

Example 12

Analysis of piled raft

Contents		Page
1	Description of the problem	3
1.1	Load and dimensions	3
1.2	Raft material and thickness	3
1.3	Pile material	3
1.4	Soil properties	4
1.5	Mathematical model	4
2	Creating the project	4
2.1	Calculation method	4
2.2	Project identification	8
2.3	FE-Net data	9
2.4	Element length of pile	11
2.5	Piles	12
2.6	Soil properties	15
2.7	Foundation properties	19
2.8	Loads	22
3	Carrying out the calculations	24
3.1	Starting <i>ELPLA-Solver</i>	24
3.2	Carrying out all computations	25
4	Viewing data and results	26
4.1	Viewing result graphics	26
4.2	Plot a diagram of the results	28
4.3	Listing data and result in tables	31
5	Index	35

1 Description of the problem

An example of piled raft is selected to illustrate some of the essential features of *ELPLA* for analyzing piled raft.

1.1 Load and dimensions

A raft on 42 bored piles is considered as shown in Figure 12-1. All piles are equal in diameter and length where pile diameter is 0.9 [m] while pile length is 20 [m]. Spacing between piles is 3.5 [m]. It is required to analyze the raft with piles as piled raft foundation due to a uniform load of $p = 150$ [kN/m] acting on the raft.

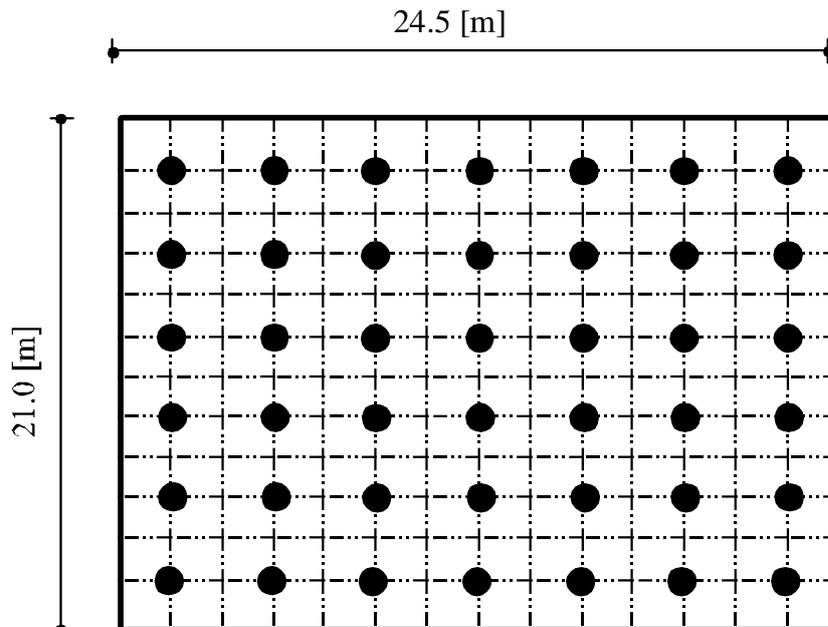


Figure 12-1 Plan with piled raft geometry

1.2 Raft material and thickness

Raft material and thickness are supposed to have the following parameters:

Raft thickness	d	$= 1.5$	[m]
Young's modulus	E_b	$= 3.4 * 10^7$	[kN/m ²]
Unit weight	γ_b	$= 25$	[kN/m ³]
Poisson's ratio	ν_b	$= 0.2$	[-]

1.3 Pile material

The pile material is supposed to have the following parameters:

Young's modulus	E_b	$= 2.35 * 10^7$	[kN/m ²]
Unit weight	γ_b	$= 25$	[kN/m ³]

1.4 Soil properties

The subsoil under the raft consists of a layer of silt with 50 [m] thickness, the water table level is 2 [m] below the ground surface. Foundation level is also 2 [m] below the ground surface. The soil is supposed to have the following parameters:

Modulus of elasticity for loading	E_s	= 10000	[kN/m ²]
Modulus of elasticity for reloading	W_s	= 10000	[kN/m ²]
Unit weight	γ_s	= 18	[kN/m ³]
Poisson's ratio	ν_s	= 0.3	[-]

1.5 Mathematical model

In this example, the Modulus of compressibility method (method 7) is chosen to analyze the piled raft. This Tutorial manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for the soil models and numerical calculation methods are well documented in the User's Guide of *ELPLA*.

2 Creating the project

In this section, the user will learn how to create a project for analyzing a piled raft foundation. The example will be processed step by step 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.

2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation method" wizard appears, Figure 12-2. This wizard will guide you through the steps required to create the project. As shown in Figure 12-2, the first form of the wizard is the "Analysis type" form. In this form, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is piled raft, select "Analysis of piled raft" then click "Next" button to go to the next page.

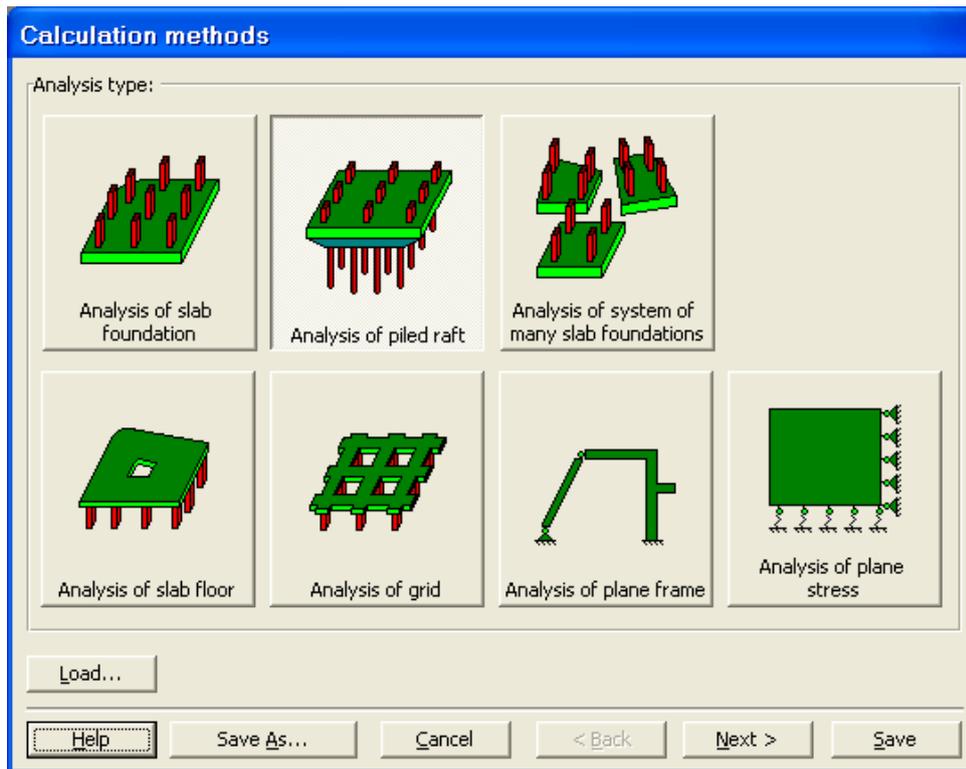


Figure 12-2 "Calculation method" wizard with "Analysis type" form

After clicking "Next" button, the "Calculation methods" form appears, Figure 12-3. To define the calculation method

- Select the calculation method "7-Modulus of compressibility (Elimination)"
- Click "Next" button to go to the next form

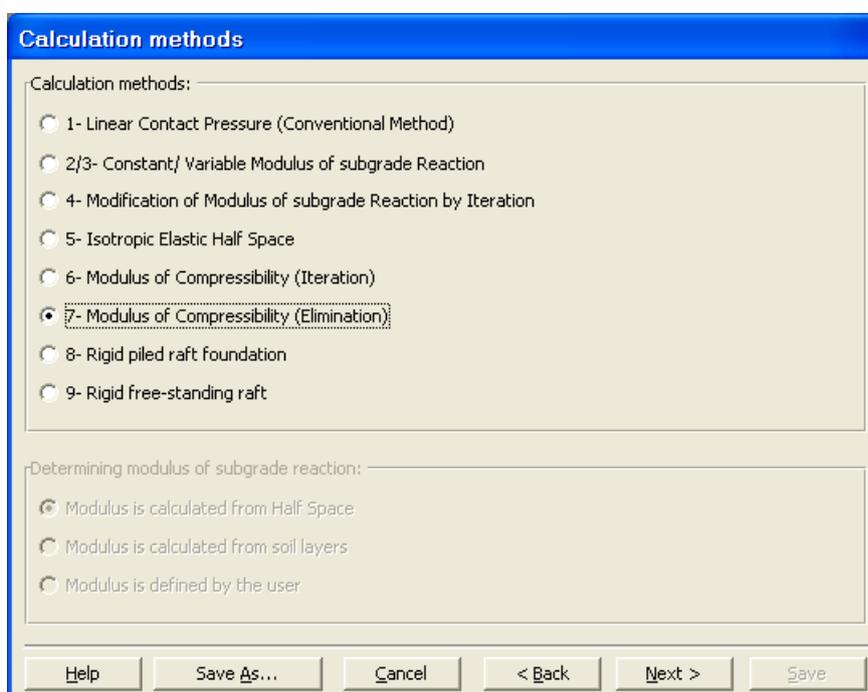


Figure 12-3 "Calculation methods" form

The next form is the "System symmetry" (Figure 12-4). In this form choose "Unsymmetrical system" then click "Next" button.

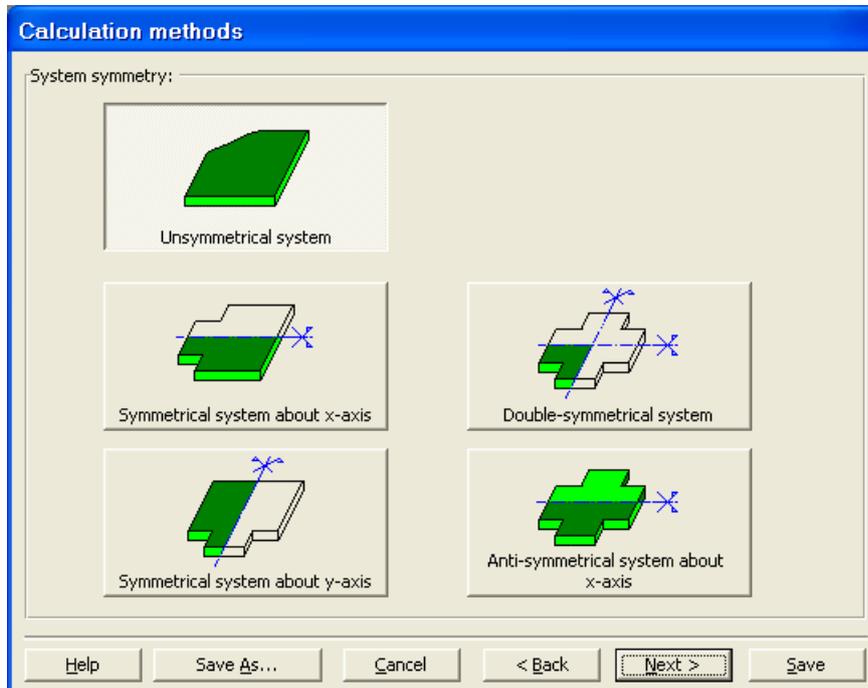


Figure 12-4 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 12-5. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other. There isn't any requested choices, so click the "Save" button.

