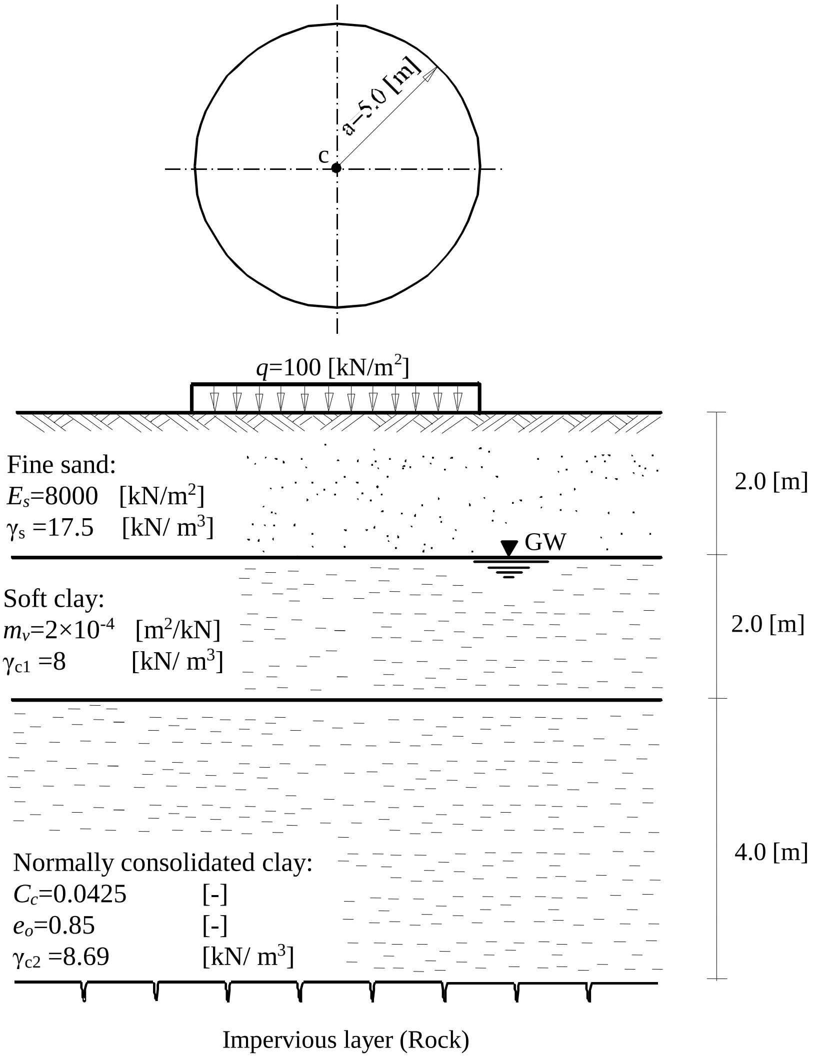


Figure 1.1 Soil profile under the circular loaded area

3 Creating the project

In this section, the user will learn how to create a project for analyzing a circular loaded area resting on different soil layers. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

3.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 1.2. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of Forms. The first Form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 1.2).

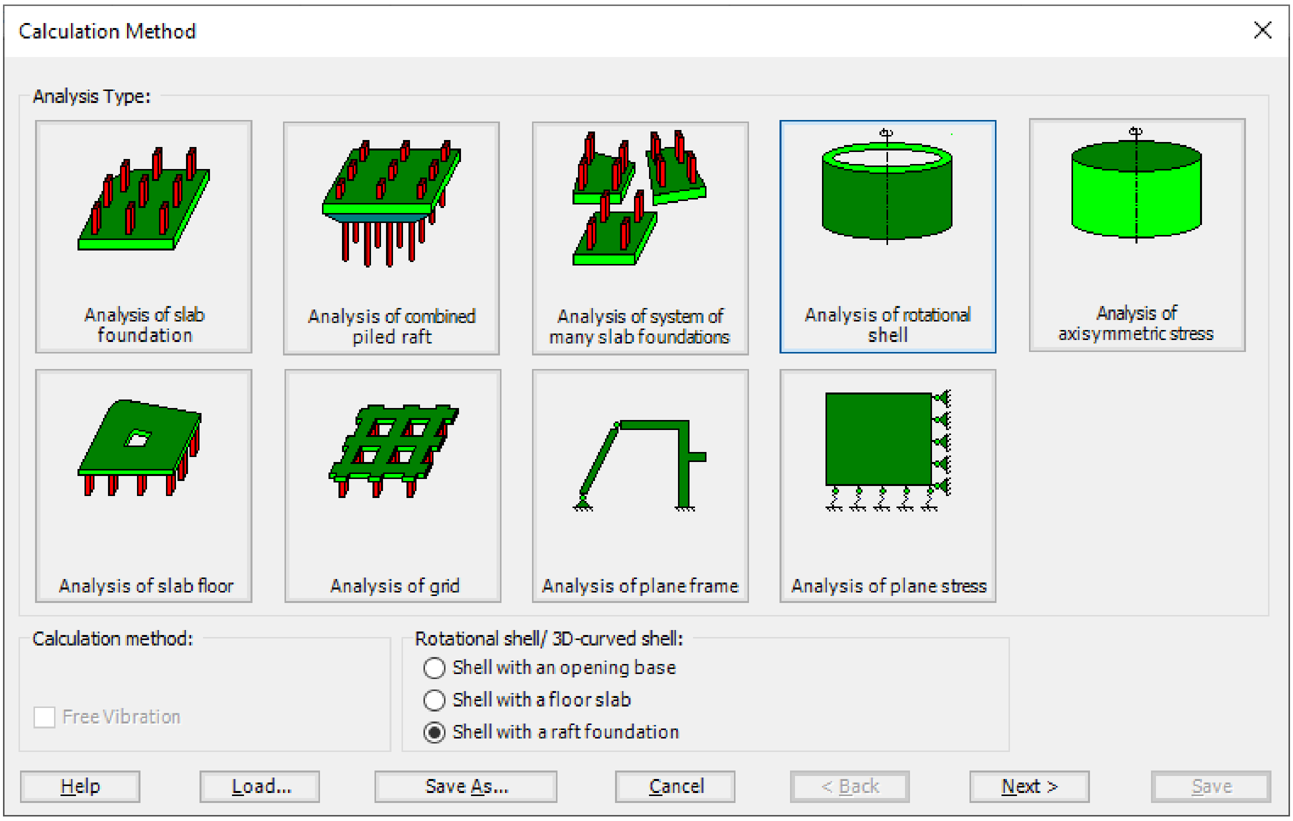


Figure 1.2 "Analysis Type" Form

As the analysis type is a circular loaded area problem, select "Analysis of rotational shell" button, and check "Shell with a raft foundation" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Calculation Methods" Form appears, Figure 1.3.

To define the calculation method:

- Select the calculation method "9-Flexible Foundation"
- Select the subsoil model as "Layered soil model"
- Click "Next" button to go to the next Form

