Example 11

Analysis of pile groups

Contents Page

1 Description of the problem

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

1.1 Load and dimensions

A pile cap on 24 vertical piles is considered as shown in Figure 11-1. All piles are equal in diameter and length where pile diameter is 0.5 [m] while pile length is 10 [m]. Spacing between piles is 1.6 [m]. It is required to analyze the pile cap as rigid free-standing raft due to a vertical load of P = 8000 [kN] acting on the pile cap with eccentricities e_x = 1.4 [m] and e_y = 1.8 [m] in both *x*- and *y*-directions.

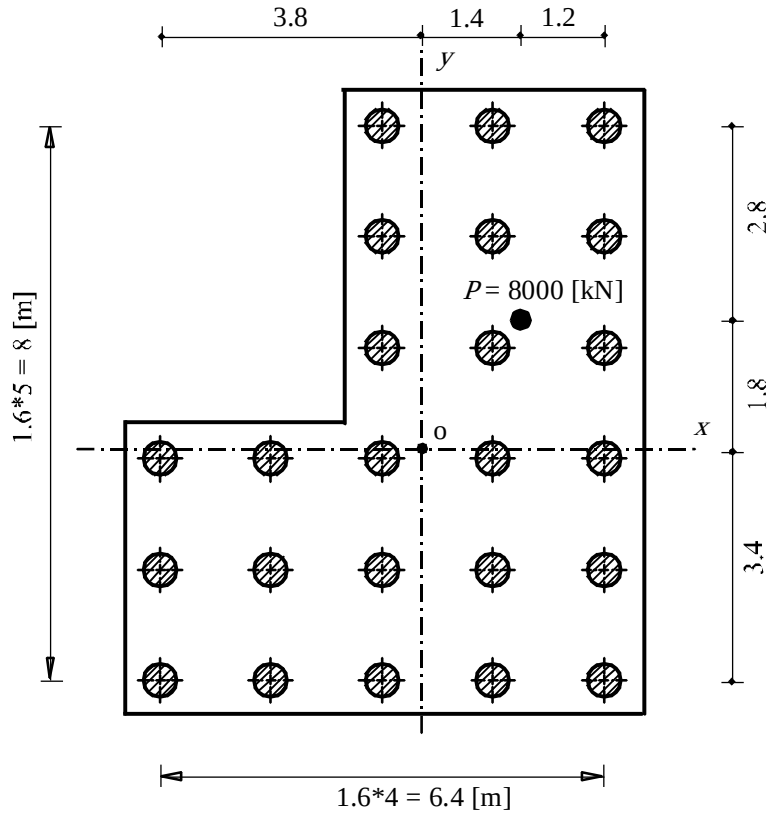


Figure 11-1 Plan with raft geometry [m], resultant force on the raft [kN] and pile locations

1.2 Soil properties

The subsoil under the raft consists of a layer of silt, 30 [m] thick. The raft lies directly on the ground surface. There is no effect of water table on the raft. 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]
Poisson's ratio	ν_s	= 0.3	[-]

1.3 Mathematical model

In this example, rigid free-standing raft (method 9) is chosen to analyze the 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 pile groups. 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 11-2. This wizard will guide you through the steps required to create the project. As shown in Figure 11-2, the first form 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 pile groups, select "Analysis of piled raft" then click "Next" button to go to the next page.