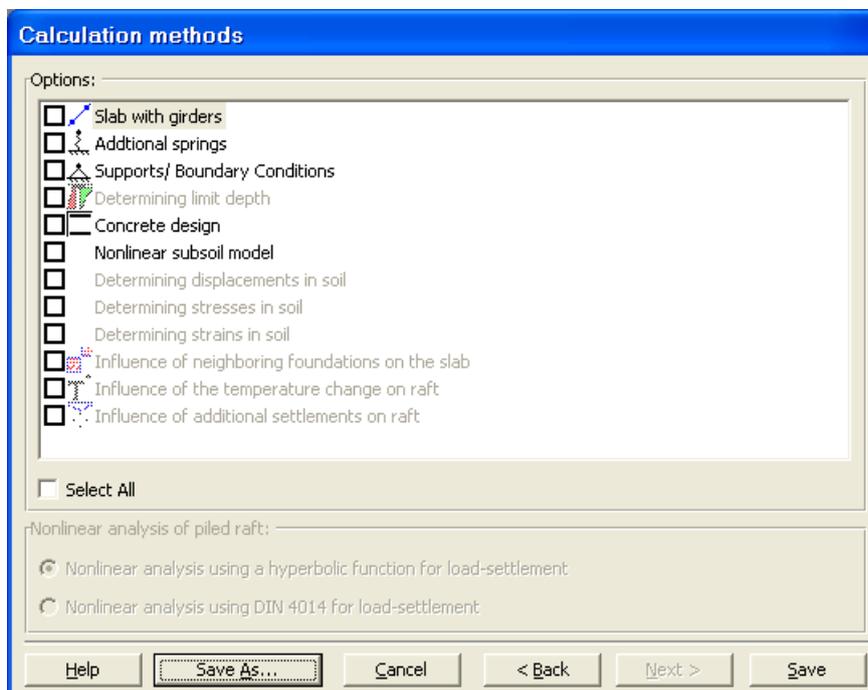


Figure 12-5 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 12-6.

In this dialog box

- Type a file name for the current project in the "File name" edit box. For example type "Piled raft". *ELPLA* will use automatically this file name in all reading and writing processes
- Click "Save" button

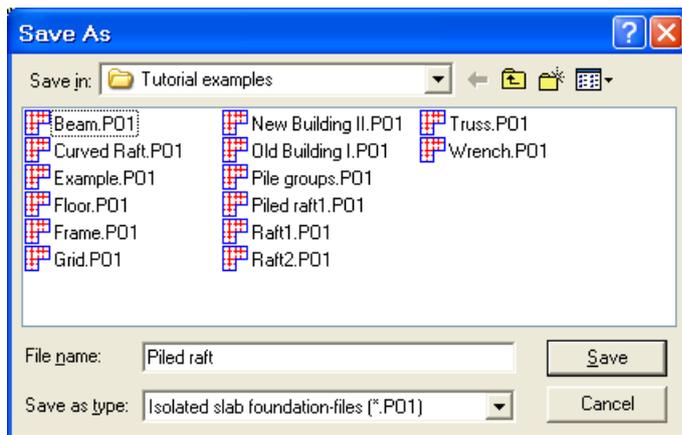


Figure 12-6 "Save as" dialog box

ELPLA will activate the "Data" menu. In addition, the file name of the current project [Piled raft] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 12-7. In the "Data" menu, the user can enter the remaining data of the project using the same sequence of commands in this menu. The first command in the menu is "Calculation methods", which has been already entered. Therefore, *ELPLA* has put the sign "√" beside this command, Figure 12-7. *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.

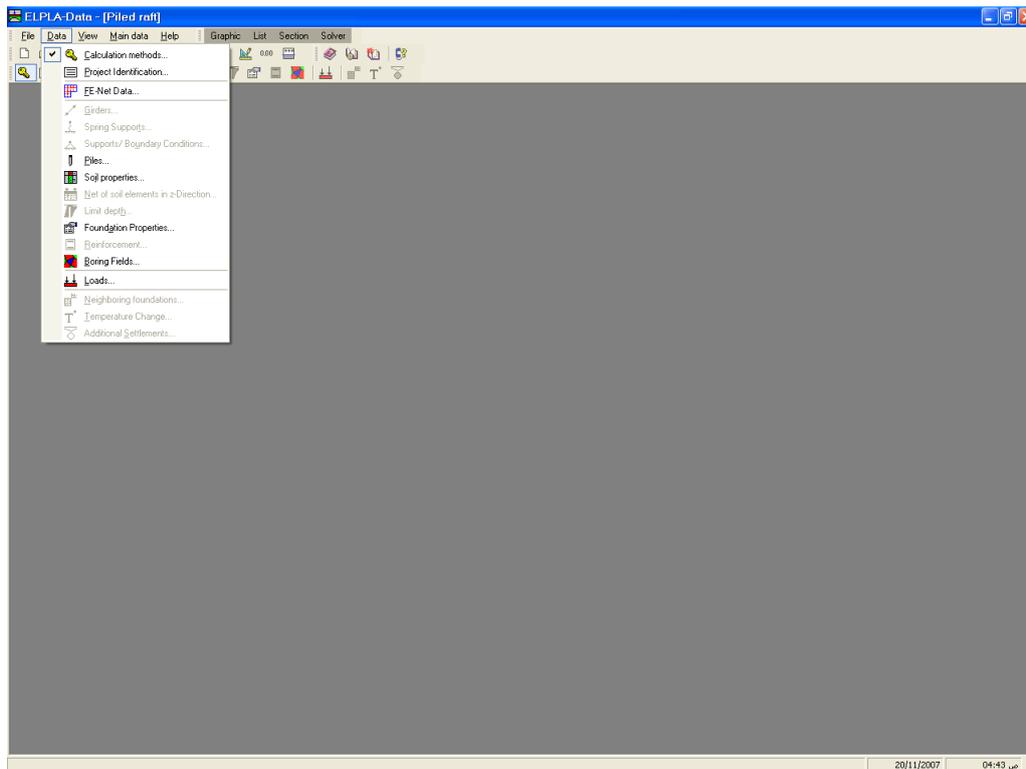


Figure 12-7 *ELPLA-Data* after defining the calculation method

2.2 Project identification

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 12-8 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box:
"Analysis of piled raft"
- Type the date of the project in the "Date" edit box
- Type "Tutorial Manual" in the "Project" edit box
- Click "Save" button

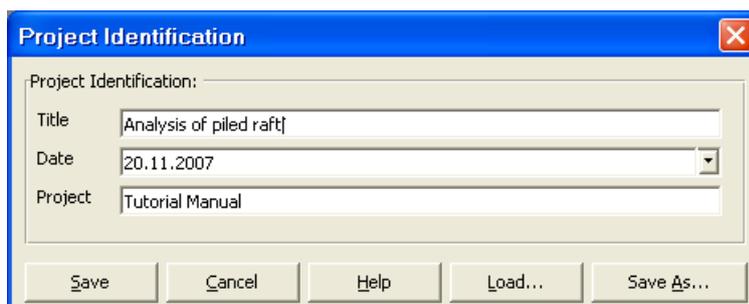


Figure 12-8 "Project identification" dialog box

2.3 FE-Net data

Choose "FE-Net data" command from "Data" menu of *ELPLA-Data*. The "FE-Net generation" wizard appears as shown in Figure 12-9. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 12-9 the first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have regular shapes. For the given problem, the foundation has a rectangular shape.

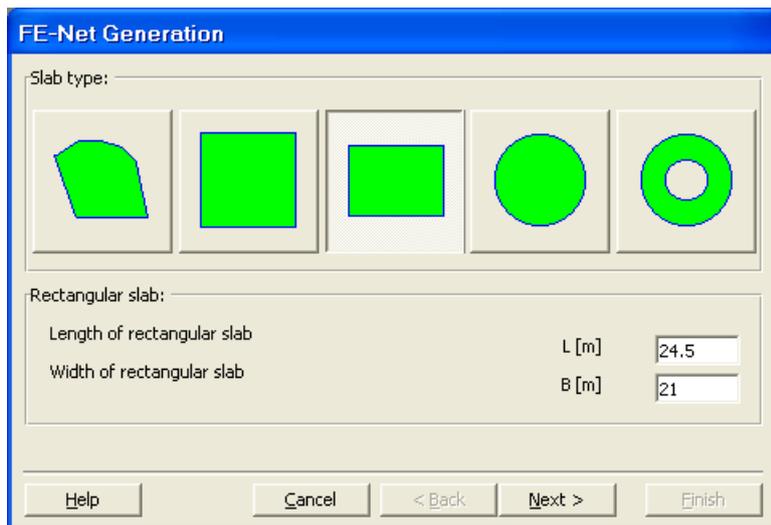


Figure 12-9 "FE-Net generation" wizard with "Slab type" form

To generate the FE-Net

- In the "Slab type" options, choose the rectangular slab option
- In the "Rectangular slab" frame, enter the total length and width of the raft in the corresponding edit boxes
- Click "Next" button to go to the next form

After clicking "Next" in "FE-Net generation" wizard, the following "Generation type" form appears, Figure 12-10. *ELPLA* can deal with various type of generations with triangle and/ or rectangular elements. Choose the first type of rectangular elements. Then click "Next" button to go to the next form.

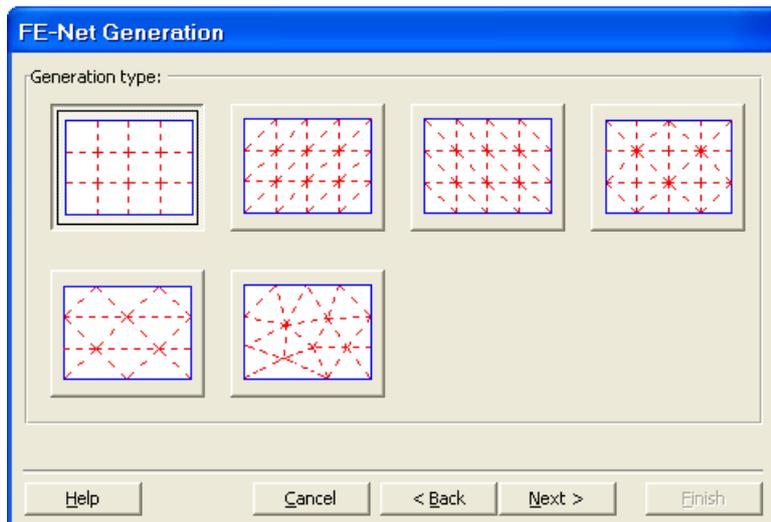


Figure 12-10 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "Grid definition" dialog box in Figure 12-11 appears with default values of constant element size.

In this dialog box

- In "Grid in x -direction" frame, type 14 in the "No. of grid spaces" edit box
- In "Grid in y -direction" frame, type 12 in the "No. of grid spaces" edit box
- Type 1.75 in the "Grid interval" edit box
- Click "OK" button

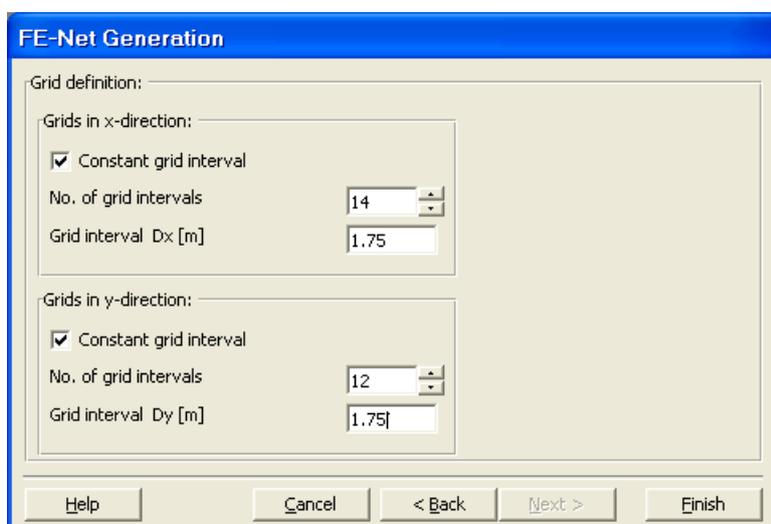


Figure 12-11 "Grid definition" form

ELPLA will generate a FE-Net for a rectangular area of 24.5 [m] length and 21 [m] width with square elements of 1.75 [m] each side. The following embedded program in Figure 12-12 appears with the generated net.