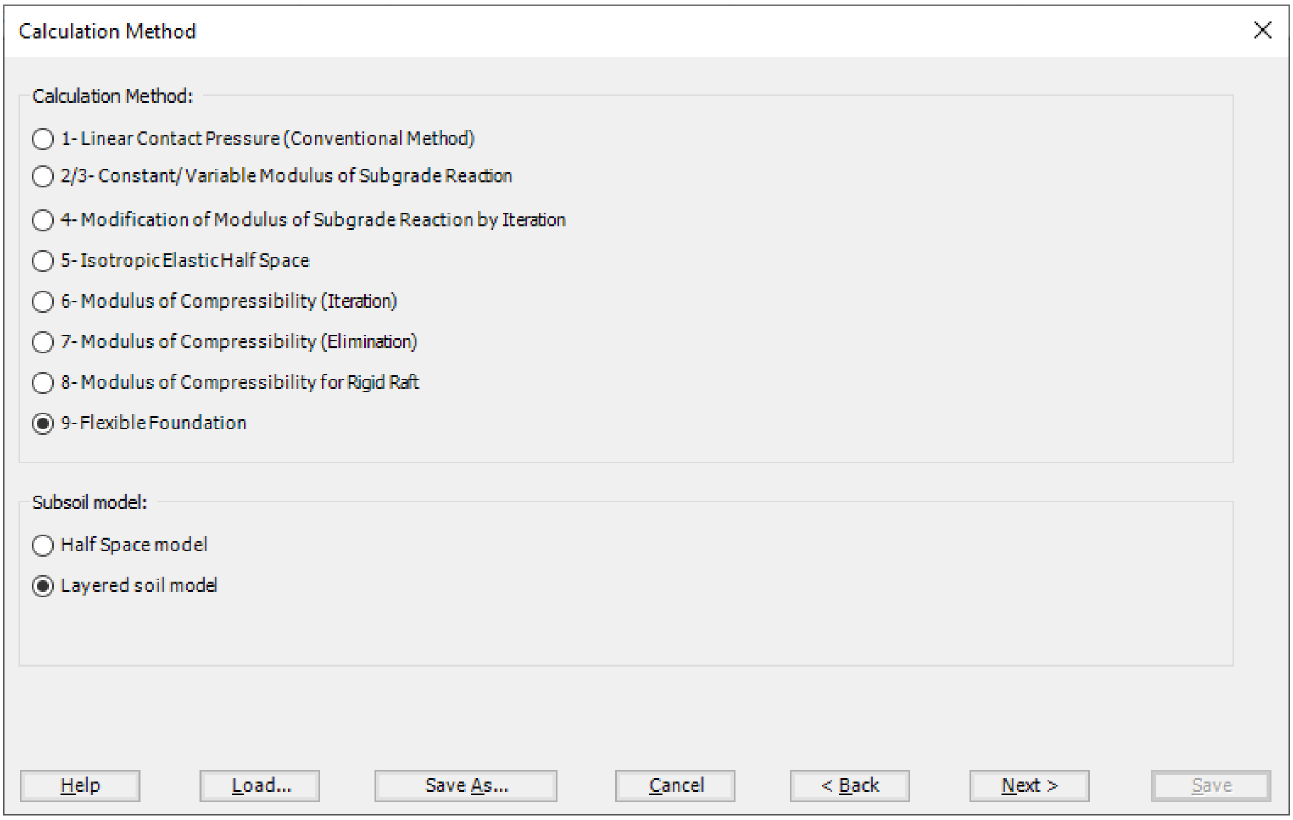


Figure 1.3 "Calculation Methods" Form

The last Form in the wizard is the "Options" Form, Figure 1.4. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Since no option will be considered in the analysis, click the "Save" button.

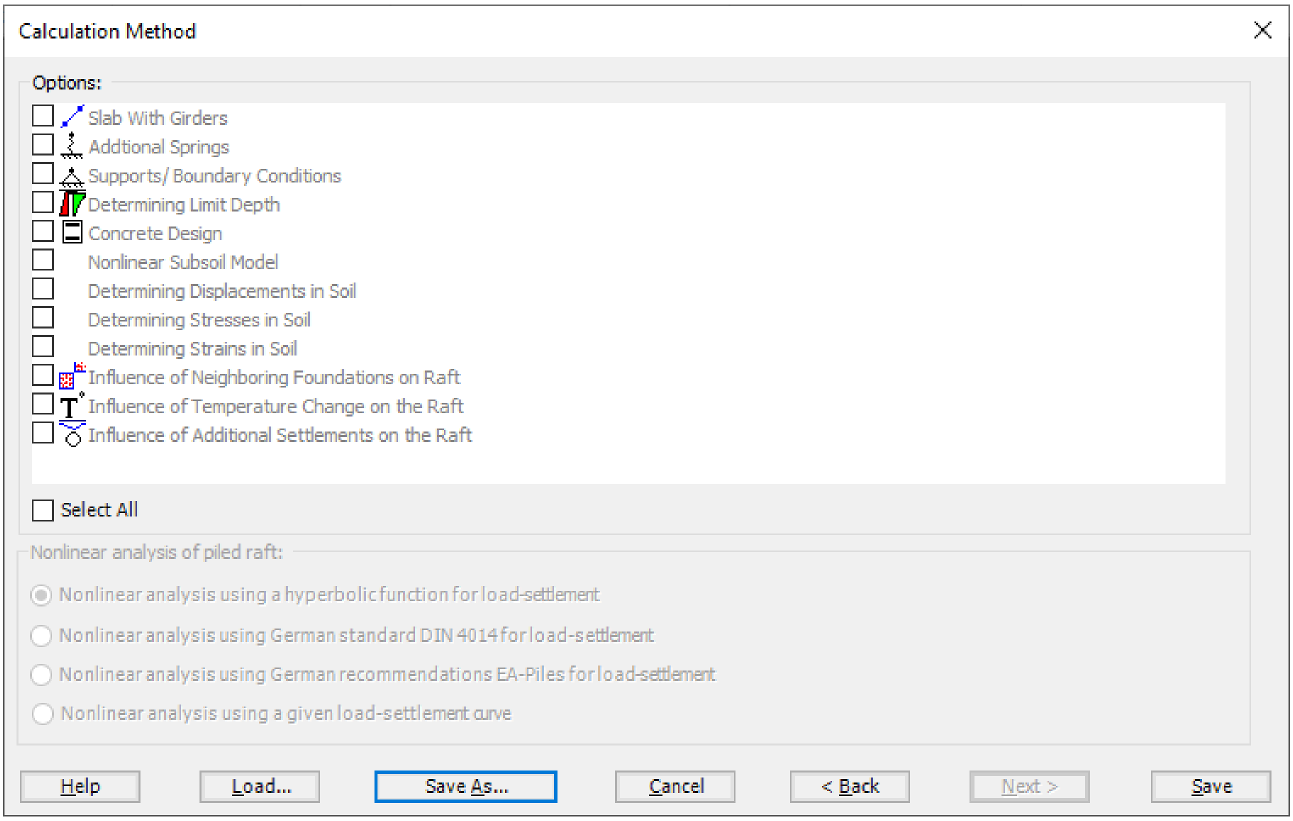


Figure 1.4 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 1.5. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Circular loaded area ". *ELPLA* will use automatically this file name in all reading and writing processes.

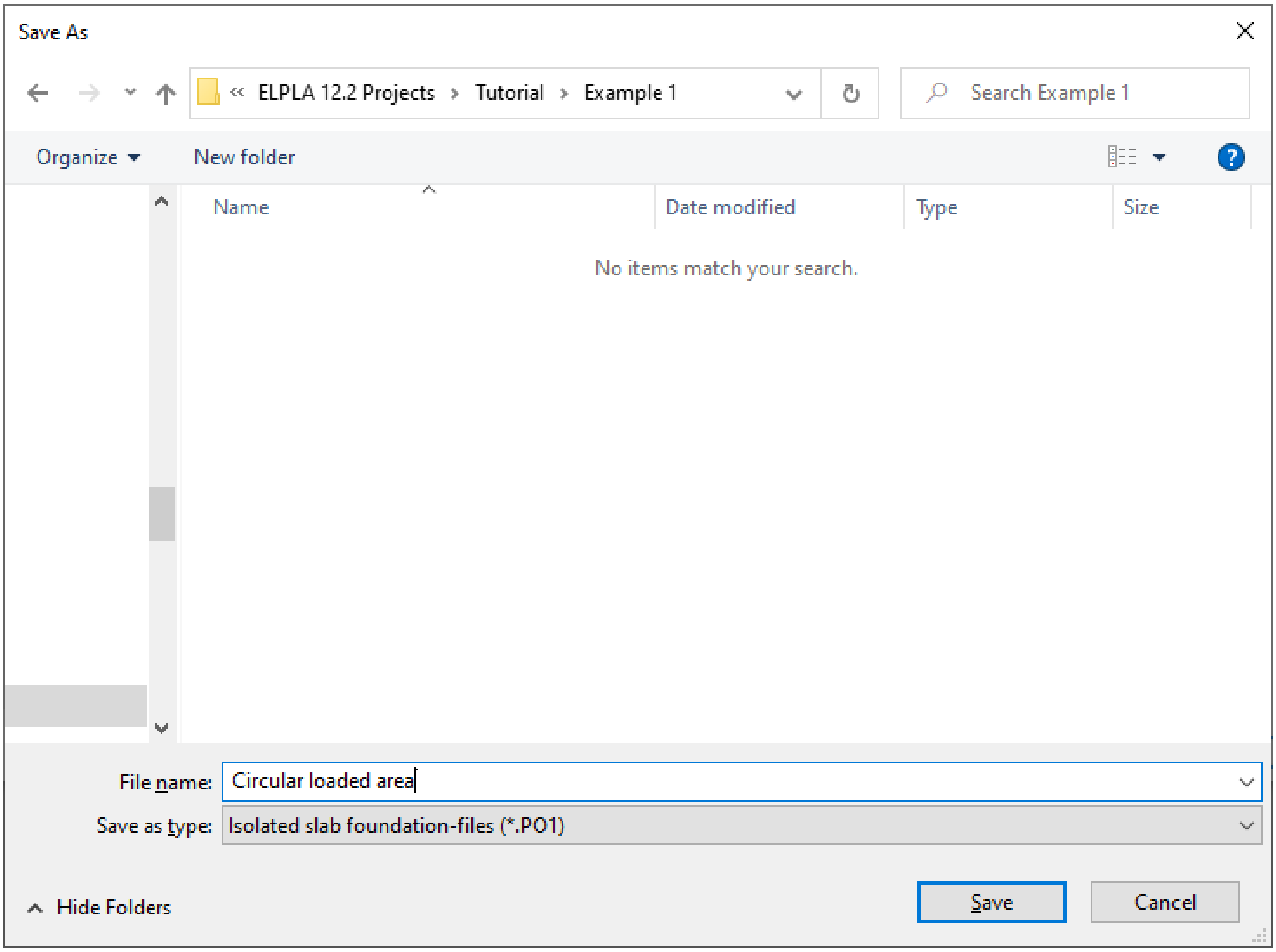


Figure 1.5 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Circular loaded area] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

3.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 1.6 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a circular loaded area resting on different soil layers"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

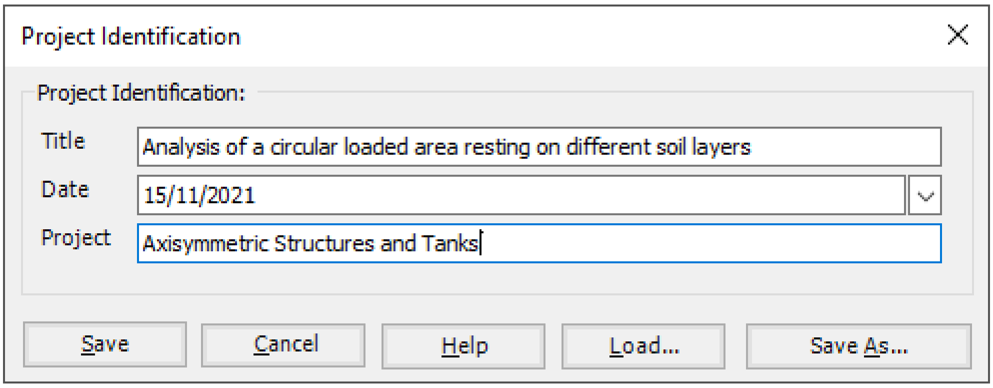


Figure 1.6 "Project Identification" dialog box

3.3 FE-Net data

For the given problem, the shell has a circular shape with a radius of a=5 [m] and is divided into 10 segments. *ELPLA* has different procedures for defining the FE-Net.