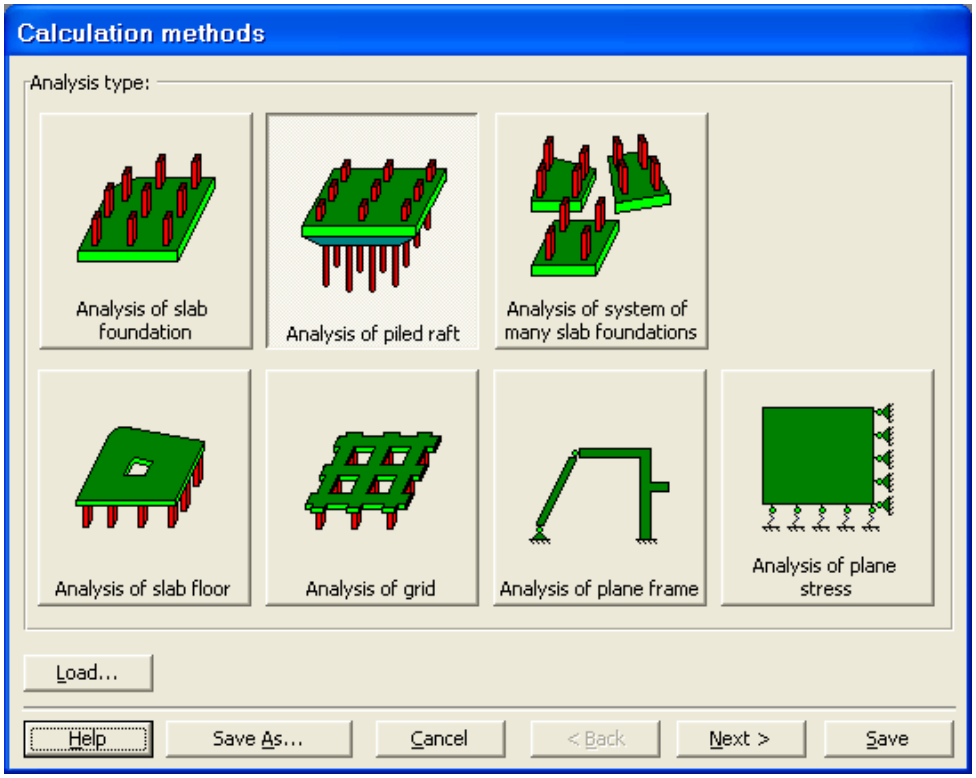


Figure 11-2 "Calculation method" wizard with "Analysis type" form

After clicking "Next" button, the "Calculation methods" form appears, Figure 11-3.

To define the calculation method

- Select the calculation method "9-Rigid free-standing raft"
- Select the subsoil model "Layered soil model"
- Click "Next" button to go to the next form

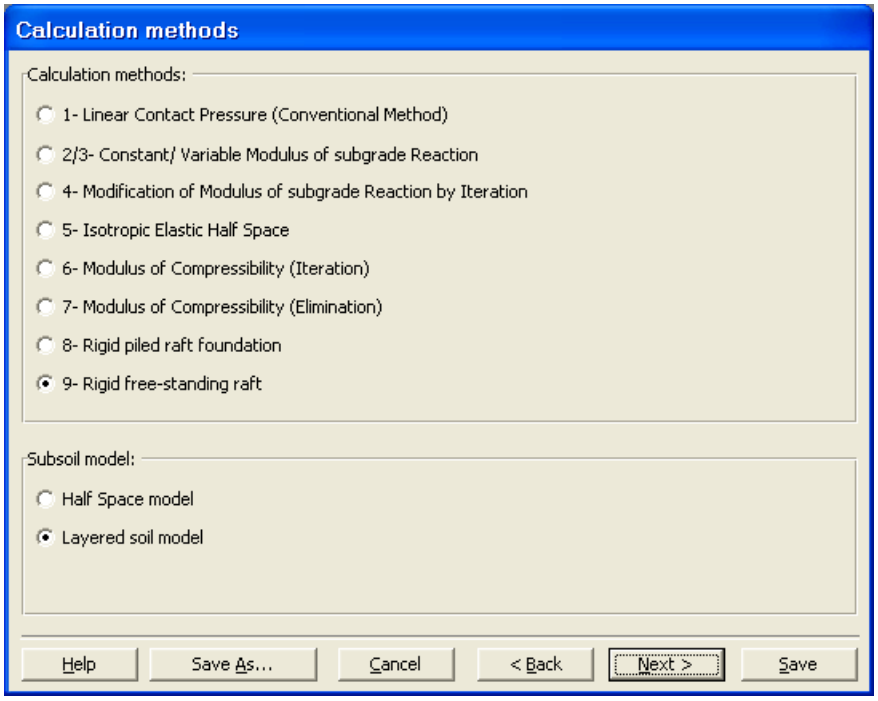


Figure 11-3 "Calculation methods" form

The next form is the "System symmetry" (Figure 11-4).