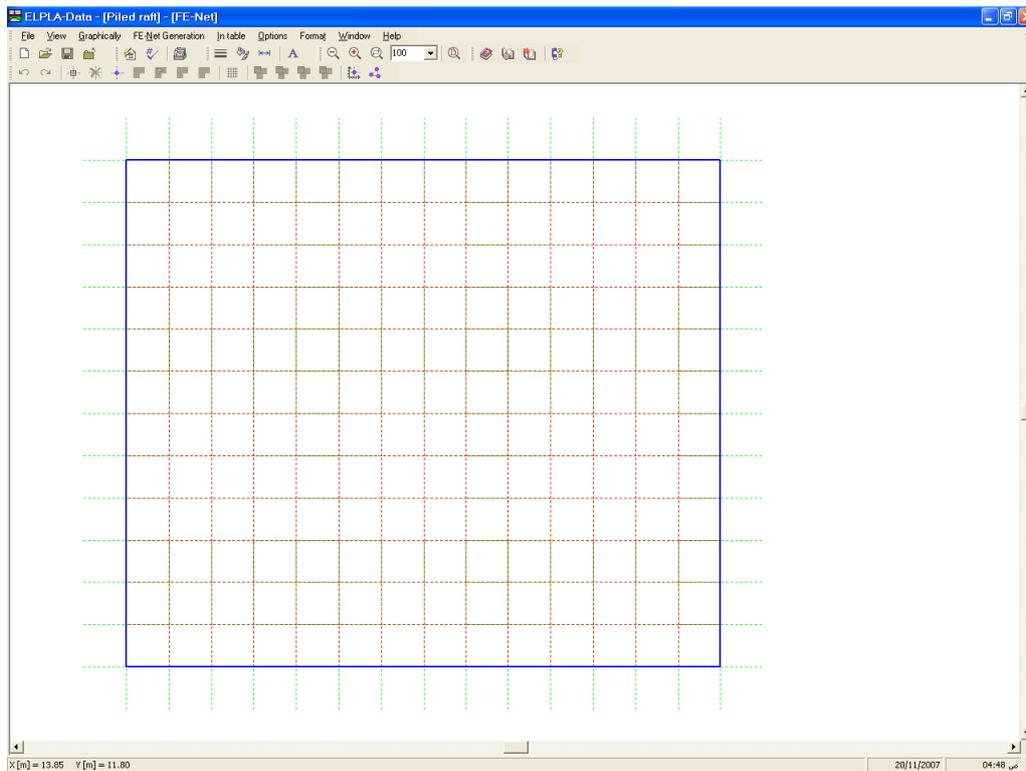


Figure 12-12 FE-Net of a rectangular slab on the screen

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

- Choose "Save FE-Net" command from "File" menu in Figure 12-12 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 12-12 to close the "FE-Net" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " \sqrt " is typed automatically beside the "FE-Net data" command in the "Data" menu of *ELPLA-Data*.

2.4 Element length of pile

To define the element length of pile, choose "Preferences" command from "Main data" menu of *ELPLA-Data*. The "Preferences" dialog box in Figure 12-13 appears.

In this dialog box

- Type 2 in the "Element length of pile" edit box
- Click "Save" button

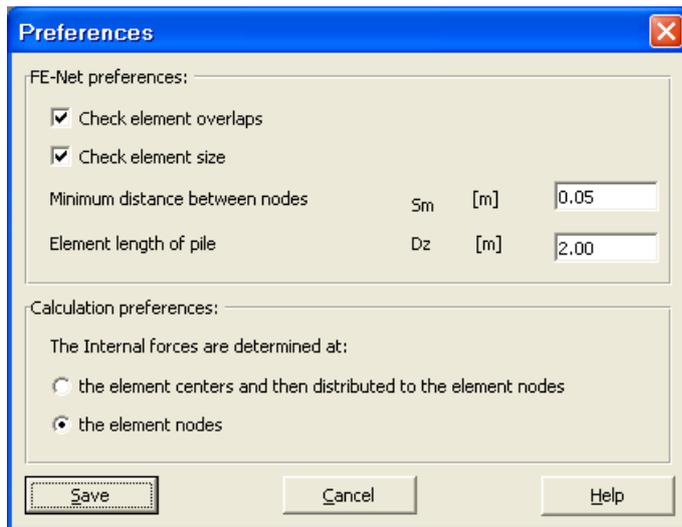


Figure 12-13 "Preferences" dialog box

2.5 Piles

To define piles, choose "Piles" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 12-14 appears.

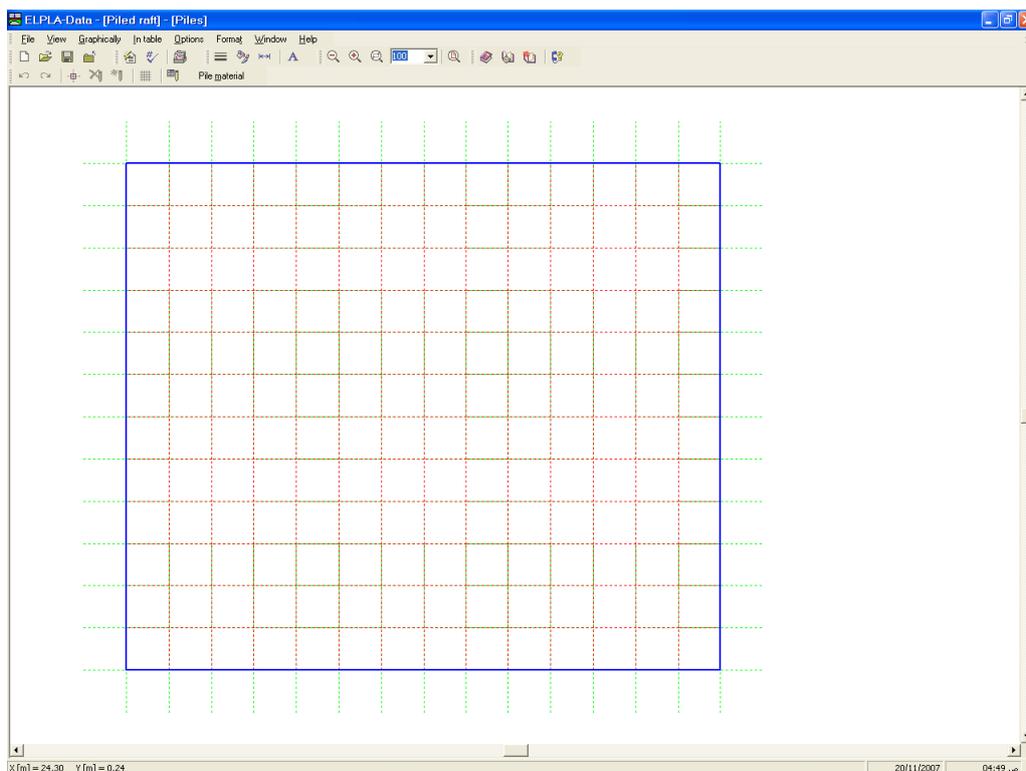


Figure 12-14 "Piles" embedded program

Pile groups

To define pile groups that have the same diameter and length, choose "Pile groups" command from "In Table" menu in Figure 12-14. When the "Pile" groups" command is chosen, the following table in Figure 12-15 appears allowing you to define the pile diameter and length.

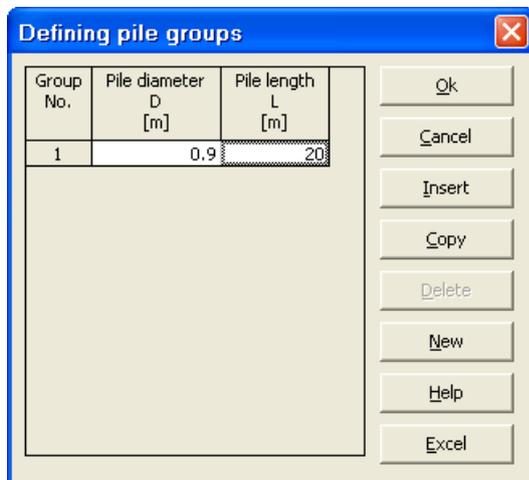


Figure 12-15 Defining pile groups

Pile material

To define pile material, choose "Pile material" command from "In Table" menu in Figure 12-14. When the "Pile material" command is chosen, the following dialog box in Figure 12-16 appears.

In this dialog box

- Type 25 in the "Unit weight of the pile concrete" edit box
- Type 2.35E+07 in the "Modulus of elasticity of pile" edit box
- Click "OK" button

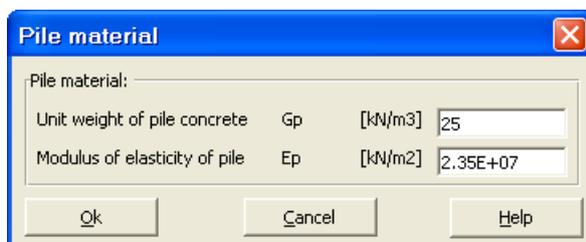


Figure 12-16 "Defining element groups" dialog box

File location

To define pile locations on the net

- Choose "Select nodes" command from "Graphically" menu in Figure 12-14
- Select nodes those have piles as shown in Figure 12-17
- Choose "Add piles" command from "Graphically" menu in Figure 12-17. The "Defining pile groups" dialog box in Figure 12-18 appears. In this dialog box click "OK" button

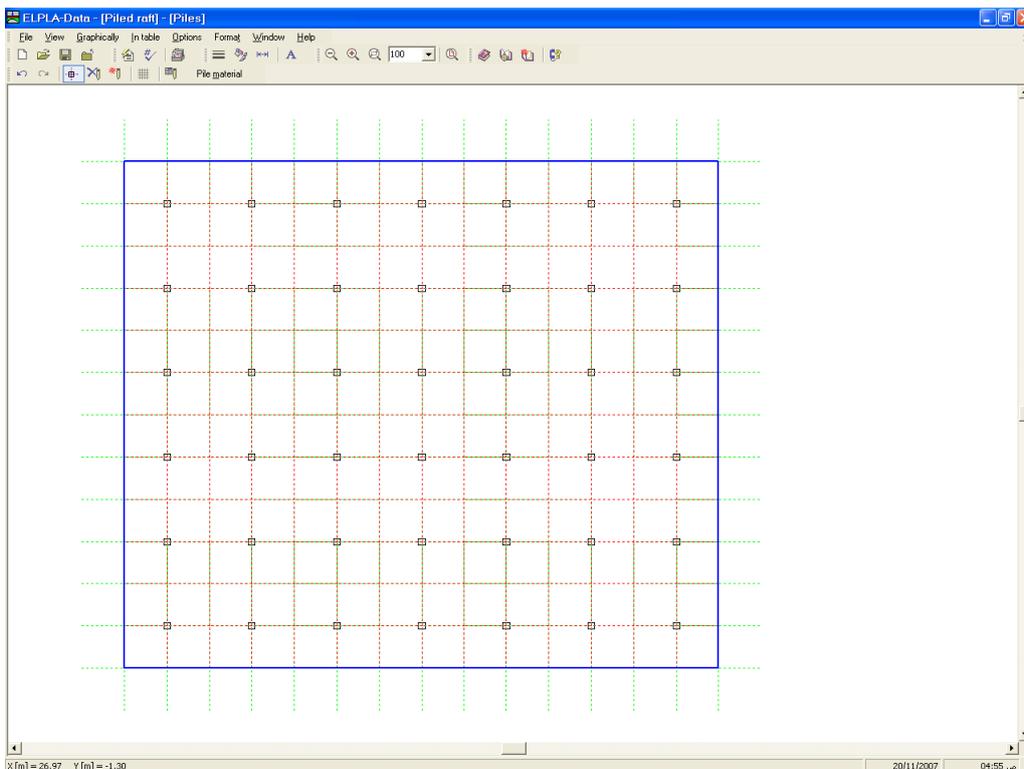


Figure 12-17 Selection of nodes that have piles

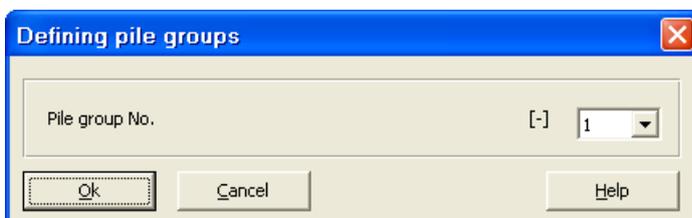


Figure 12-18 "Defining pile groups" dialog box

After you have completed the definition of the piles, the screen should look like the following Figure 12-19.