To define the FE-Net for this shell, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 1.7. This wizard will guide you through the steps required to generate a FE-Net. The first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of shells. These net templates are used to generate standard nets.

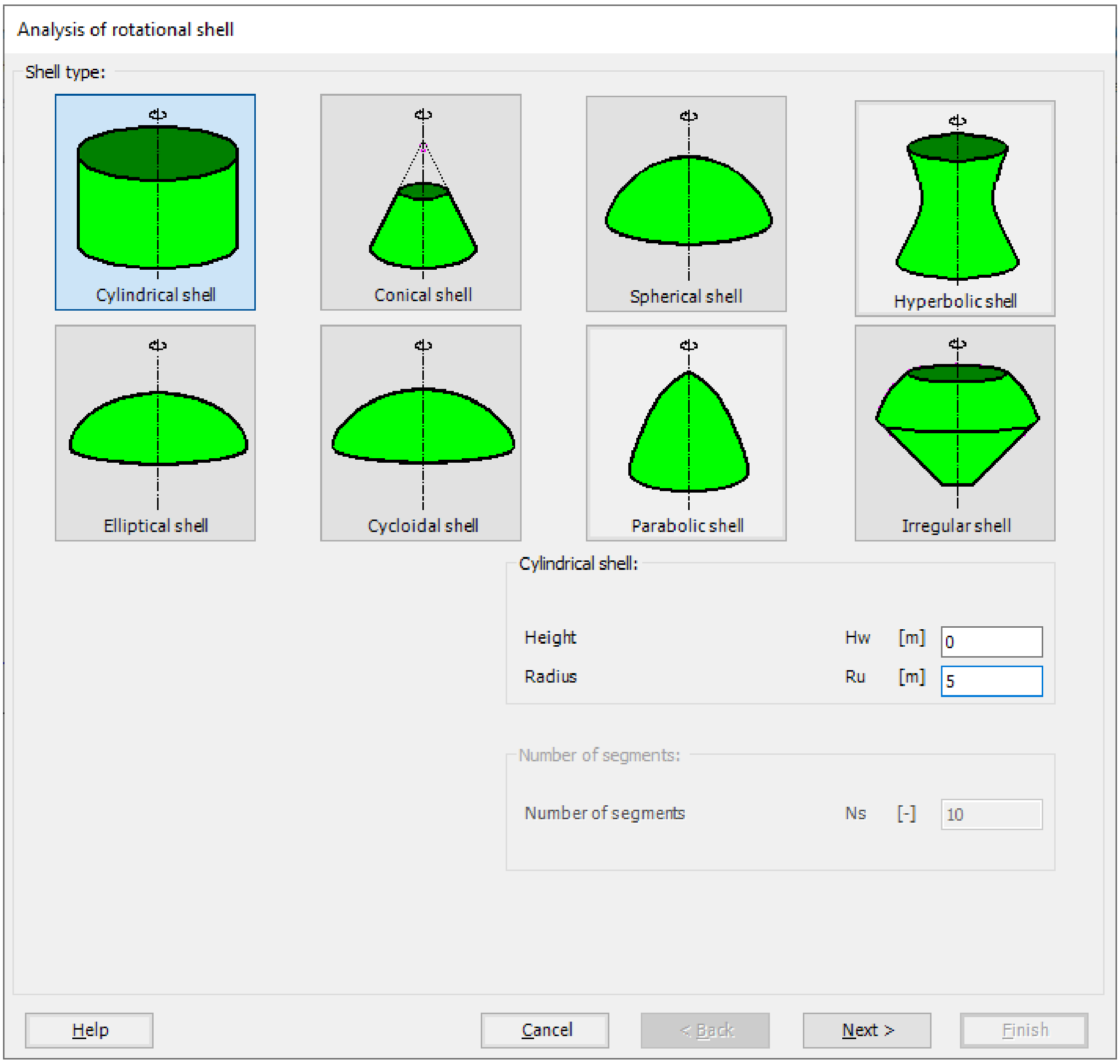


Figure 1.7 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Cylindrical shell" button
- Type 0 in the "Height" edit box, as the example is a circular plate
- Type 5 in the "Radius" edit box
- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Net of Base" Form appears Figure 1.8.

To edit the grid spacing in *x*-direction, do the following steps in "Grid in *x*-direction" frame:

- Choose "Constant grid interval" check box
- Type 10 in the "No. of grid intervals" edit box

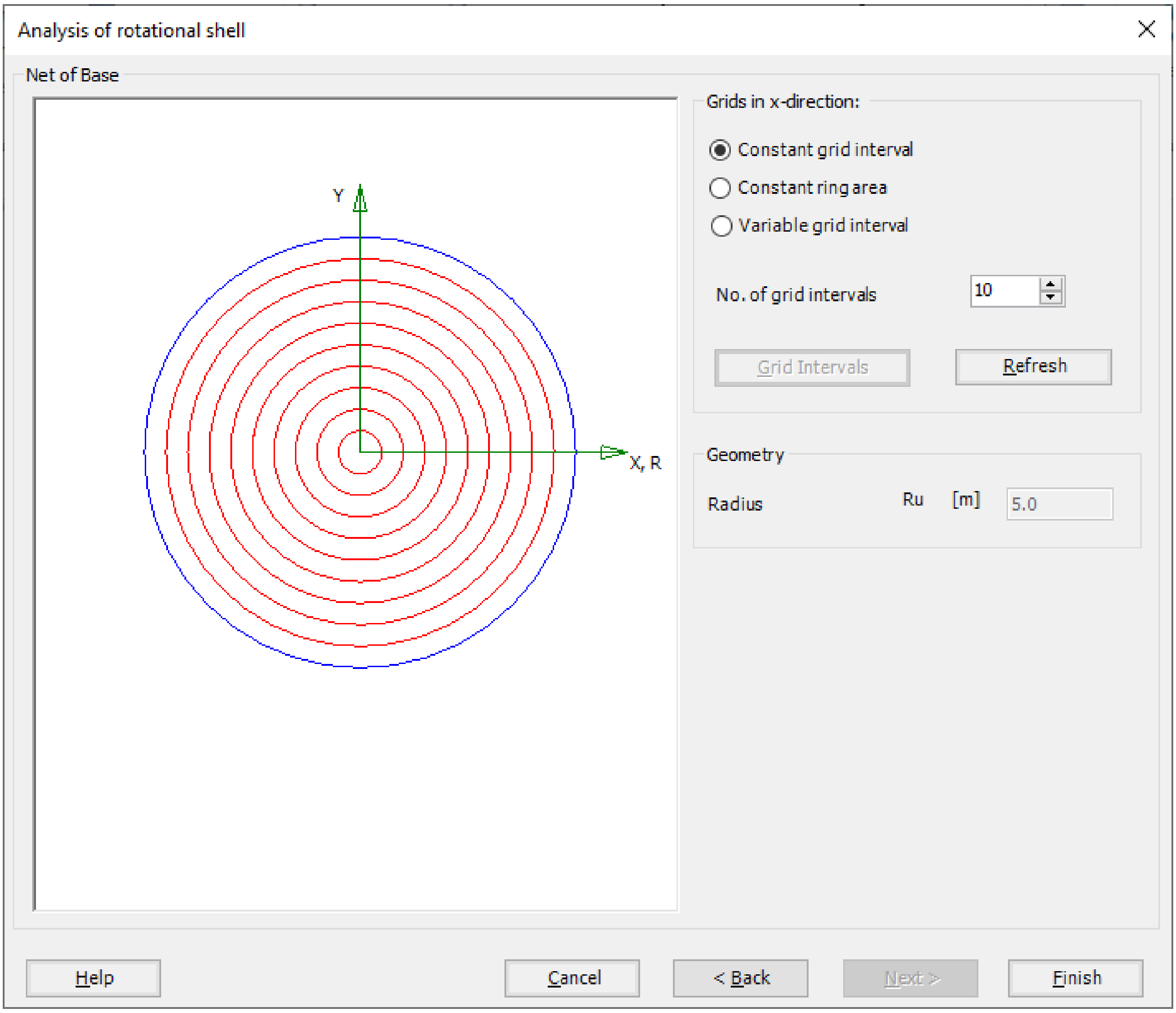


Figure 1.8 "Net of Base" Form

ELPLA will generate a sector from the circular area with 10 circular elements. The following window in Figure 1.9 appears with the generated net.