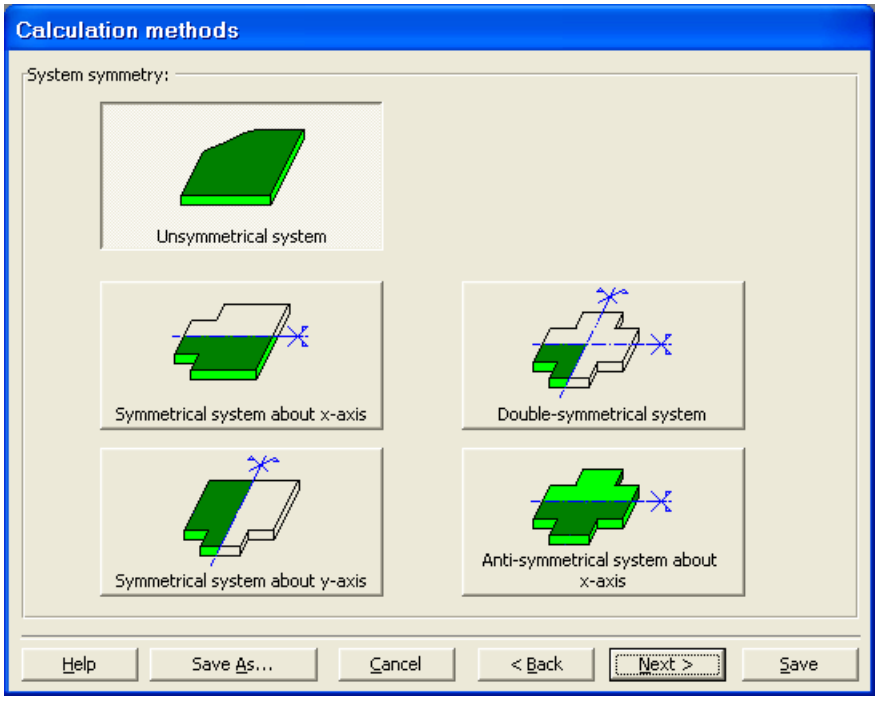


Figure 11-4 "System symmetry" form

In this form choose "Unsymmetrical system" then click "Next" button.

The last form of the wizard assistant contains the "Option" list, Figure 11-5. 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.

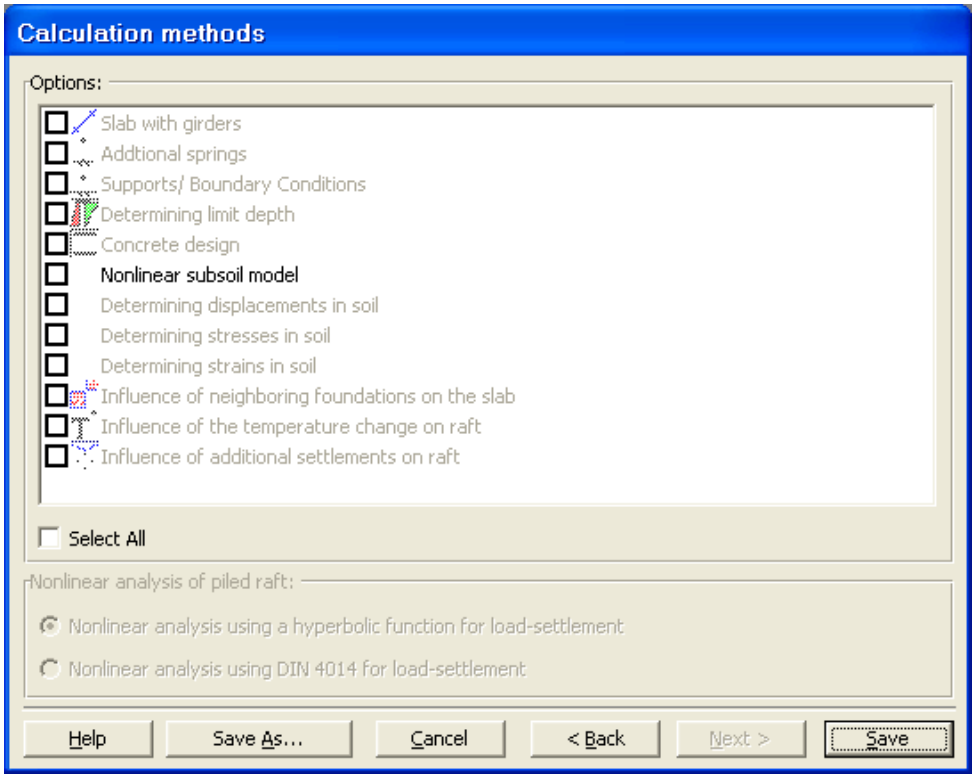


Figure 11-5 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 11-6.