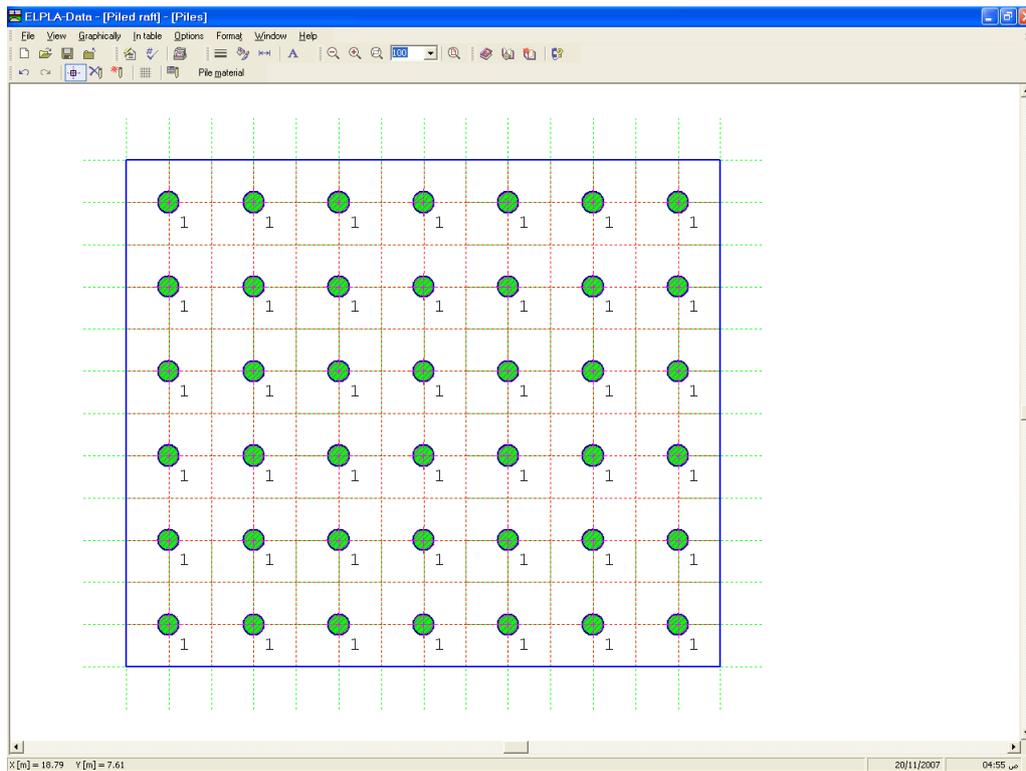


Figure 12-19 Piles on the screen

After entering all data of piles, do the following two steps:

- Choose "Save piles" command from "File" menu in Figure 12-19 to save the data of piles
- Choose "Close piles" command from "File" menu in Figure 12-19 to close the *ELPLA-Piles* sub program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Piles" command in the "Data" menu of *ELPLA-Data*.

2.6 Soil properties

To define the soil properties, choose "Soil properties" command from "Data" menu of *ELPLA-Data*. The following sub program in Figure 12-20 appears with a default-boring log.

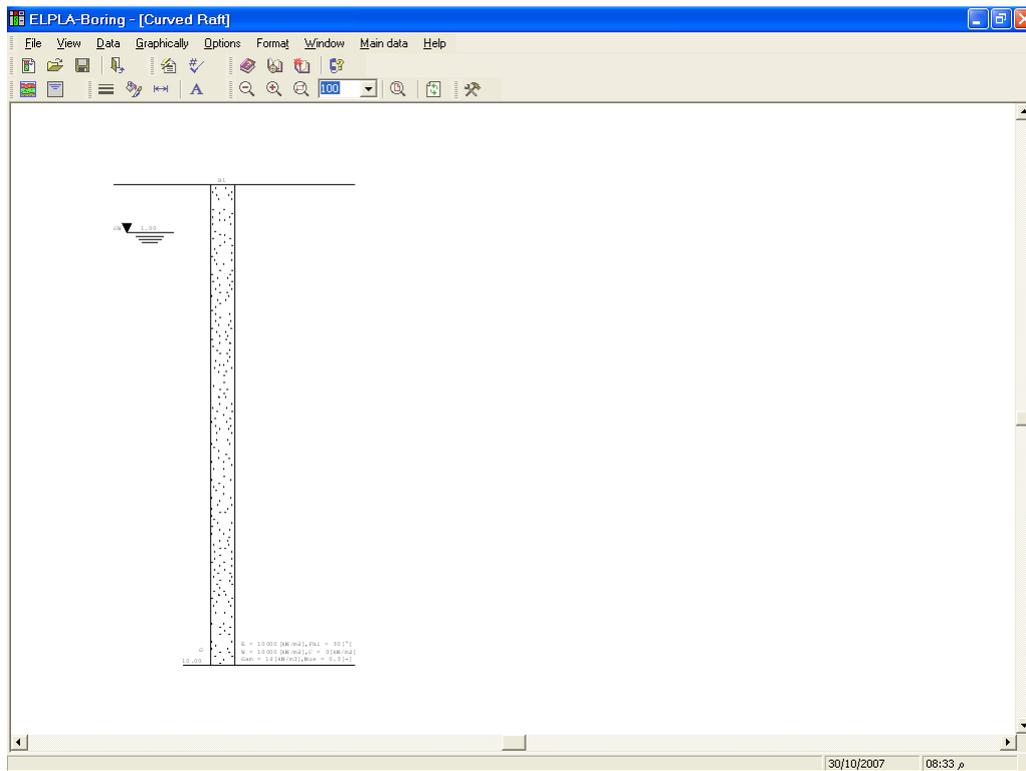


Figure 12-20 *ELPLA-Boring* sub program with a default-boring log

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 12-20. The following dialog box in Figure 12-21 appears with default-boring log data.

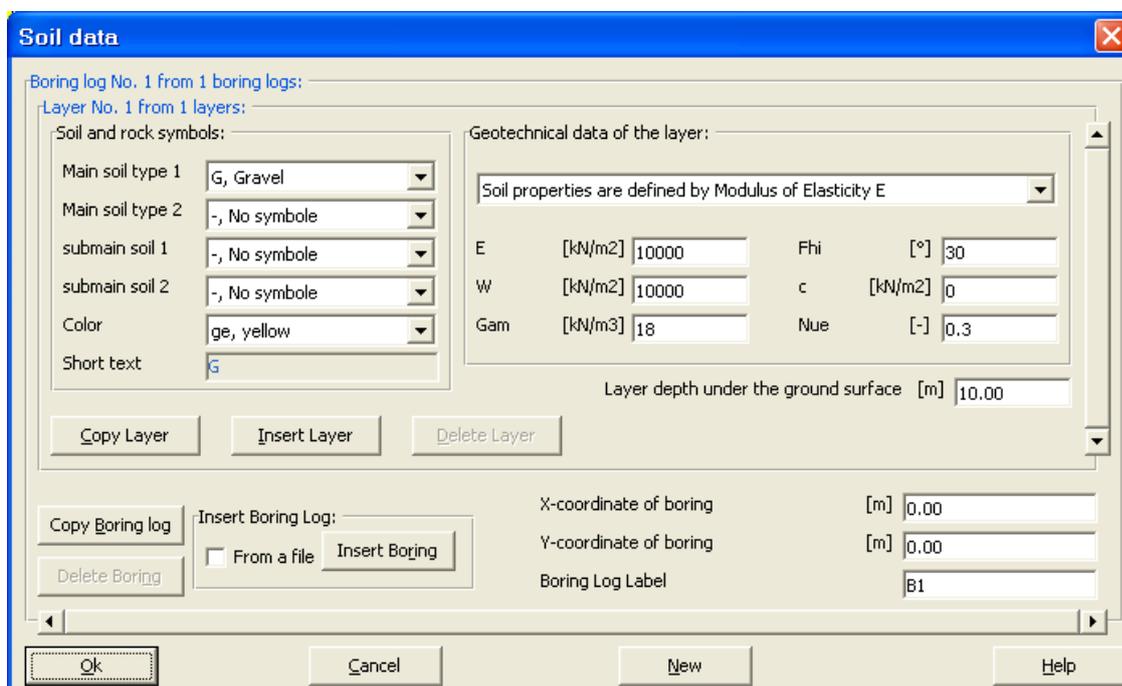


Figure 12-21 "Soil data" dialog box with default-boring log data

In the "Geotechnical data of the layer" dialog group box in Figure 12-21, define the geotechnical data of the first soil layer of the boring log as follows:

$$\begin{aligned} E_s &= 10000 && [\text{kN/m}^2] \\ W_s &= 10000 && [\text{kN/m}^2] \\ \gamma_s &= 18 && [\text{kN/m}^3] \\ \nu_s &= 0.3 && [-] \end{aligned}$$

In the current example, the angle of internal friction φ and the cohesion c of the soil are not required because the selected type of the analysis is linear analysis. Therefore, the user can let the default values of the internal friction and the cohesion.

$$\begin{aligned} \varphi_s &= 30 && [^\circ] \\ c &= 0 && [\text{kN/m}^2] \end{aligned}$$

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 "U, Silt" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box, Figure 12-21. The color of the silt according to the German Standard DIN 4023 will be automatically created. The user can change this color. Also, a short text "U" will be automatically created for the silt.

To enter the second layer

- Click "Layer copy" button in Figure 12-21. 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/m³] to 9 [kN/m³]
- Change the value of the layer depth under the ground surface from 2 [m] to 50 [m]

Note that the unit weight of the soil is used to determine the overburden pressure q_v [kN/m²] due to the removed soil, which is equal to $\gamma_s * d_f$. This means that the unit weight of the soil under the foundation depth d_f is not required. However, the unit weight of the soil under the foundation depth for all soil layers is entered by the value 9 [kN/m³].

After editing the geotechnical data, the boring coordinates and labels which describe the boring will be entered.

To enter the boring coordinates and label

- Type 0 for x -coordinate in " x -coordinate of boring log [m]" edit box
- Type 0 for y -coordinate in " y -coordinate of boring log [m]" edit box
- Type B1 as a label name for the first boring in "Label of boring log" edit box

To enter the main soil data for all layers, choose "Main soil data" command from "Data" menu in Figure 12-20. The following dialog box in Figure 12-22 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 in this example. Therefore, the user can take these data from the default soil properties.

In the dialog box of Figure 12-22, enter the settlement reduction factor α [-] and the groundwater depth under the ground surface G_w [m]. Then click "OK" button.

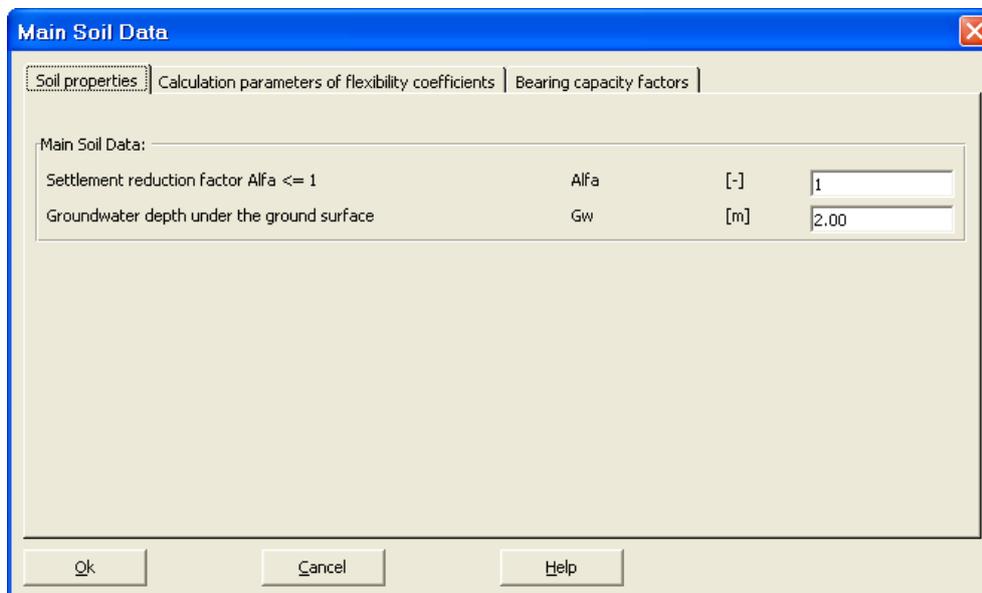


Figure 12-22 "Main soil data" dialog box

Now all data and parameters for the boring log have been entered. After finishing the creation of the boring log, click "OK" button in "Soil data" dialog box in Figure 12-21 to see the defined boring log on the screen where the user can control or modify the input data and parameters, Figure 12-23.