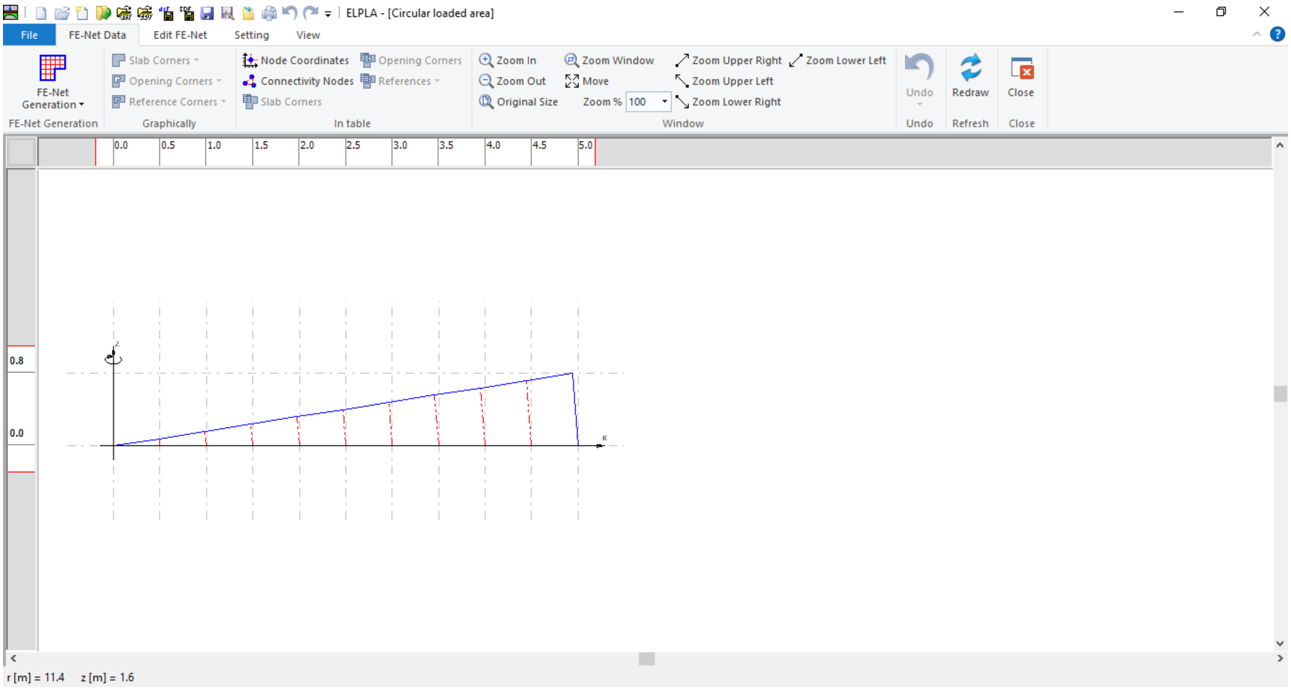


Figure 1.9 Generated FE-Net

After finishing the generation of the FE-Net, do the following two steps:

- Choose "Save" command from "File" menu in Figure 1.9 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 1.9 to close the "FE-Net" window and return to *ELPLA* main window

3.4 Soil Properties

In *ELPLA*, there are three different soil models with several calculation methods. Therefore, the soil properties for each method are required to be defined according to the used soil model. In the current example, the soil model, which is used in the analysis, is a "Layered Soil Model". This model requires that the subsoil have to be defined by boring logs. In the example, the boring log has multi-layers with different soil materials. The geotechnical data for each layer is unit weight of the soil γ_s and modulus of Elasticity for loading E_s and reloading W_s or Compression index Cc and recompression index Cr.

To define the soil properties, choose "Soil Properties" command from “Data” Tab. The following "Soil Properties" Window in Figure 1.10 appears with a default-boring log.

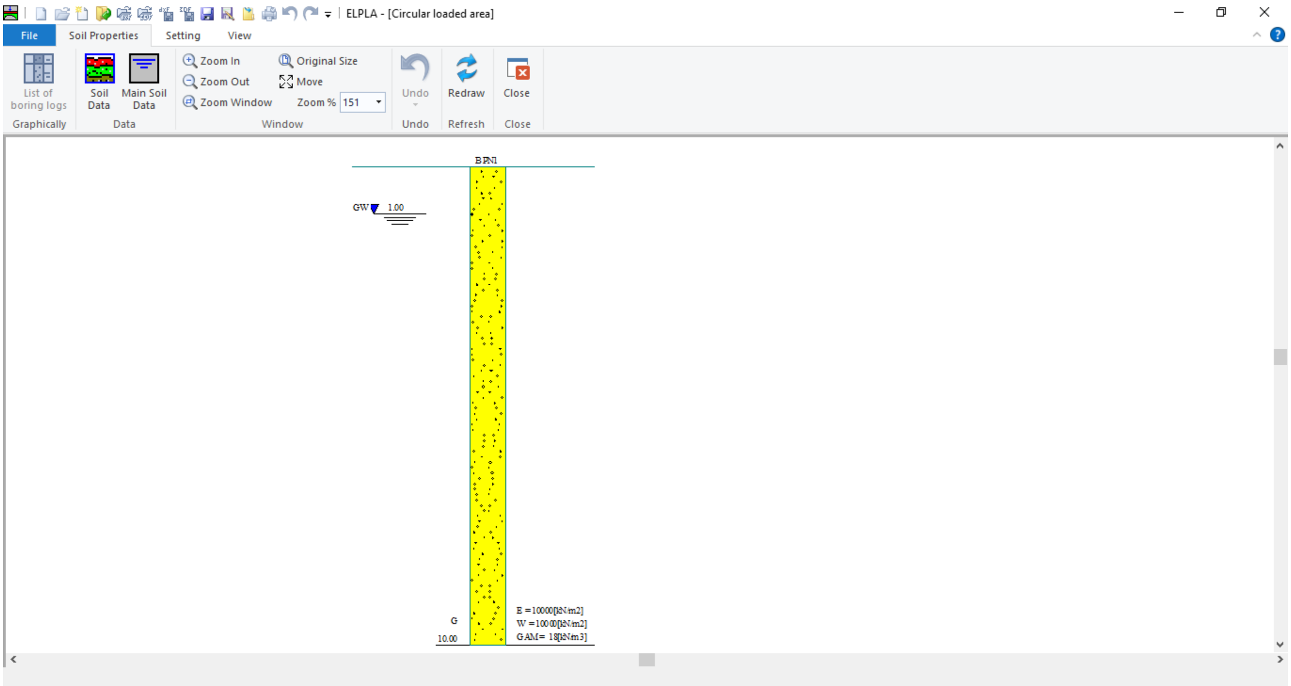


Figure 1.10 "Soil Properties" window with a default-boring log

In Figure 1.10, soil properties are defined through the "Data" menu, which contains the following two commands:

- "Soil Data" command defines the individual boring logs
- "Main Soil Data" command defines the general data for all soil layers

To enter the soil properties for the boring log of the current example

- Choose "Soil Data" command from "Data" menu in the window of Figure 1.10.
  The following dialog box in Figure 1.11 with default-boring log data appears