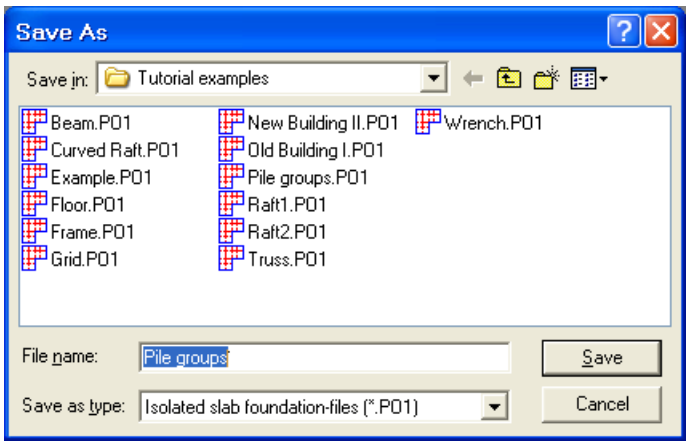


Figure 11-6 "Save as" dialog box

In this dialog box

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

ELPLA will activate the "Data" menu. In addition, the file name of the current project [Pile groups] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 11-7. 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 11-7. *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.

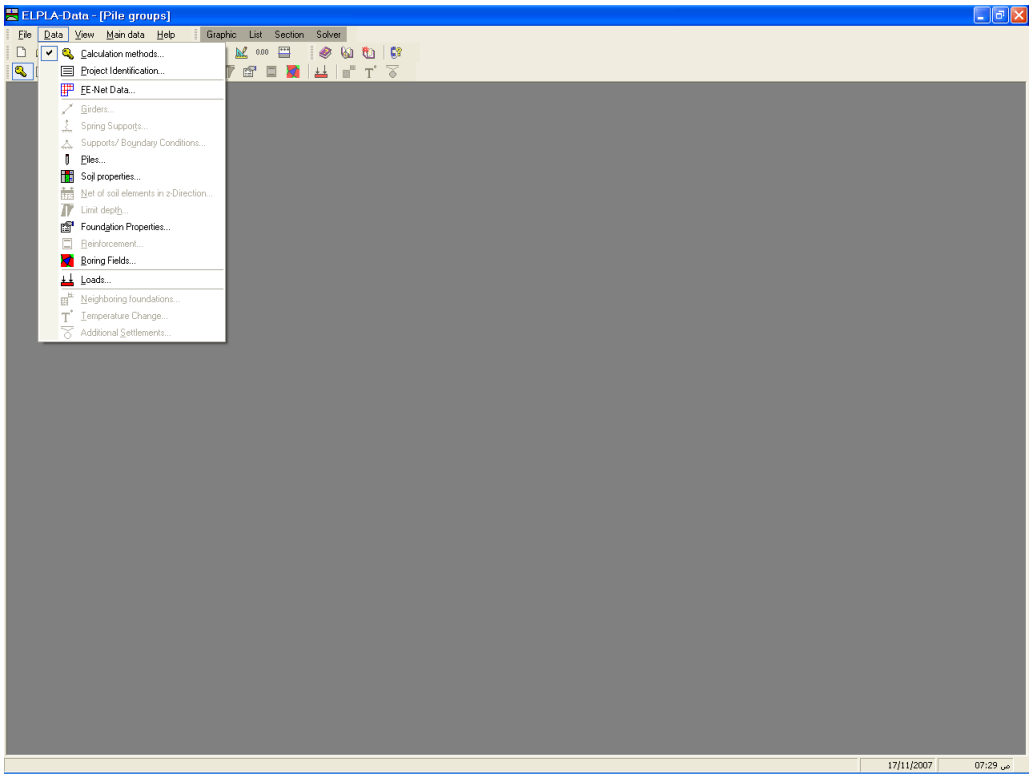


Figure 11-7 *ELPLA-Data* after defining the calculation method

2.2 Project identification

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 11-8 appears.
In this dialog box