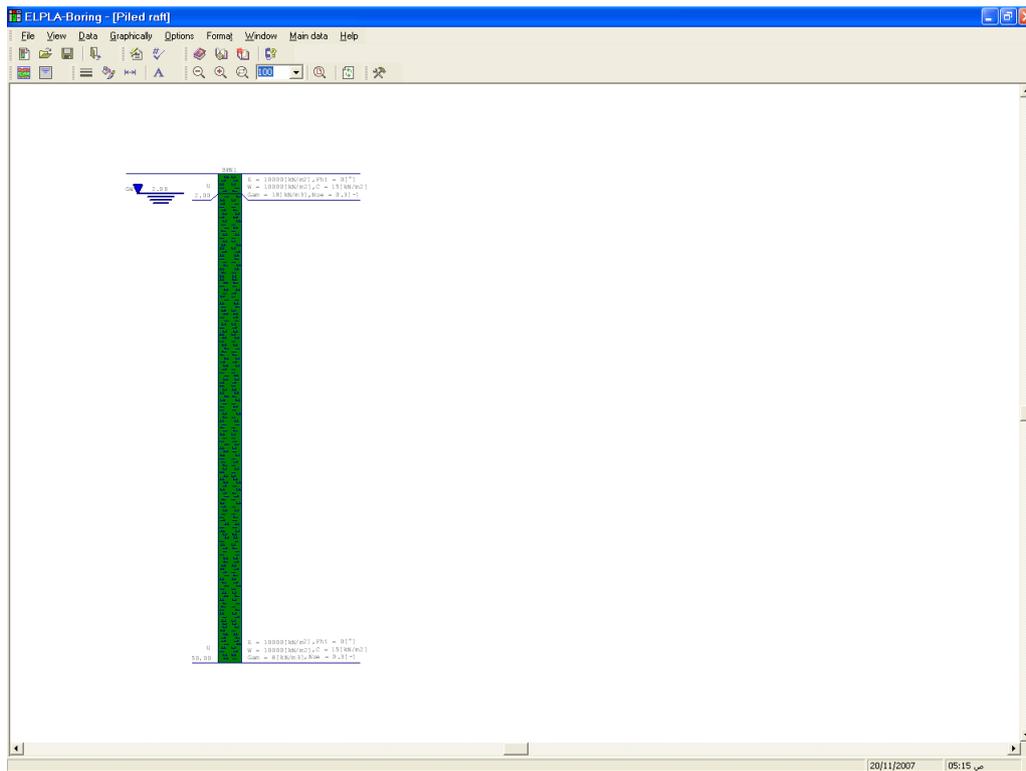


Figure 12-23 Boring log on the screen

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

- Choose "Save boring logs" command from "File" menu in Figure 12-23 to save the data of boring log
- Choose "Close boring logs" command from "File" menu in Figure 12-23 to close the *ELPLA-Boring* sub program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{\quad}$ " is typed automatically beside the "Soil properties" command in the "Data" menu of *ELPLA-Data*.

2.7 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 12-24 appears with default foundation properties. The data of foundation properties for the current example, which are required to define, are foundation material and foundation thickness. Any other data corresponding to foundation properties in the program menus are not required. Therefore, the user can take these data from the default foundation properties.

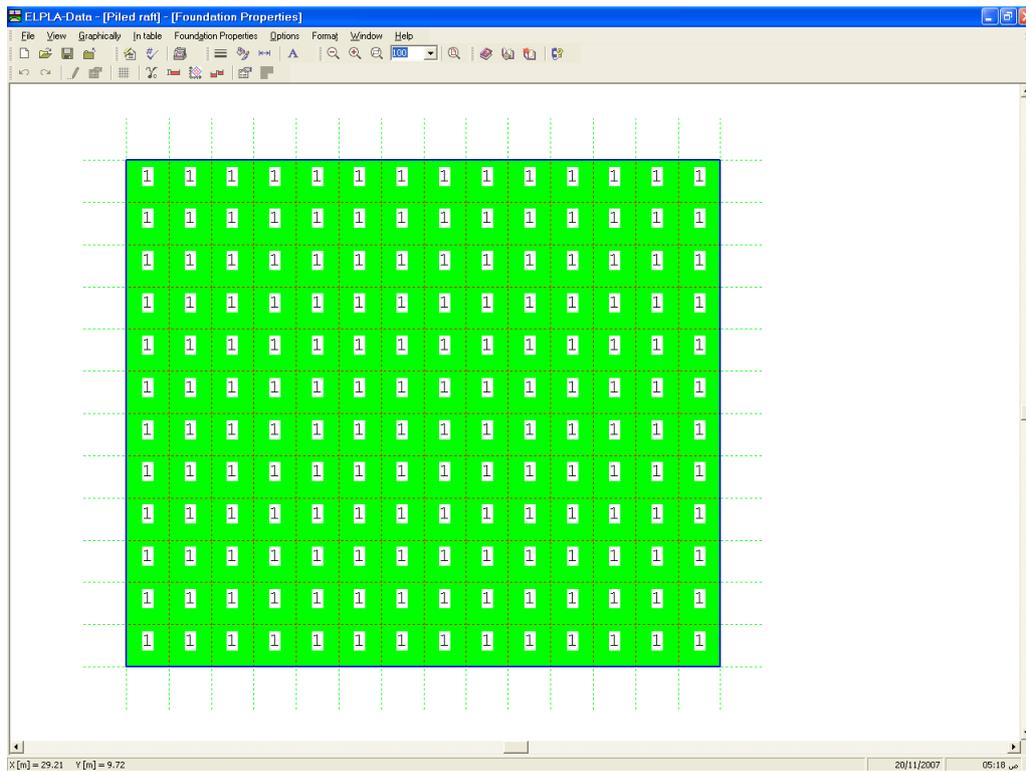


Figure 12-24 "Foundation properties" embedded program

To enter the foundation material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 12-24. The following list box in Figure 12-25 with default data appears. To enter or modify a value in this list box, type that value in the corresponding cell then press "Enter" key. In the list box of Figure 12-25, enter E-Modulus of the foundation, *Poisson's* ratio of the foundation and foundation thickness
- Click "OK" button

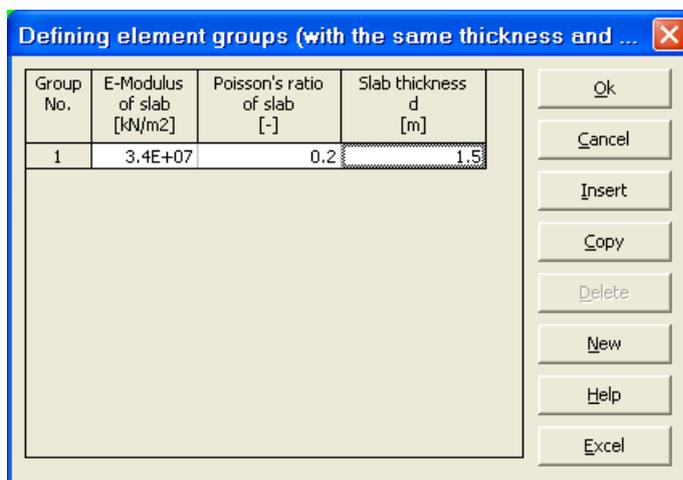


Figure 12-25 "Defining element groups" list box

To enter the unit weight of the foundation

- Choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 12-24. The following dialog box in Figure 12-26 with a default unit weight of 25 [kN/m³] appears. Let the default value as written in the edit box "Unit weight of the foundation"
- Click "OK" button



Figure 12-26 "Unit weight of the foundation" dialog box

To enter the foundation level

- Choose "Foundation depth" command from "Foundation properties" menu in the window of Figure 12-24. The following dialog box in Figure 12-27 appears
- In this dialog box type 2 in the "Foundation depth under ground surface (a)/ (b)" edit box
- Click "OK" button

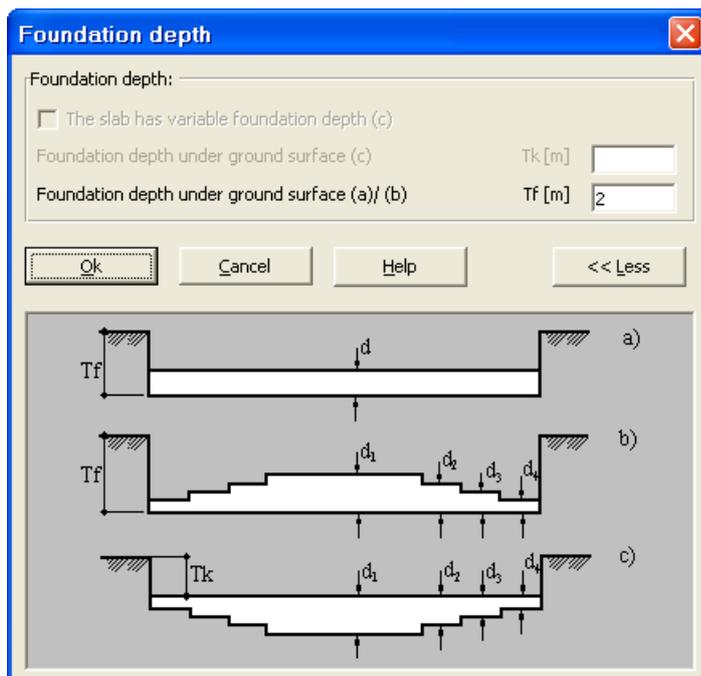


Figure 12-27 "Foundation depth" dialog box

After entering the foundation properties, do the following two steps:

- Choose "Save foundation properties" command from "File" menu in Figure 12-24 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 12-24 to close the "Foundation properties" embedded program and to return to the main window of *ELPLA-Data*

Note that the sign " \sqrt " is typed automatically beside the "Foundation properties" command in the "Data" menu of *ELPLA-Data*.

2.8 Loads

To define the loads, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 12-28 appears.

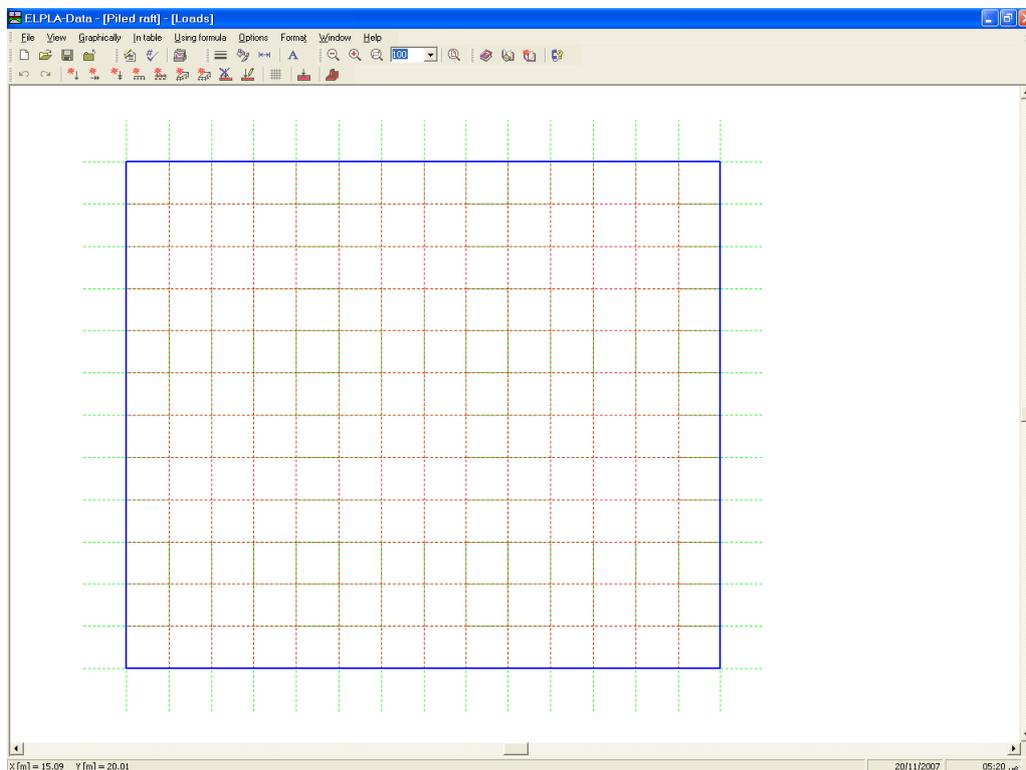


Figure 12-28 "Loads" embedded program

To enter loads

- Choose "Distributed load (rectangular)" command from "Graphically" menu in the window of Figure 12-28. When "Distributed loads (rectangular)" command is chosen, the cursor is changed from an arrow to a cross hair. Then, the load can be defined by holding the left mouse button down at the starting point of the distributed load. As the mouse is dragged, a box appears, indicating a distributed load is being defined. When the left mouse button is released, the following dialog box in Figure 12-29 appears with the load value and coordinates