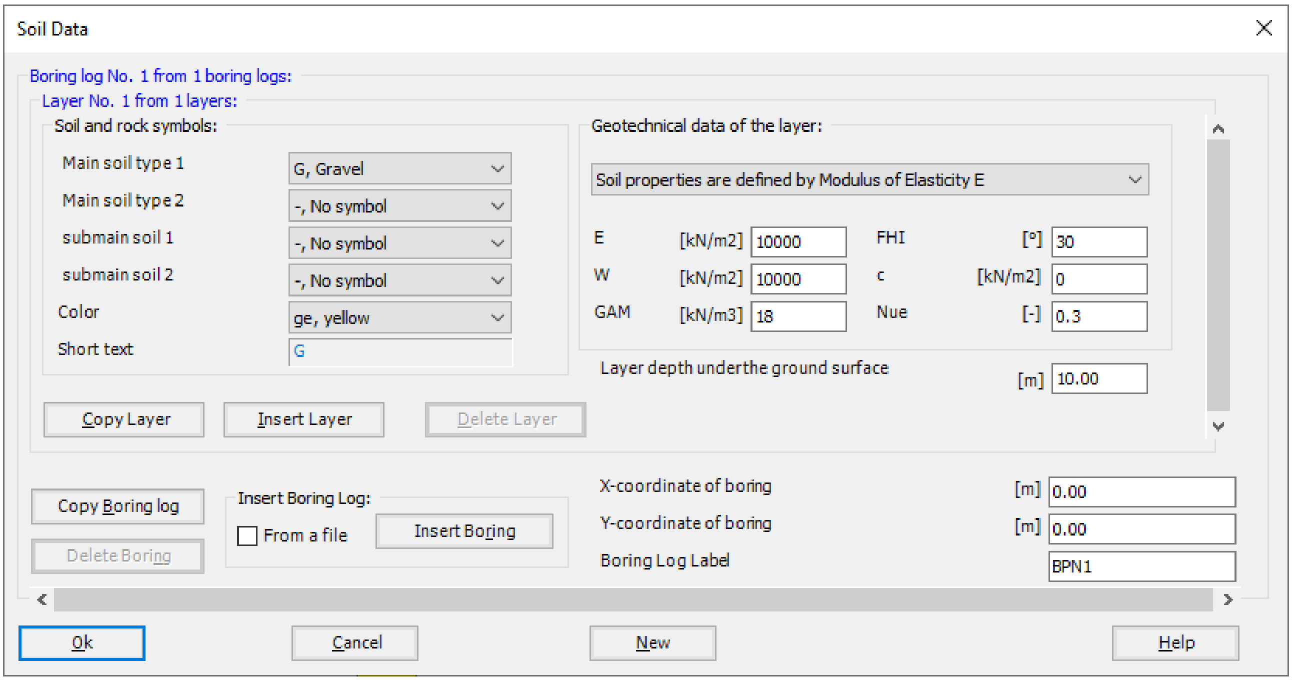


Figure 1.11 "Soil data" dialog box with default-boring log data

In the "Geotechnical data of the layer" dialog group box in Figure 1.11, Soil properties are defined by Modulus of compressibility, define the geotechnical data of the first soil layer of the boring log as follows:

E_s	= 8 000	[kN/m^2]
W_s	= 8 000	[kN/m^2]
Gam	= 18	[kN/m^3]
C	= 10	[kN/m^2]
φ	= 0	[°]

Due to the presence of the ground water, the soil above the ground water level has a differential unit weight from the soil under that level. Therefore, the layer depth of the first layer is taken to be 2 [m], which is equal to the ground water level. Now, type this value in "Layer depth under the ground surface" edit box.

In order to draw the soil layers by different symbols according to the German Standard DIN 4023, the soil type and color for each layer must be defined.

To define the soil type and color for the first layer, select "fS, Fine sand" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box in Figure 1.11. The color of the fine sand layer according to the German Standard DIN 4023 will be automatically created. The user can change this color. In addition, a short text "fS" will be automatically created for the fine sand.

To enter the second layer of the boring log

- Click "Copy Layer" button in Figure 1.11. A layer that has the same properties of the first layer will be copied
- Use the vertical scrollbar to move to the second soil layer. Layer No. will be typed automatically at the upper-left corner of the main dialog box of soil layers as a head title
- Change the value of the unit weight of the soil for the second soil layer from 18 [kN/m^3] to 8 [kN/m^3]
- Change the value "*ES*" and "*Ws*" from 8000 [kN/m^2] to 5000 [kN/m^2]
- Change the value of the layer depth under the ground surface from 2 [m] to 4 [m]
- From "Soil and rock symbols" dialog group box, Change "fS, Fine sand" into "T, Clay" as the soil type of the second layer is clay

To enter the third layer

- Click "Insert Layer" button in Figure 1.11, then a layer will be inserted
- Use the vertical scrollbar to move to the third soil layer
- In "Geotechnical data of the layer" dialog group box in Figure 1.11, Soil properties are defined by Compression Index *Cc*, define the geotechnical data of the clay layer as follows:

Cc	= 0.04		Phi	= 30	[°]
Cr	= 0.04		*c*	= 10	[kN/m^2]
Gam	= 8	[kN/m^3]	*eo*	= 0.85	

- Select "T, Clay" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box
- Type 8 in the "Layer depth under the ground surface" edit box

Note that the unit weight of the soil is used to determine the uniform load *q* [kN/m^2] on the circular area, which is equal to $\gamma_s \times d_f$.

Now all data and parameters for the boring log (Figure 1.12) have been entered.

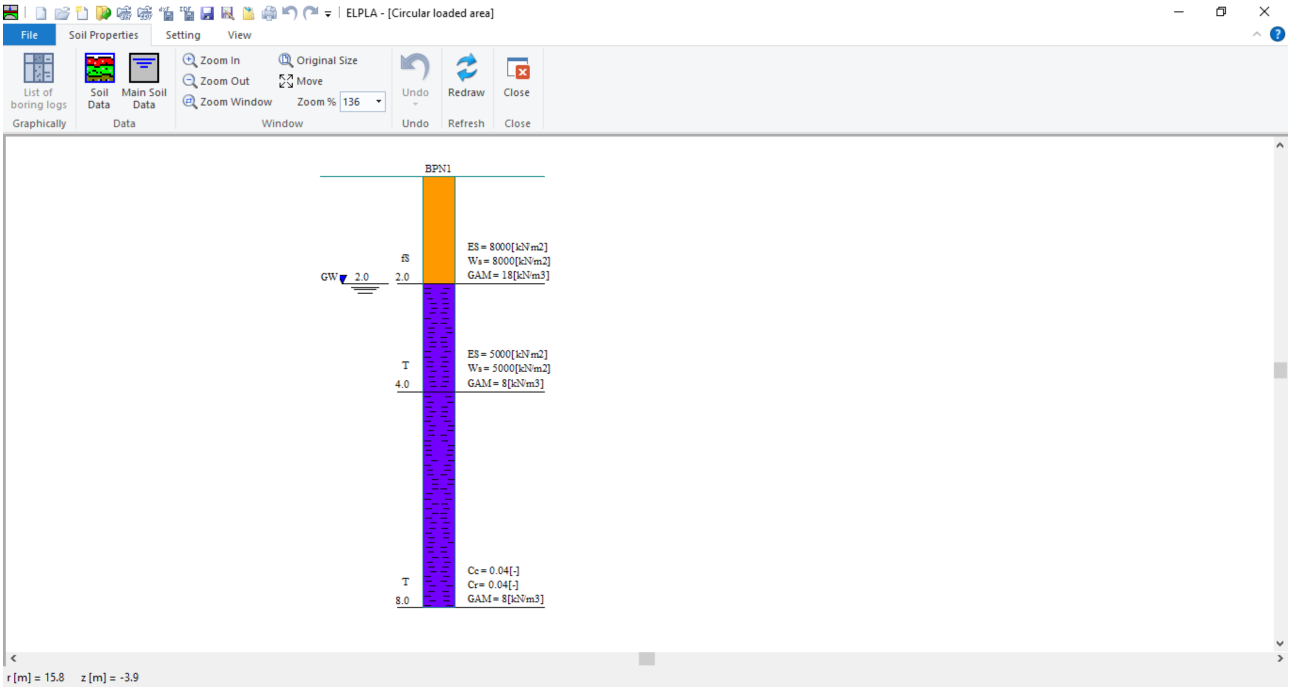


Figure 1.12 boring log

To enter the main soil data for all layers, choose "Main Soil Data" command from "Data" menu in Figure 1.10. The following dialog box in Figure 1.13 appears with default main soil data. The main soil data for the current example, which are required to be defined, are the settlement reduction factor α [-] and the groundwater depth under the ground surface G_w [m]. Any other data corresponding to main soil data are not required for this example. Therefore, the user can use the default values.

In the dialog box of Figure 1.13, enter the settlement reduction factor α [-] and the groundwater depth under the ground surface G_w [m]. Then click "OK" button.

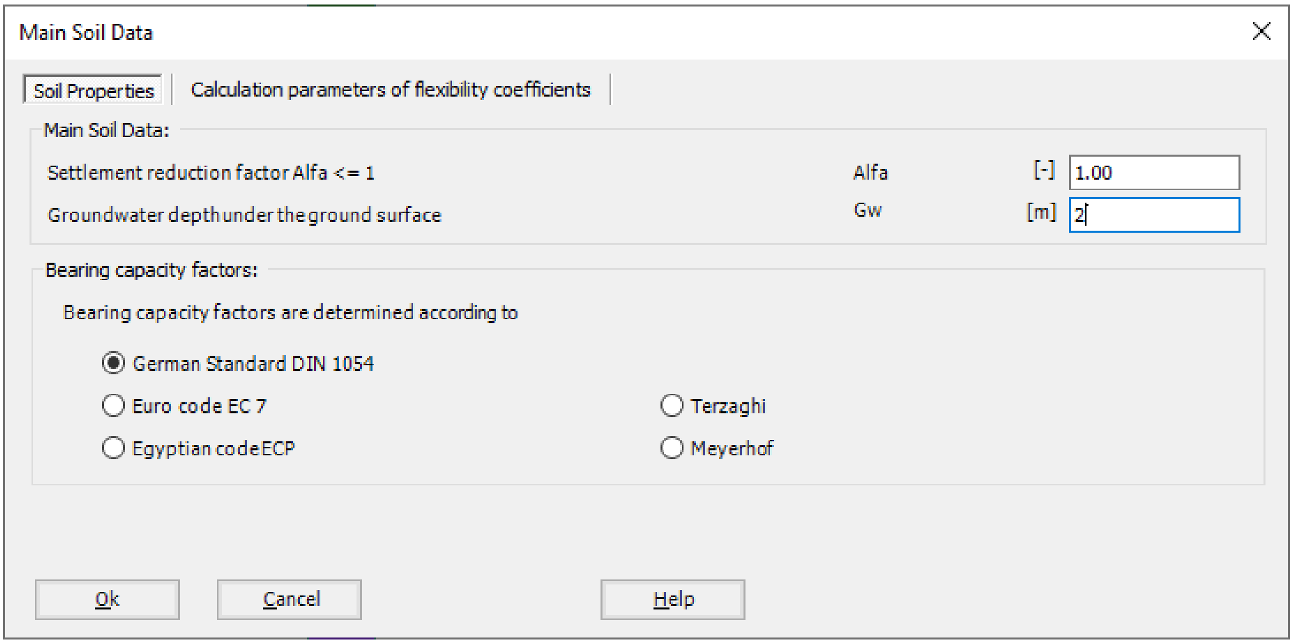


Figure 1.13 "Main Soil Data" dialog box

After entering all data and parameters of the boring log, do the following two steps:

- Choose "Save" command from "File" menu in Figure 1.10 to save the data of boring logs
- Choose "Close" command from "File" menu in Figure 1.10 to close "Soil properties" window and return to *ELPLA* main window

3.5 Shell properties

To define the shell properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 1.14 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, and the unit weight of the shell. Any other data corresponding to shell properties in the program menus are not required for this example.