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

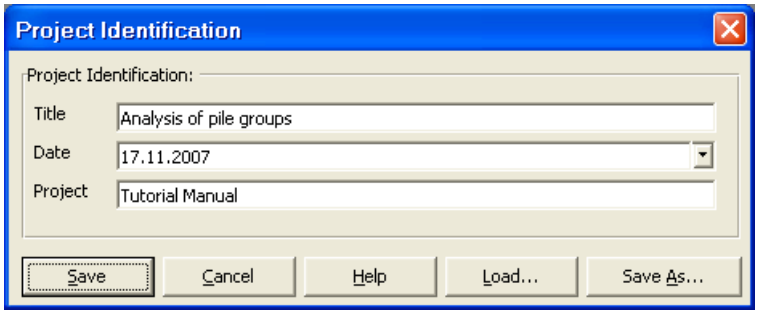


Figure 11-8 "Project identification" dialog box

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 11-9. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 11-9 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 an irregular shape.

In pile group problems, it is not required to generate a FE-Net for the entire dimension of the raft. Only a net containing pile location is enough. *ELPLA* has different procedures for defining the FE-Net. The easy procedure to define the FE-Net of these pile groups is generating a mesh for the entire area firstly, then removing the unnecessary nodes to get the foundation shape.

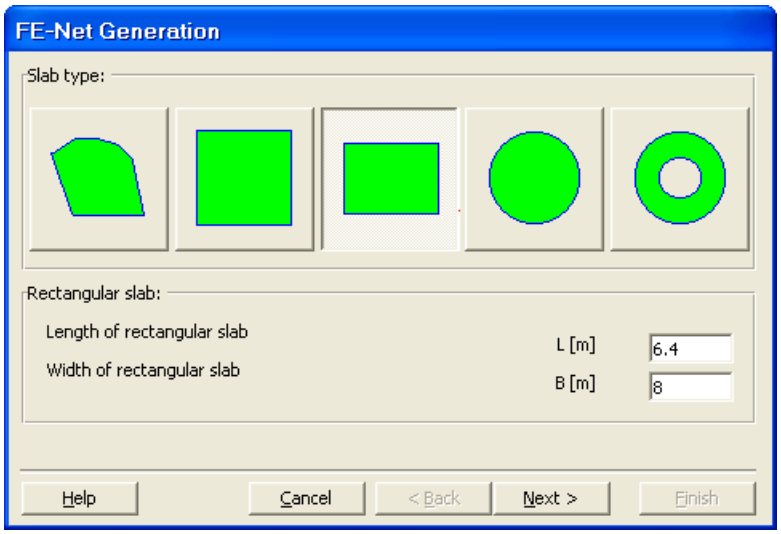


Figure 11-9 "FE-Net generation" wizard with "Slab type" form

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 11-10. *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.