In this dialog box

- Type 150 in the "Load value" edit box
- Modify the load coordinate
- Click "OK" button

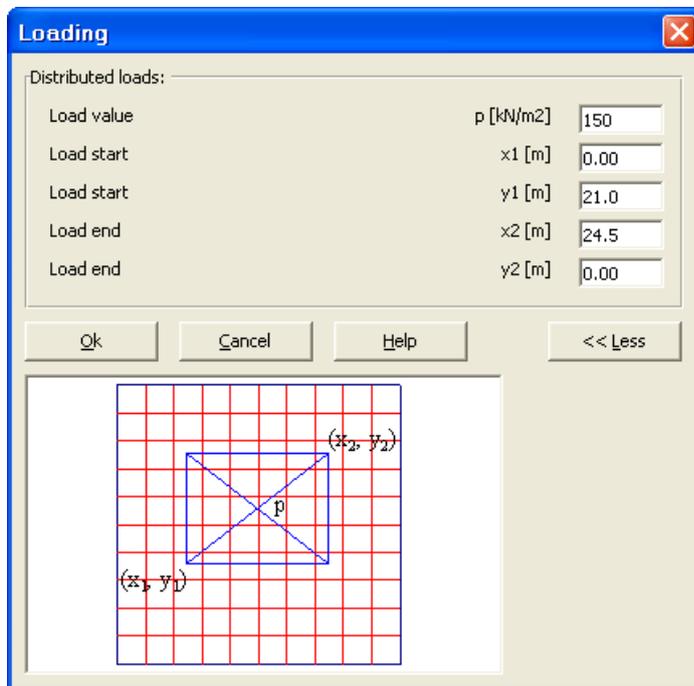


Figure 12-29 "Distributed loads p " list box

After you have completed the definition of all load data, the screen should look like the following Figure 12-30.

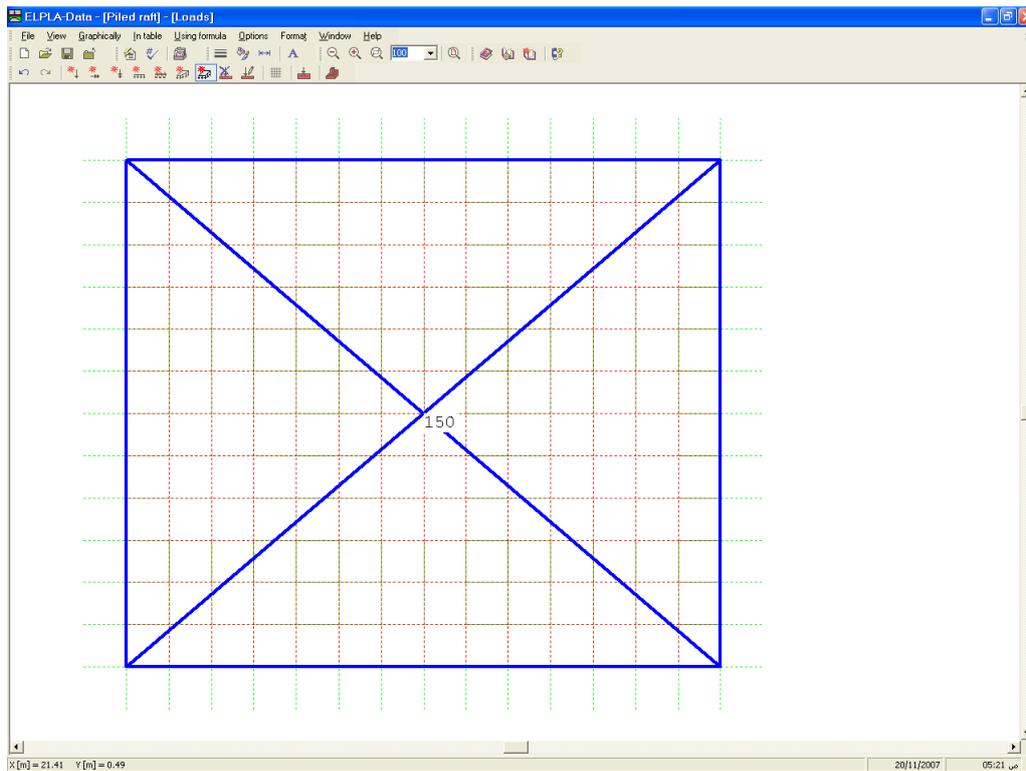


Figure 12-30 Loads on the screen

After finishing the definition of load data, do the following two steps:

- Choose "Save loads" command from "File" menu in Figure 12-30 to save the load data
- Choose "Close loads" command from "File" menu in Figure 12-30 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*.

Note that the sign " $\sqrt{\quad}$ " is typed automatically beside the "Loads" command in the "Data" menu of *ELPLA-Data*.

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

3 Carrying out the calculations

3.1 Starting *ELPLA-Solver*

To analyze the problem, leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window appears, Figure 12-31.

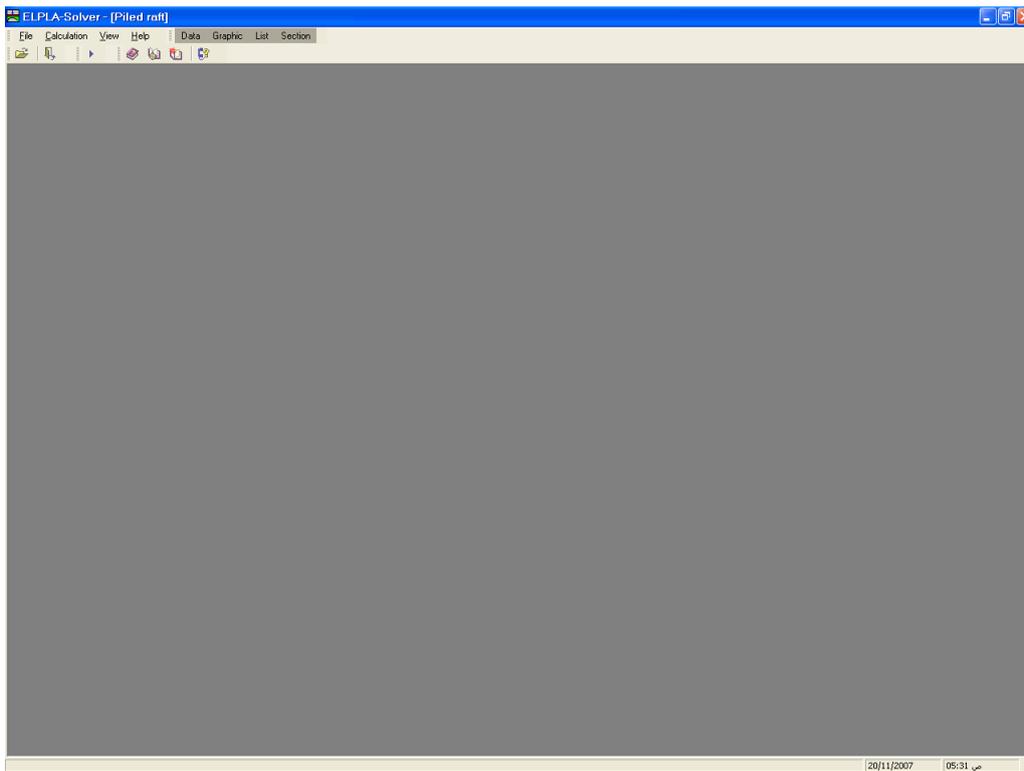


Figure 12-31 Opening screen of the sub program *ELPLA-Solver*

ELPLA-Solver automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. The "Calculation" menu 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
- Assembling the soil stiffness matrix
- Assembling the slab stiffness matrix
- Solving the system of linear equations (unsymmetrical matrix)
- Determining deformation, internal forces, contact pressures

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

3.2 Carrying out all computations

To carry out all computations in one time, choose "Computation of all" command from "Calculation" menu in *ELPLA-Solver* window. The analysis progress menu in Figure 12-32 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

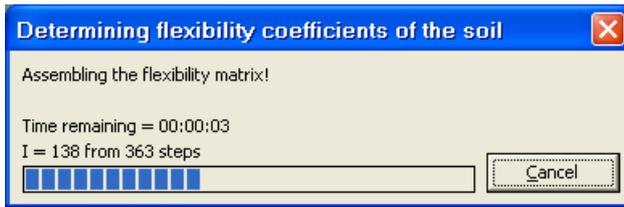


Figure 12-32 Analysis progress menu

Once the analysis is complete, a check menu of the solution appears, Figure 12-33. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

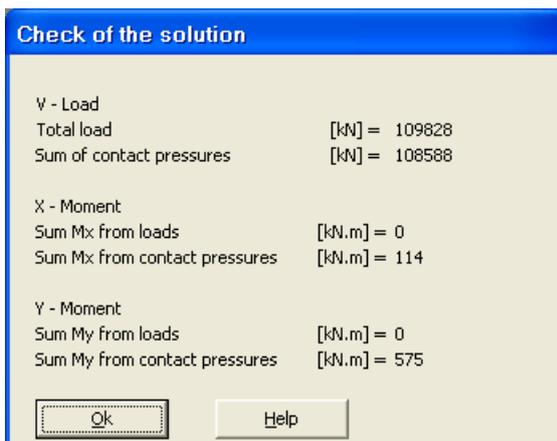


Figure 12-33 Menu "Check of the solution"

To finish analyzing the problem, click "OK" button.

4 Viewing data and results

ELPLA can view and print a wide variety of results in graphics, diagrams or tables through the three sub programs *ELPLA-Graphic*, *ELPLA-Section* and *ELPLA-List*. Data can also be viewed again and printed by the sub programs *ELPLA-Graphic* and *ELPLA-List*. Note that *ELPLA-Data* is used only to define and view the data of the problem. *ELPLA-Graphic* is used to print data graphically while *ELPLA-List* is used to print data numerically.

4.1 Viewing result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, leave *ELPLA-Solver* to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window.

ELPLA-Graphic window appears, Figure 12-34. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

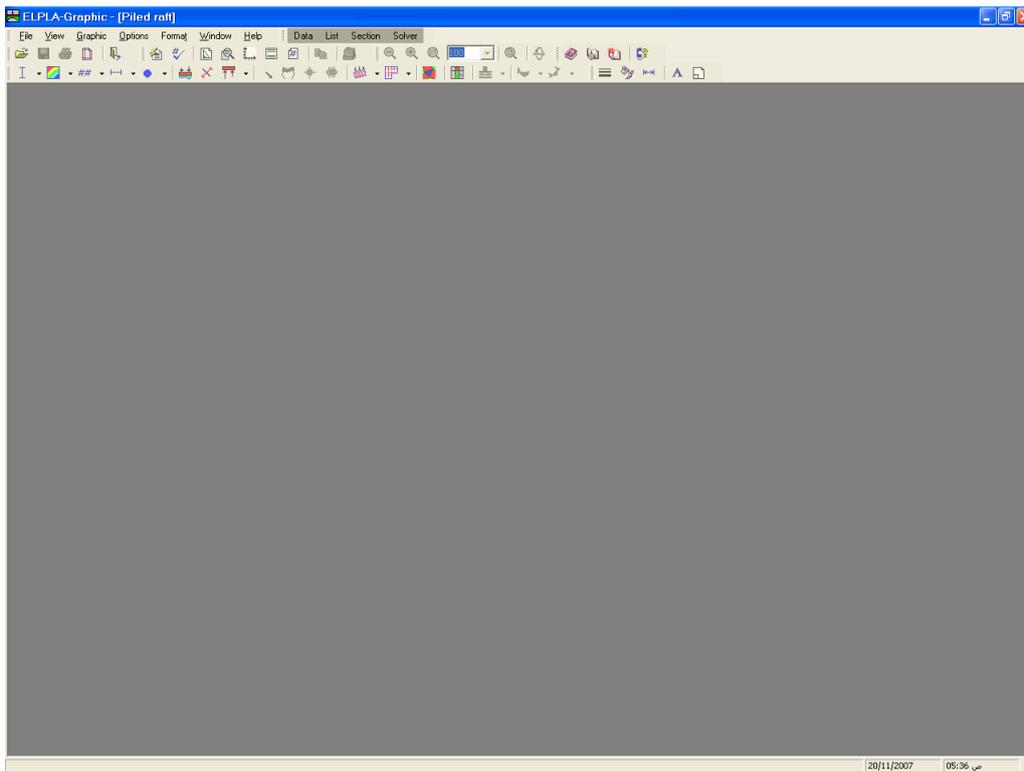


Figure 12-34 Opening screen of the sub program *ELPLA-Graphic*

To view the results for the raft as contour lines

- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 12-35 appears
- In "Results as contour lines" option box, select "Settlements s " as a sample for the results to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for the raft as shown in Figure 12-36.