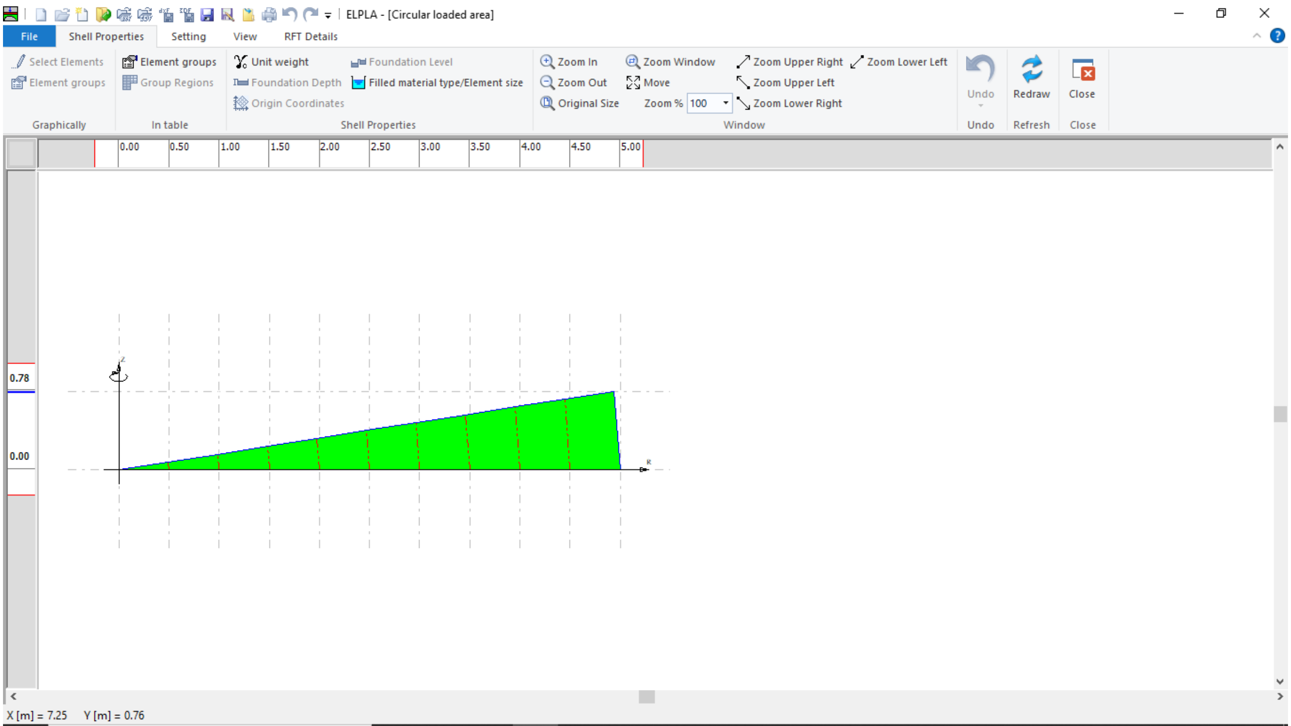


Figure 1.14 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 1.15 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness. Then click "OK" button.

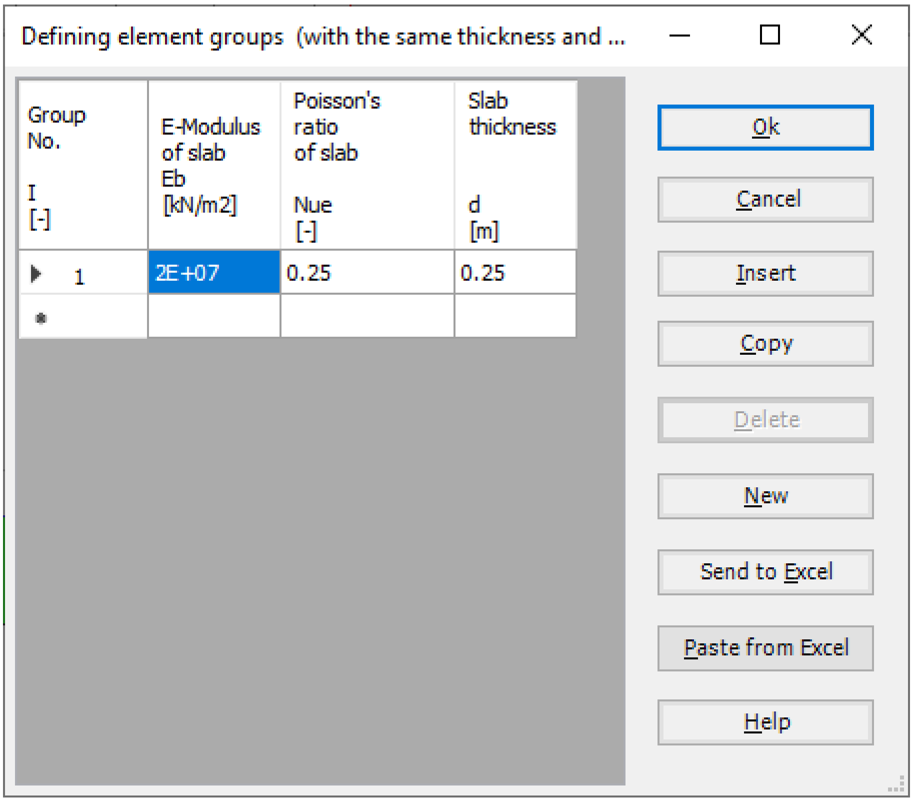


Figure 1.15 "Defining element groups" list box

To enter the unit weight of the shell, choose "Unit weight" command from "Shell Properties" menu in the window of Figure 1.14. The following dialog box in Figure 1.16 with a default unit weight of 25 [kN/m^3] appears. Type 400 in the "Unit weight" edit box, to define the uniform load on the circular area. Then click "OK" button.

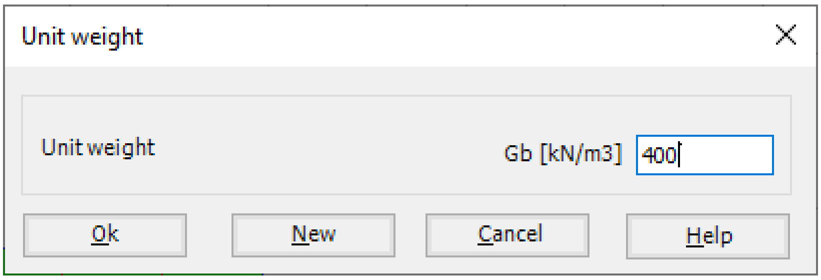


Figure 1.16 "Unit weight" dialog box

After entering the Shell Properties, do the following two steps:

- Choose "Save" command from "File" menu in Figure 1.14 to save the shell properties
- Choose "Close" command from "File" menu in Figure 1.14 to close the "Shell Properties" window and return to *ELPLA* main window

3.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 1.17 appears. In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 1.17. In this example, there is not applied load, as the load has been already defined by the unit weight of the plate.