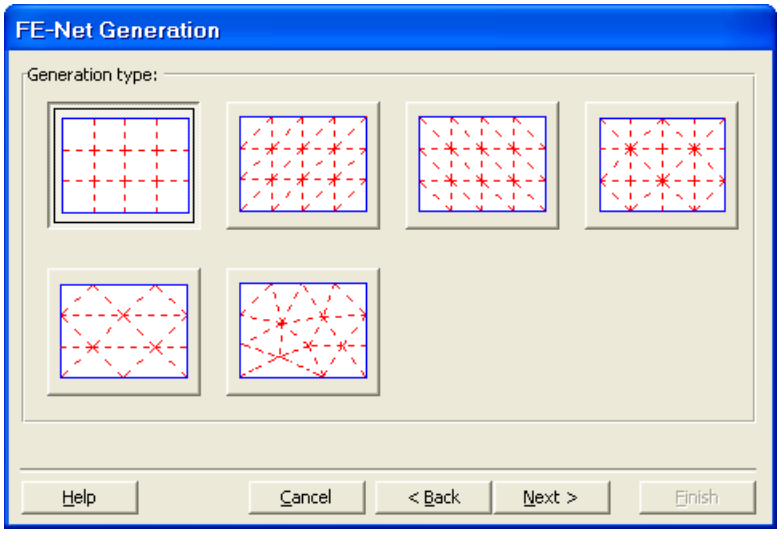


Figure 11-10 "Generation type" form

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

In this dialog box

- In "Grid in *x*-direction" frame, type 4 in the "No. of grid spaces" edit box
- In "Grid in *y*-direction" frame, type 5 in the "No. of grid spaces" edit box
- Type 1.6 in the "Grid interval" edit box
- Click "OK" button

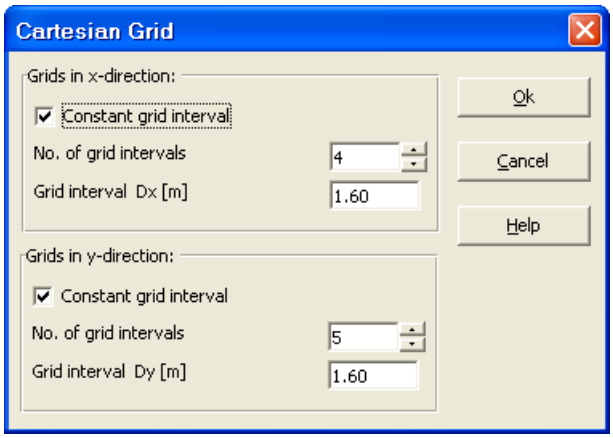


Figure 11-11 "Grid definition" form

ELPLA will generate a FE-Net for a rectangular area of 6.4 [m] length and 8 [m] width with square elements of 1.6 [m] each side. The following embedded program in Figure 11-12 appears with the generated net.