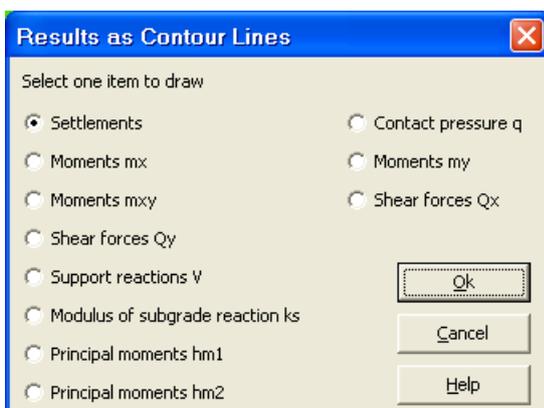


Figure 12-35 "Results as contour lines" option box

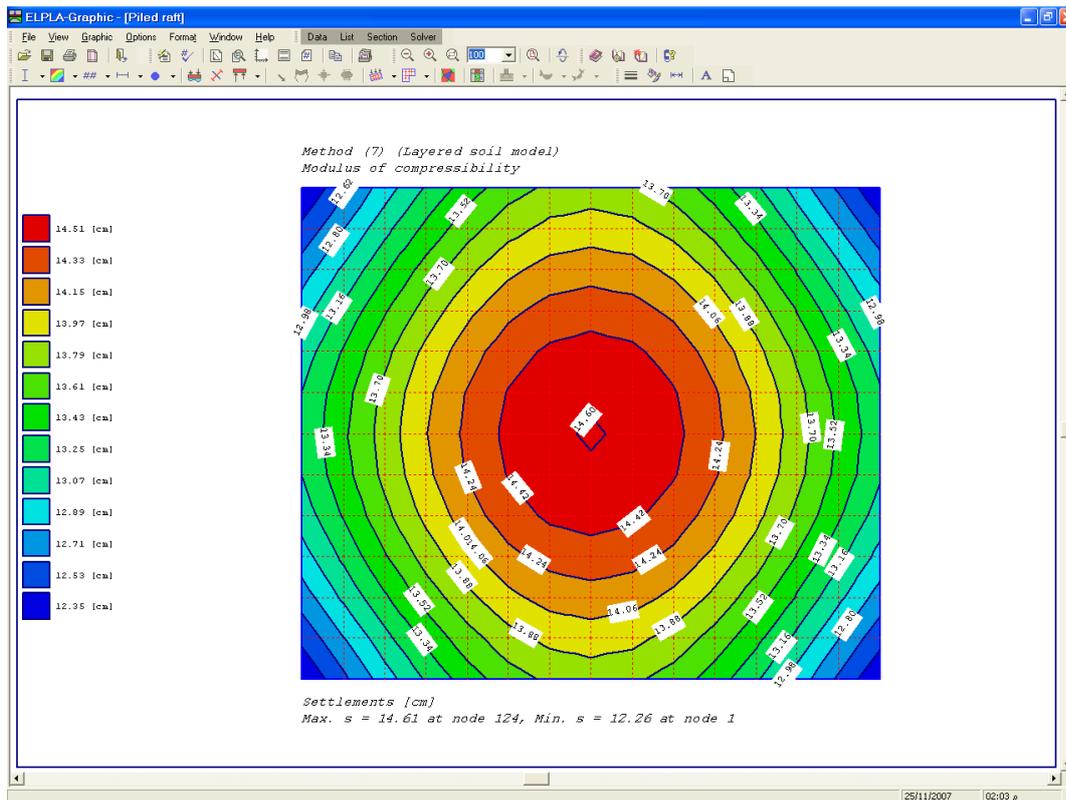


Figure 12-36 Settlements as contour lines for the raft

4.2 Plot a diagram of the results

To plot a diagram of the results, leave *ELPLA-Graphic* to *ELPLA-Section*. This is done by clicking on "Section" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Graphic* window. *ELPLA-Section* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Section* window.

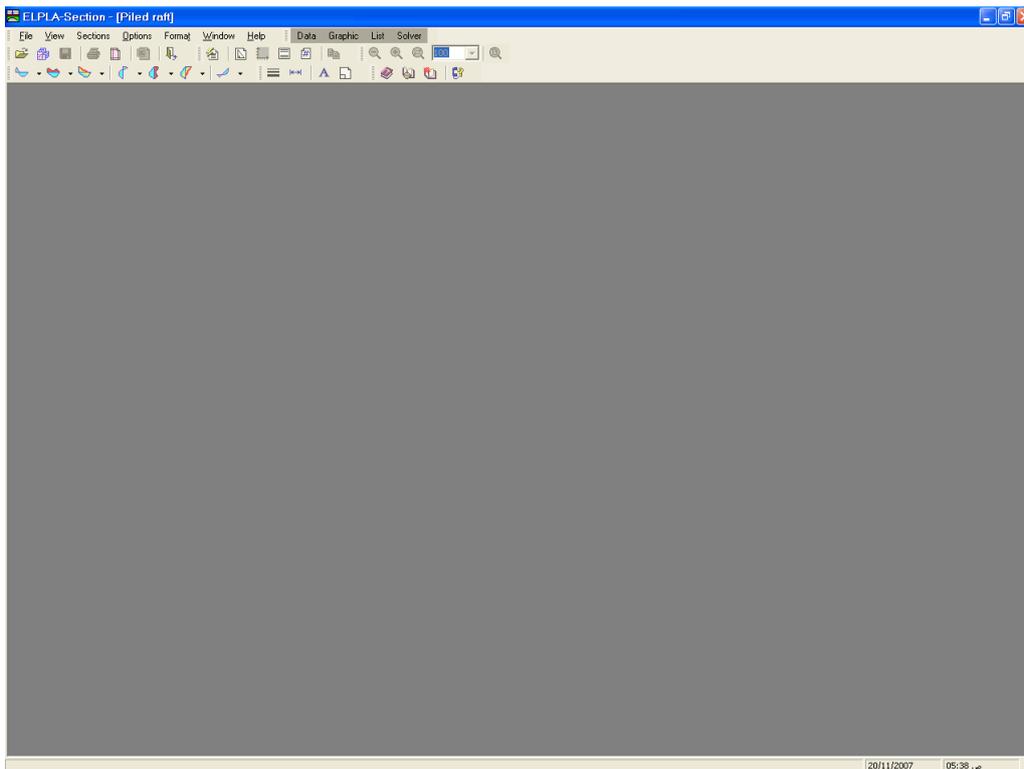


Figure 12-37 Opening screen of the sub program *ELPLA-Section*

Only the first command of the "Sections" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Options", "Format" and "Window" menus, which are used to define the preferences of the drawing such as plot parameters, scale, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

To plot a section in x -direction

- Choose "Section in x -direction" command from "Sections" menu.
The following option box in Figure 12-38 appears
- In the "Section in x -direction" option box, select "Settlements s " as an example for the results to be plotted in a diagram
- Click "OK" button

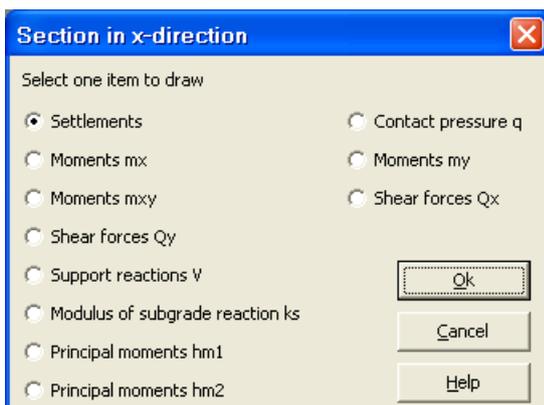


Figure 12-38 "Section in x -direction" option box

The following option box in Figure 12-39 appears to specify the section in x -direction that is required to be plotted.

In this dialog box

- Type 10.5 in the "Section at y-coordinate" edit box to plot a diagram at the middle of the raft
- Click "OK" button

The settlements are now plotted in a diagram at the middle of the raft as shown in Figure 12-40.

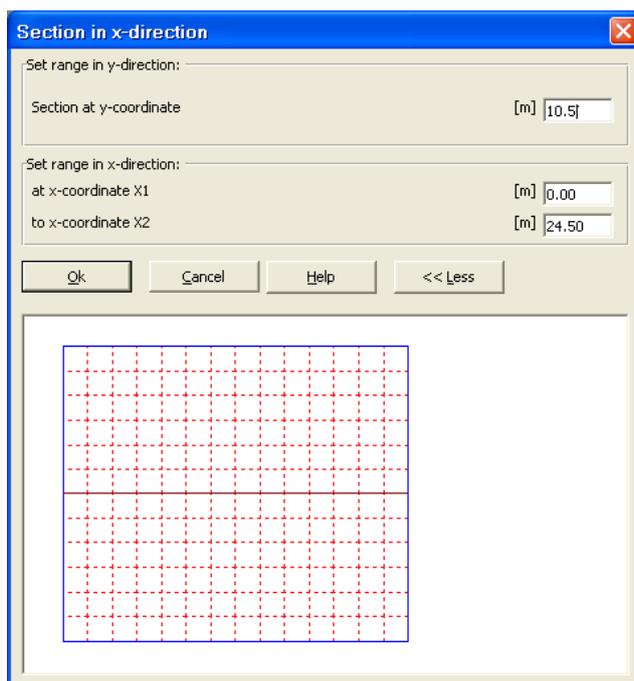


Figure 12-39 "Section in x -direction" dialog box

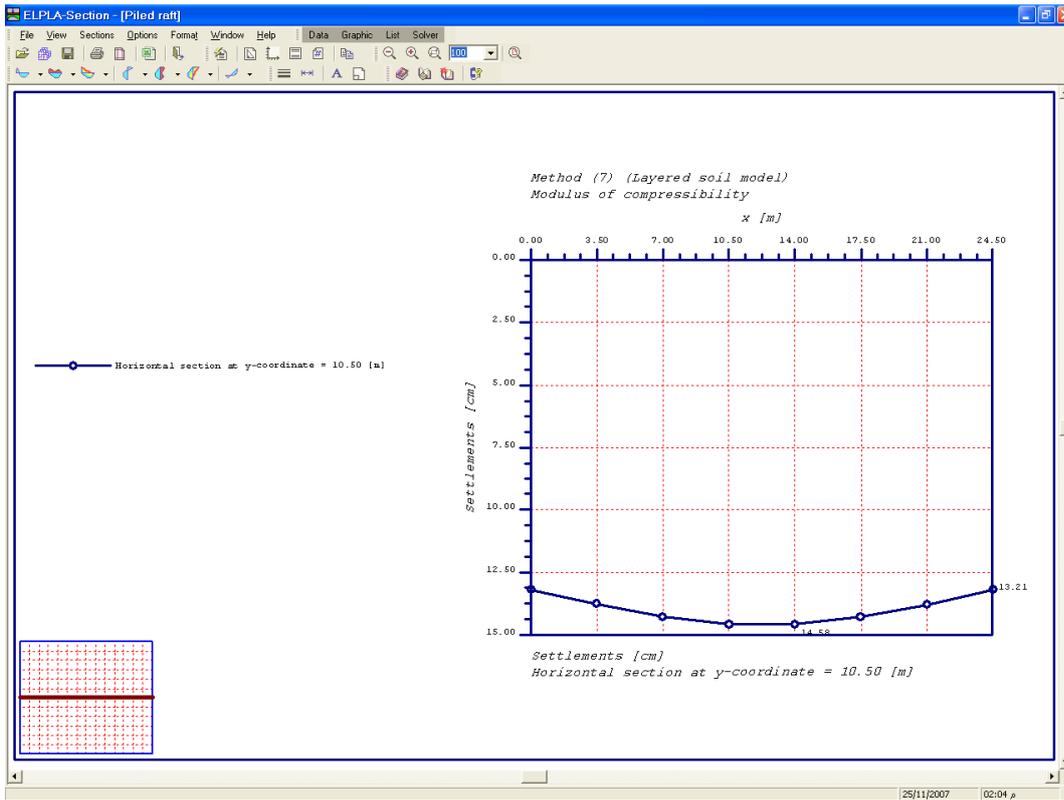


Figure 12-40 Diagram of settlements in x -direction at the middle of the raft

4.3 Listing data and result in tables

To list tables of data and results, switch to *ELPLA-List*. This is done by clicking on "List" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Section* window. *ELPLA-List* window appears, Figure 12-41.