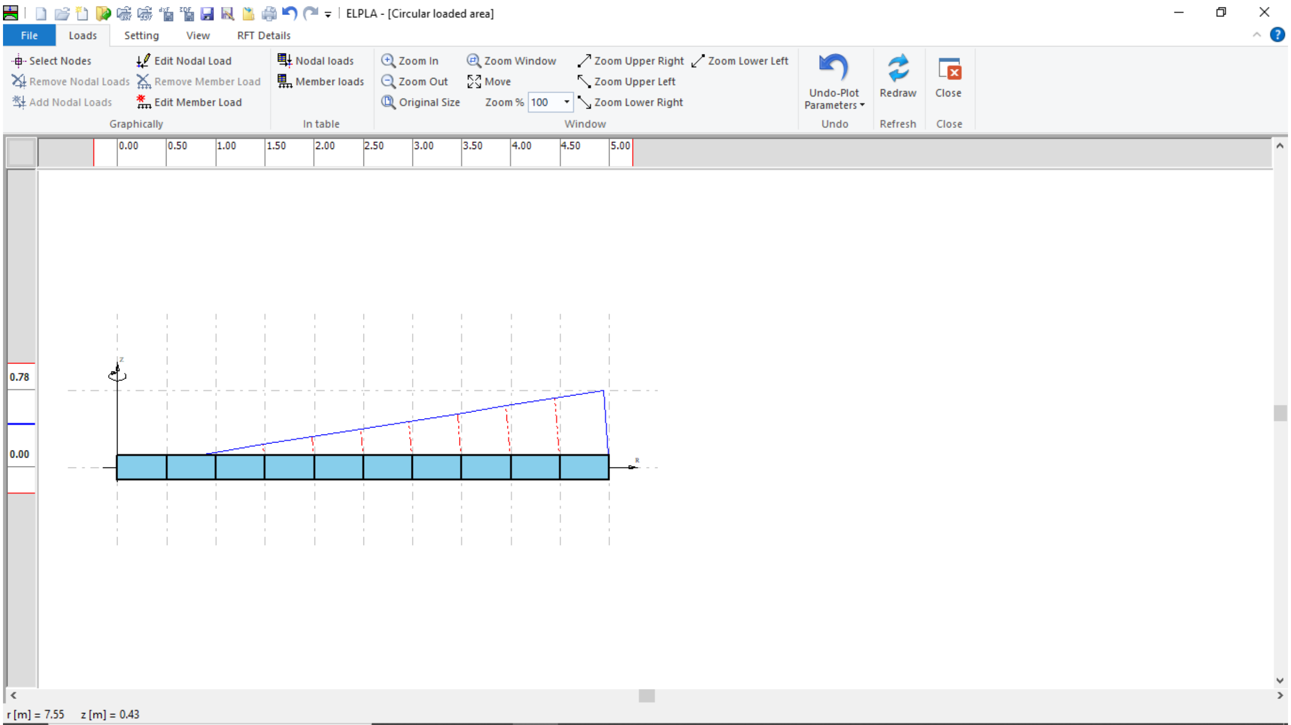


Figure 1.17 "Loads" Window

Do the following two steps:

- Choose "Save" command from "File" menu in Figure 1.17 to save the load data
- Choose "Close" command from "File" menu in Figure 1.17 to close the "Loads" window and return to *ELPLA* main window

Creating the project is now complete. It is time to analyze this project. In the next section, you will learn how to use *ELPLA* for analyzing projects.

4 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 1.18.

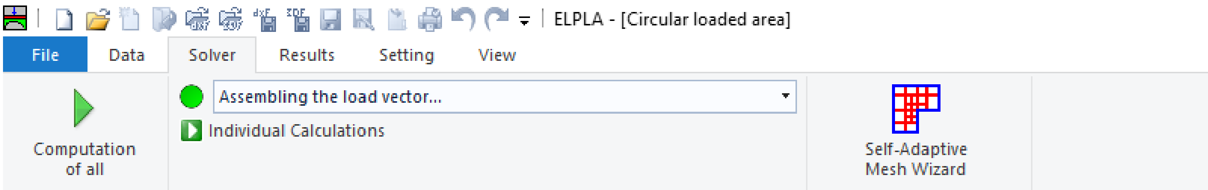


Figure 1.18 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Analysis of the flexible foundation
- Determining deformation, internal forces, contact pressures

These calculation items can be carried out individually or in one time

Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab window.
- The progress of all computations according to the defined method will be carried out automatically with displaying Information through menus and messages.

Analysis progress

Analysis progress menu in Figure 1.19 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

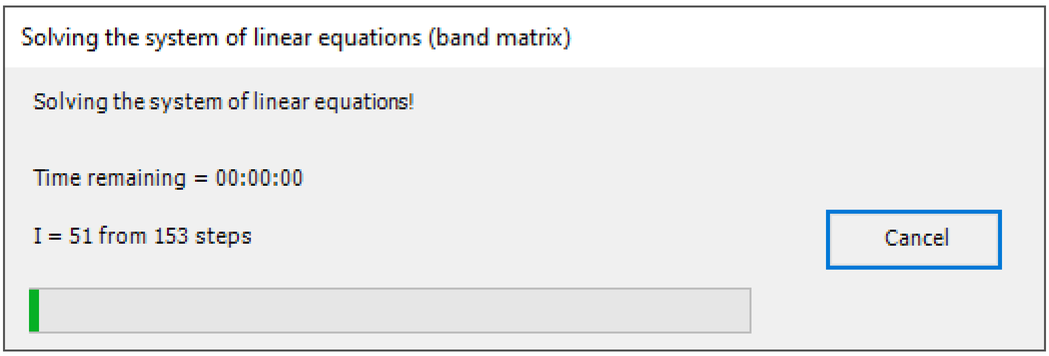


Figure 1.19 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 1.20. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

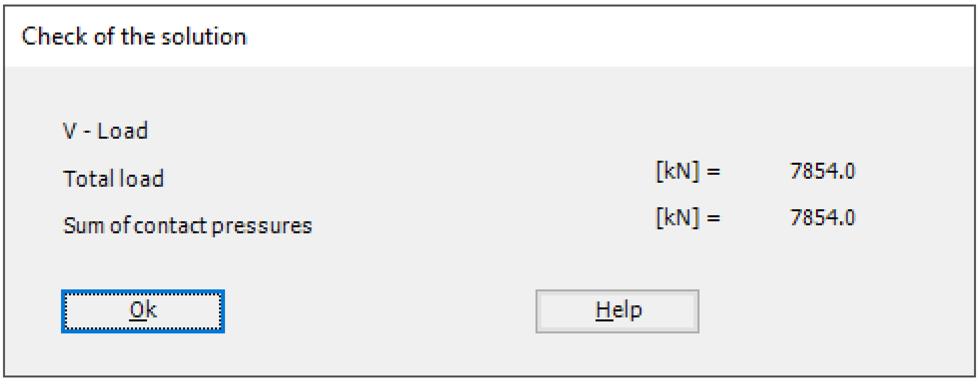


Figure 1.20 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

5 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab.

5.1 Viewing data and result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 1.21).

Figure 1.21 "Results" Tab

The "Result" Tab contains the commands of drawing. These commands depend on the used calculation method in the analysis. For the current example, the commands for presenting the data and results are:

- Data in the plan
- Boring logs and limit depth
- Sections in the shell base

To view sections in shell base

- Choose "Sections in shell base" command from "Section" menu. The following option box in Figure 1.22 appears
- In the "Sections in shell base" option box, select "Base settlements w" as an example for the results to be displayed
- Click "OK" button

The Settlements are now displayed as shown in Figure 1.23.

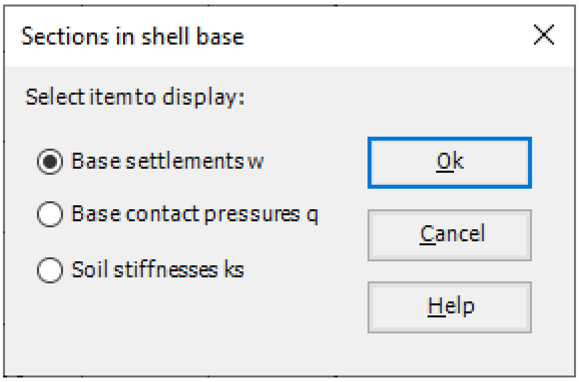


Figure 1.22 "Sections in shell base" option box

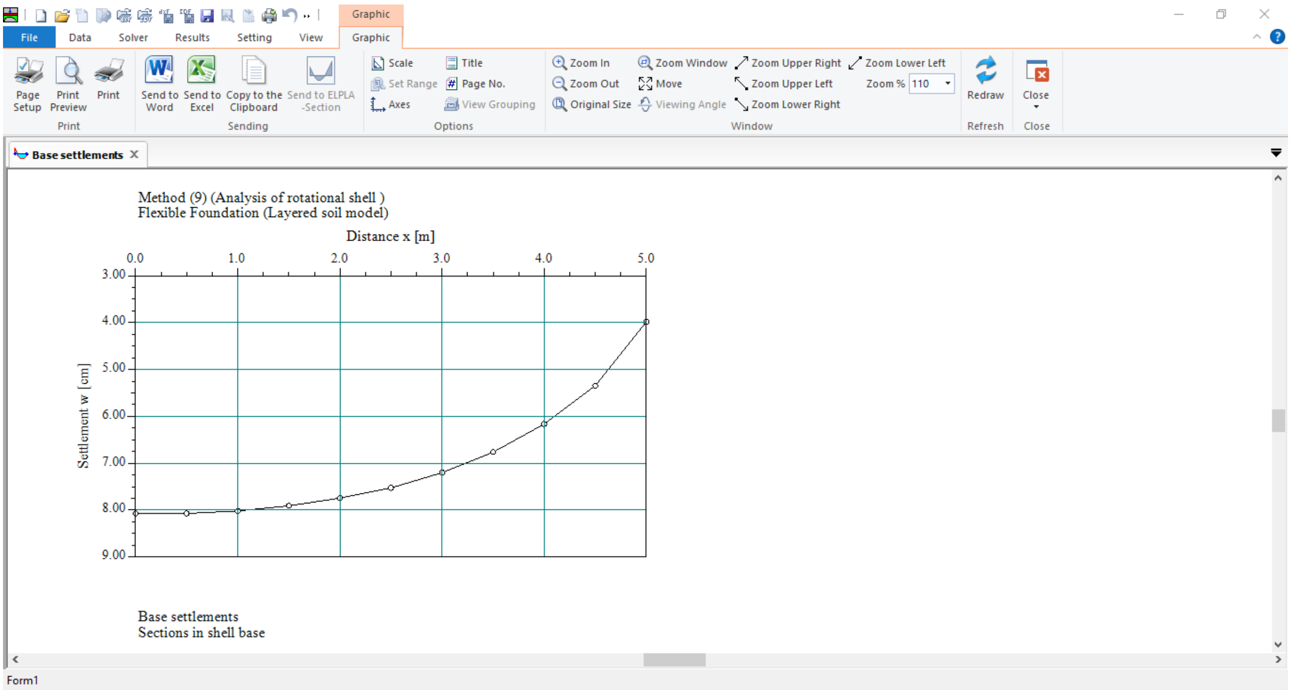


Figure 1.23 Base settlements Sections in shell base

5.2 Listing data and results in tables

The "Result" Tab contains the commands of listing data and results. The commands for listing data and results in tables are:

- Display tables of data
- List tables of data through Text-Editor
- Display tables of results
- List tables of results through Text-Editor

To list results in a table

- Choose "Display tables of Results" command from "List" menu. The following option box in Figure 1.24 appears
- In the "Display Tables of Results" option box, select "Settlements/Contact pressures" as an example for the result to be listed in a table
- Click "OK" button. The loading results are now listed (Figure 1.25)
- Choose "Send to Excel" from "Sending" menu if you wish to export the table to a MS Excel application, Figure 1.26

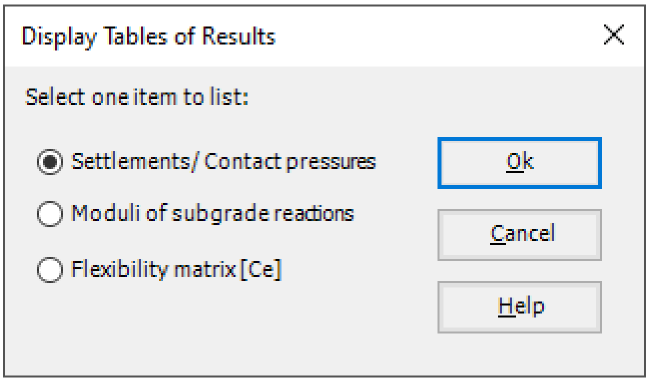


Figure 1.24 "Display Tables of Results" option box

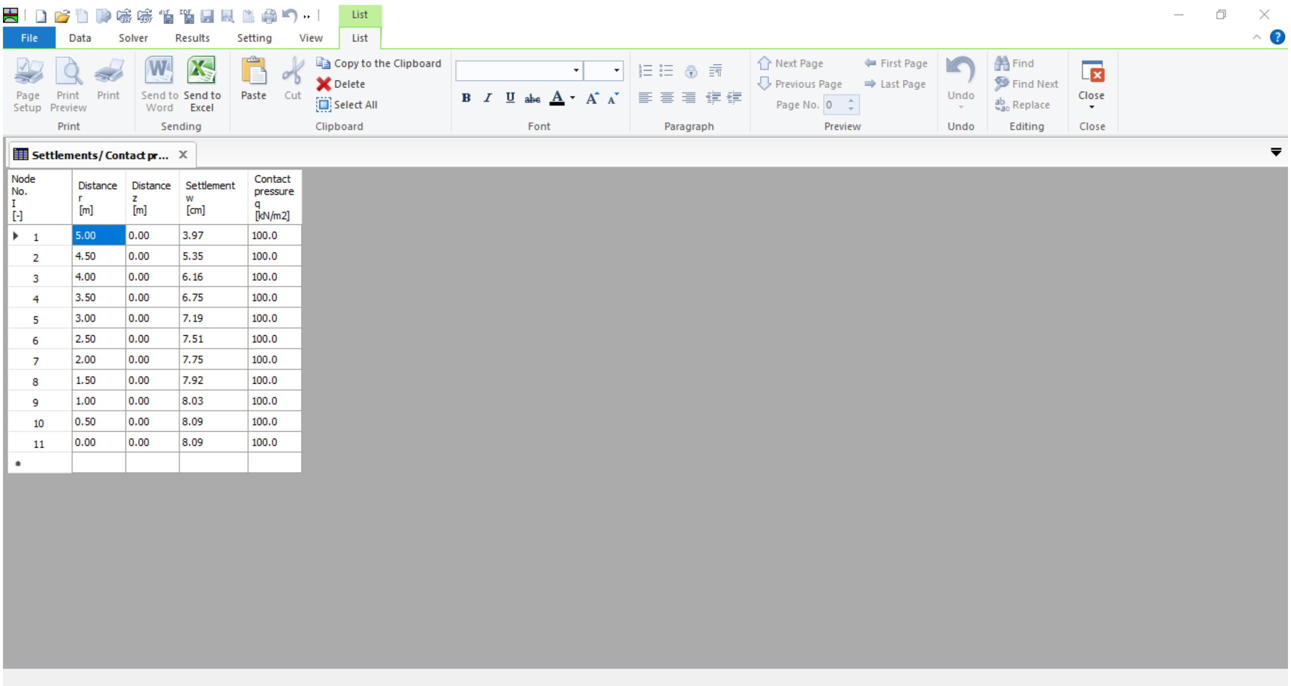
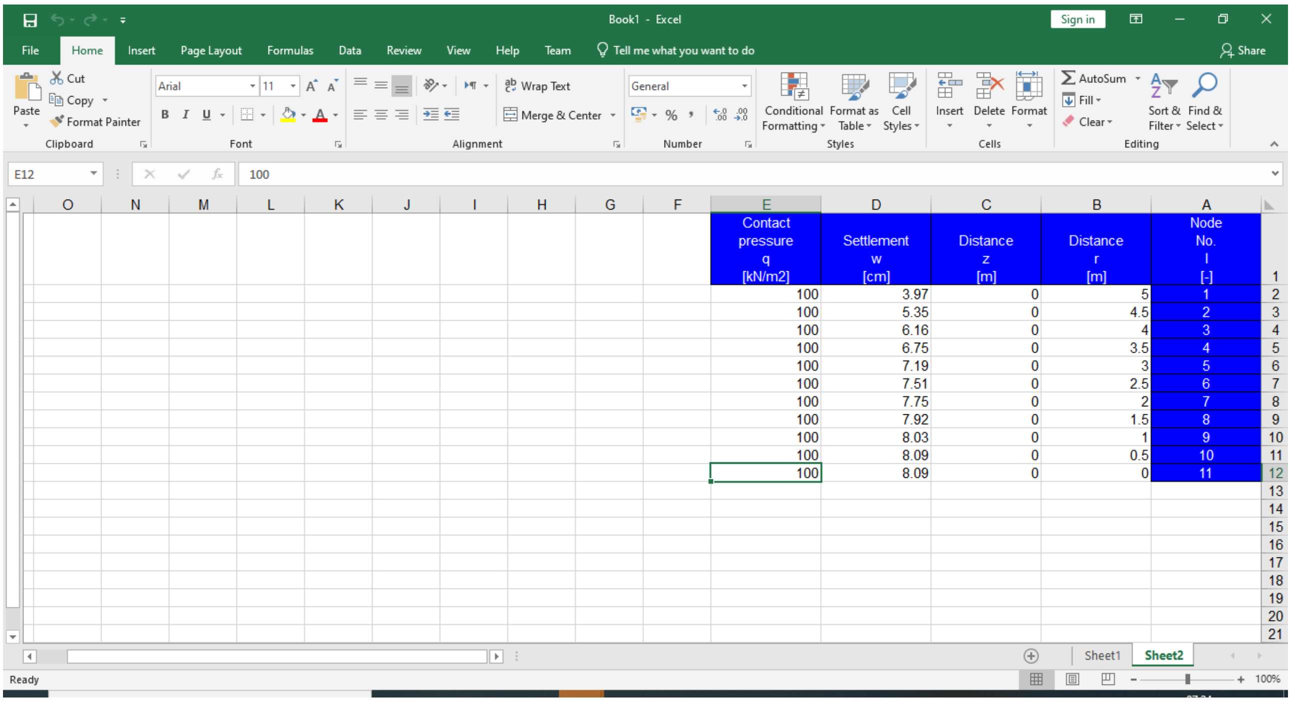


Figure 1.25 List of loading data

Example 1



Contact pressure q [kN/m2]	Settlement w [cm]	Distance z [m]	Distance r [m]	Node No. I [-]
100	3.97	0	5	1
100	5.35	0	4.5	2
100	6.16	0	4	3
100	6.75	0	3.5	4
100	7.19	0	3	5
100	7.51	0	2.5	6
100	7.75	0	2	7
100	7.92	0	1.5	8
100	8.03	0	1	9
100	8.09	0	0.5	10
100	8.09	0	0	11

Figure 1.26 Exported data in MS Word