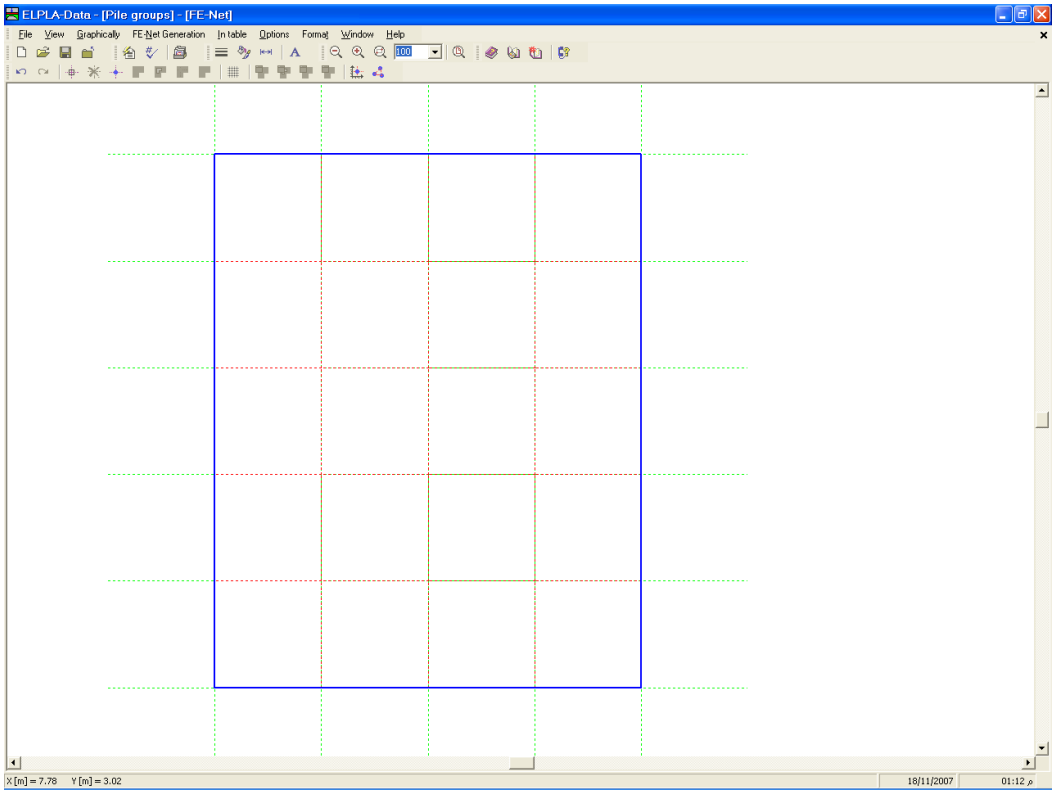


Figure 11-12 FE-Net of a rectangular slab on the screen

Deleting nodes from the FE-Net

To select the unnecessary nodes that are required to be removed from the net, first choose "Select nodes" command from "Graphically" menu. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair. The command "Remove nodes" in the menu "Graphically" will be enabled, indicating the mode in which is being operated. Next, select the required nodes by clicking on each node individually or selecting a group of nodes as shown in Figure 11-13. To remove the selected nodes, choose "Remove nodes" command from the "Graphically" menu. The action of this command is indicated in Figure 11-14.