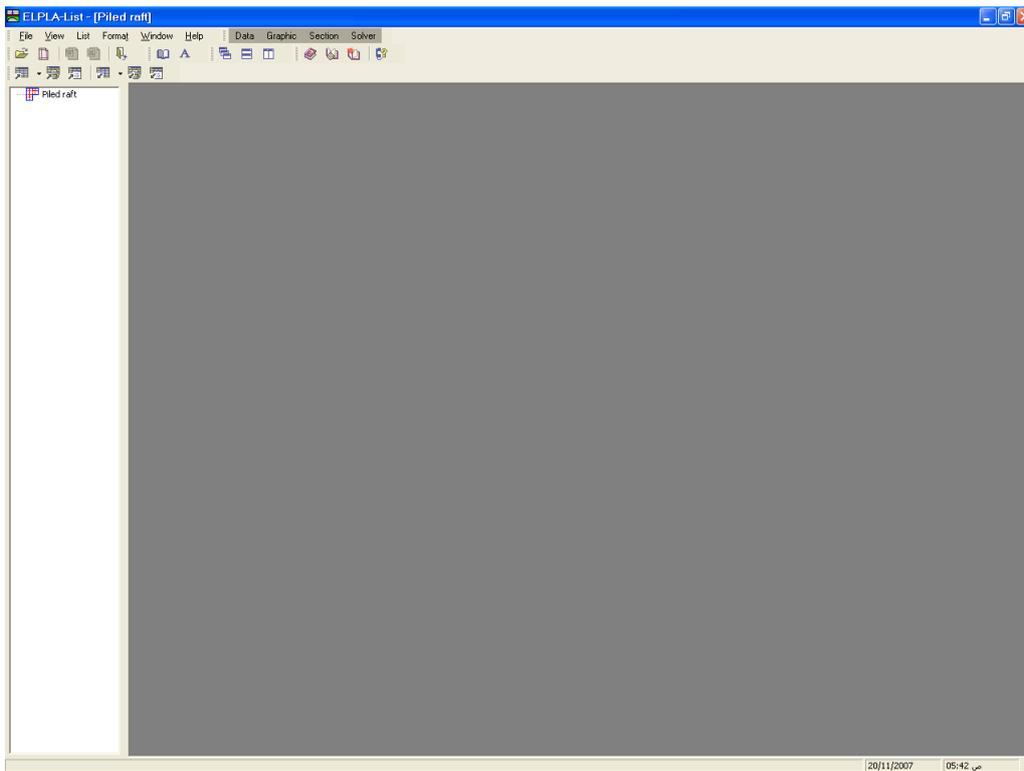


Figure 12-41 Opening screen of the sub program *ELPLA-List*

The function of *ELPLA-List* is listing and printing data and results in tables. The data and results can be exported to other Windows applications to prepare reports or add further information. *ELPLA-List* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-List* window.

Only one command of the "List" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Format" and "Window" menus, which are used to define the preferences of the tables such as page format, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

To list results in tables

- Choose "Display tables of results" command from "List" menu.
The following option box in Figure 12-42 appears
- In this option box, select "Pile results" as an example for the results to be listed in a table
- Click "OK" button. The pile results are now listed on the screen (Figure 12-43)
- Choose "Send to Word" from "File" menu if you wish to export the text to the MS Word application, Figure 12-44

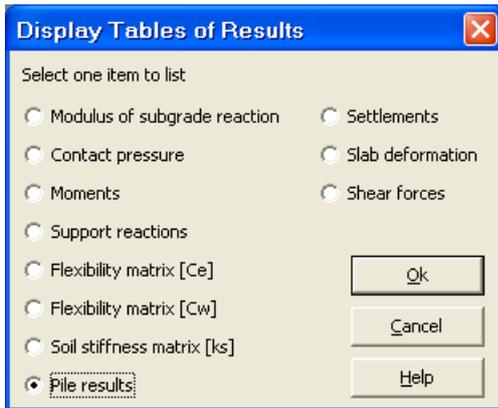


Figure 12-42 "Display tables of results" option box

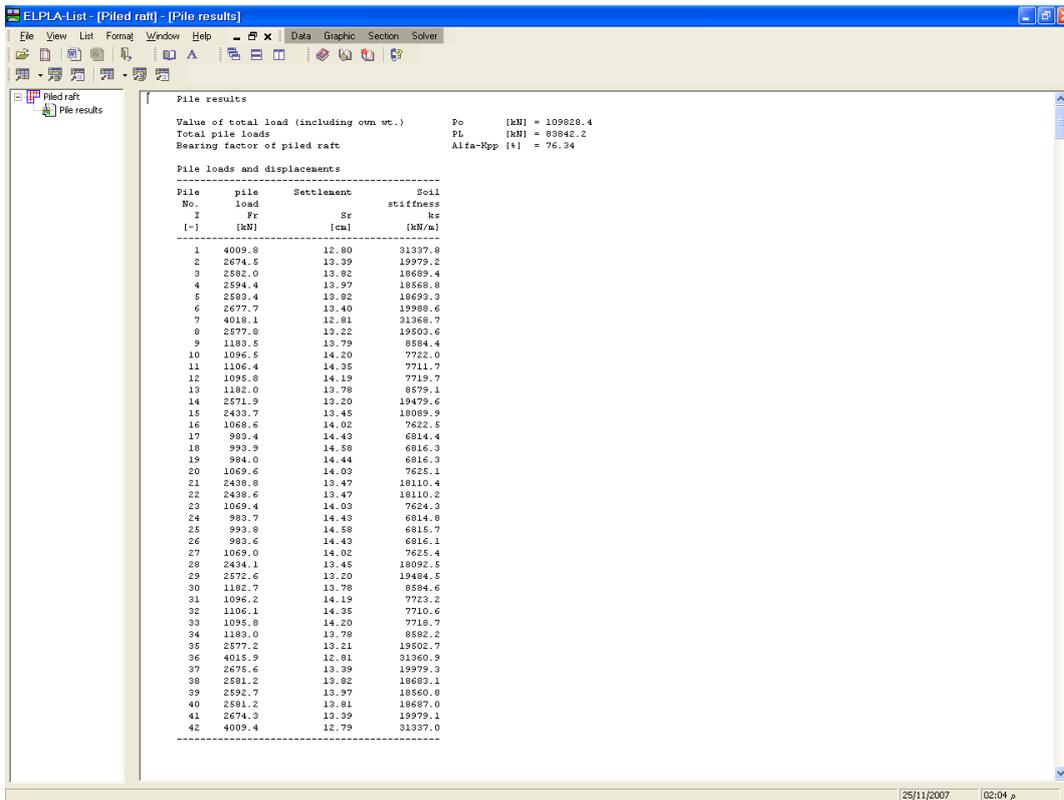


Figure 12-43 List of pile results

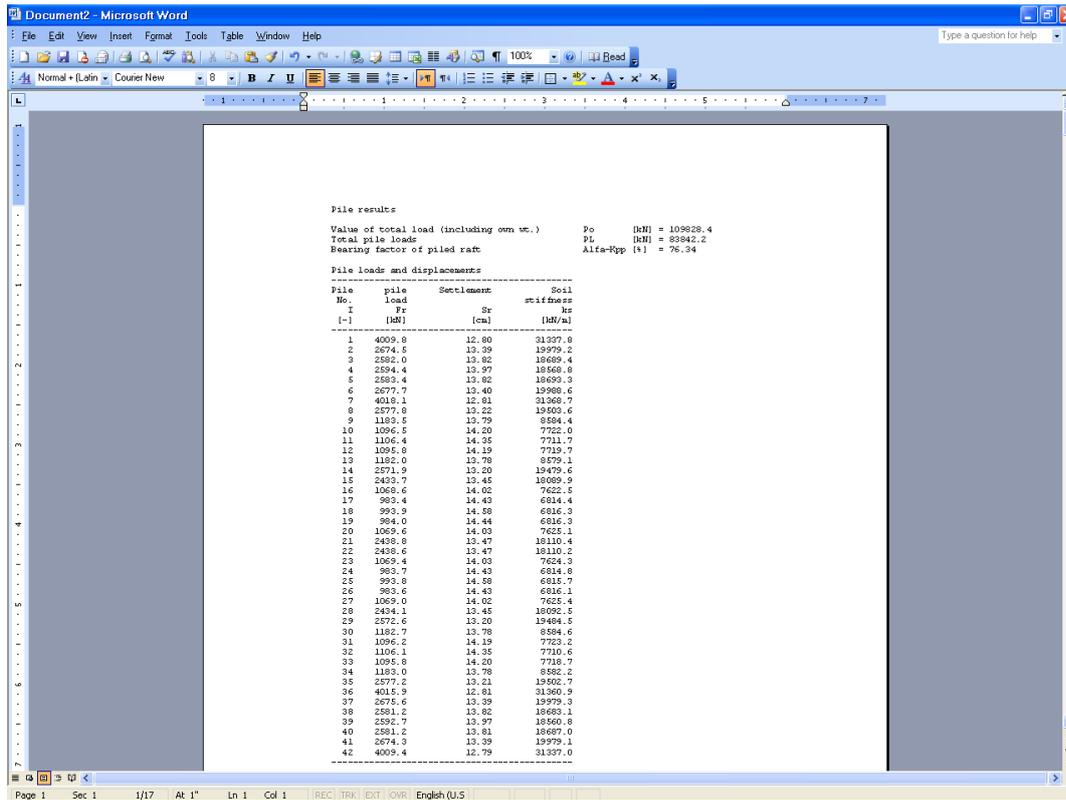


Figure 12-44 Exported results in MS Word

5 Index

A

Analysis..... 4, 8, 17, 25
 Automatically..... 7, 15, 17, 19, 22, 24, 25, 28, 32

B

Boring log 15, 17, 19

C

Calculation 24
 Calculation methods..... 4, 5
 Check of the solution..... 26
 Computation of all..... 25
 Contact pressures 25
 Contour lines 27

D

Deformation 25
 Diagram..... 26, 28, 31

E

ELPLA-Boring 16, 19
ELPLA-Data 4, 8, 9, 11, 12, 15, 19, 22, 24, 26
ELPLA-Graphic 26, 27, 28
ELPLA-List 26, 31, 32
ELPLA-Section..... 26, 28
ELPLA-Solver 24, 25, 26
 E-Modulus..... 20

F

FE-Net..... 9
 File 4, 7, 11, 15, 22, 24, 26, 32
 Flexibility coefficients..... 25
 Foundation 4, 9, 17, 20, 21
 Foundation properties 19

G

Generation type 9, 10
 Graphic..... 26

I

Internal forces 25

K

Key..... 20

L

List 31, 32
 Load vector 25
 Loads..... 22, 24

M

Main soil data..... 17, 18
 Modulus of elasticity for loading..... 4
 Modulus of elasticity for reloading..... 4

N

Numerical calculation 4

O

Options..... 6, 29

P

Poisson's ratio..... 3, 4, 20
 Project..... 4, 7, 8, 24
 Project identification..... 8

R

Raft 3, 4, 7, 9, 24, 27, 28, 30, 31

S

Save..... 7, 8, 11, 15, 22, 24
 Save as 7
 Soil..... 4, 17, 25
 Soil properties 4, 15, 16, 19
 Soil stiffness matrix..... 25
 System symmetry 6

T

Tutorial manual 4

U

Unit weight 3, 4, 17, 21

Unsymmetrical system 6

User's Guide 4, 29, 32

Y

Young's modulus 3, 4