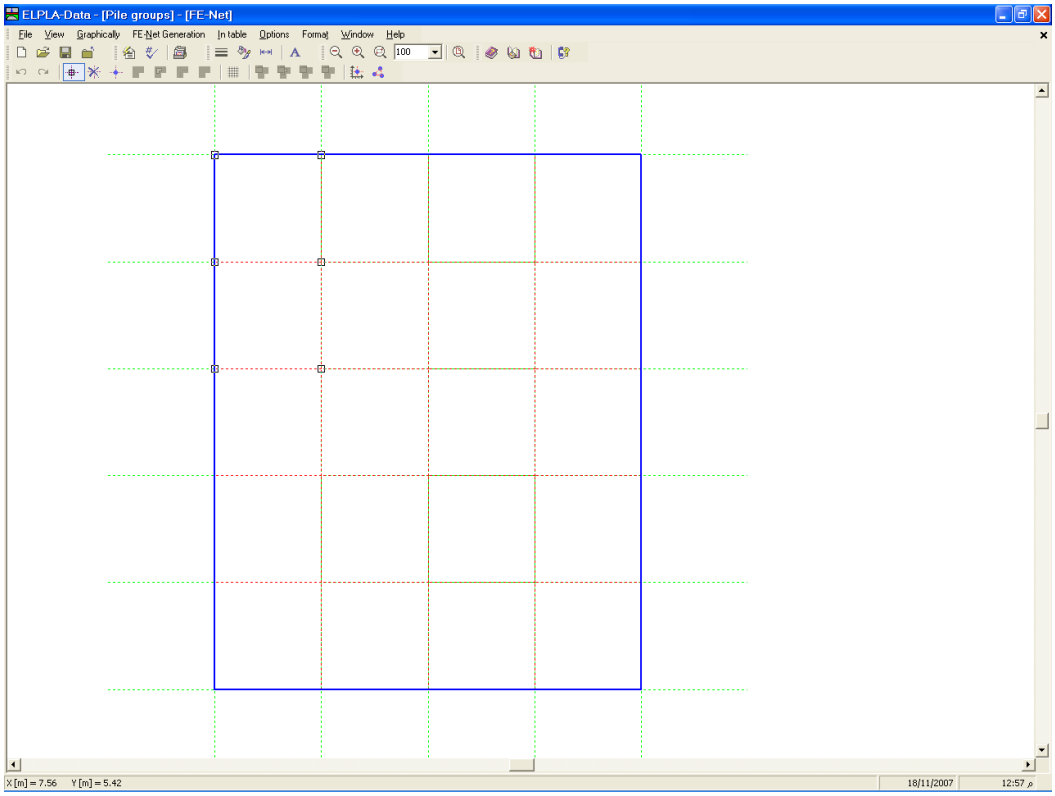


Figure 11-13 Generated FE-Net after selecting the unnecessary nodes

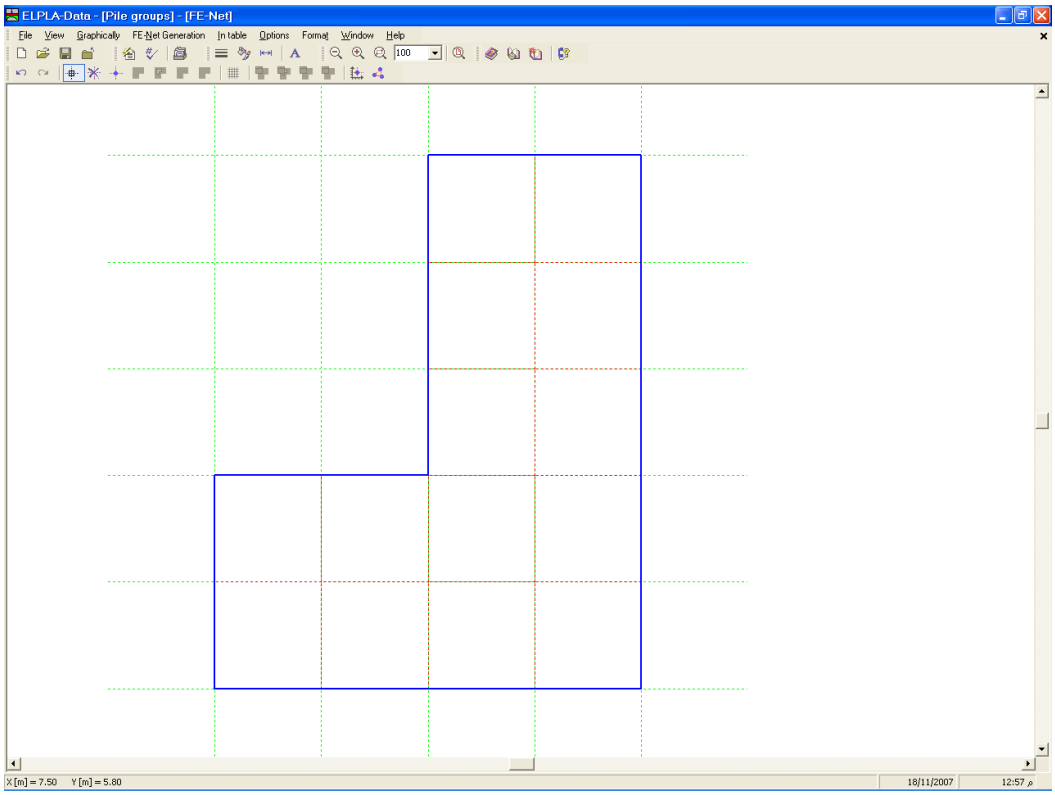


Figure 11-14 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save FE-Net" command from "File" menu in Figure 11-14 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 11-14 to close the "FE-Net" embedded program and return to the main window of *ELPLA-Data*

Note that the sign "√" 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 11-15 appears.

In this dialog box

- Type 0.5 in the "Element length of pile" edit box
- Click "Save" button

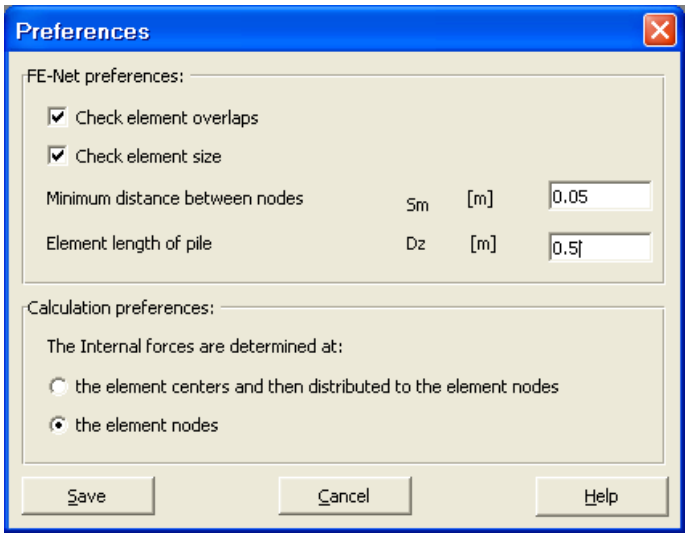


Figure 11-15 "Preferences" dialog box

2.5 Piles

To define piles, choose "Piles" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 11-16 appears.

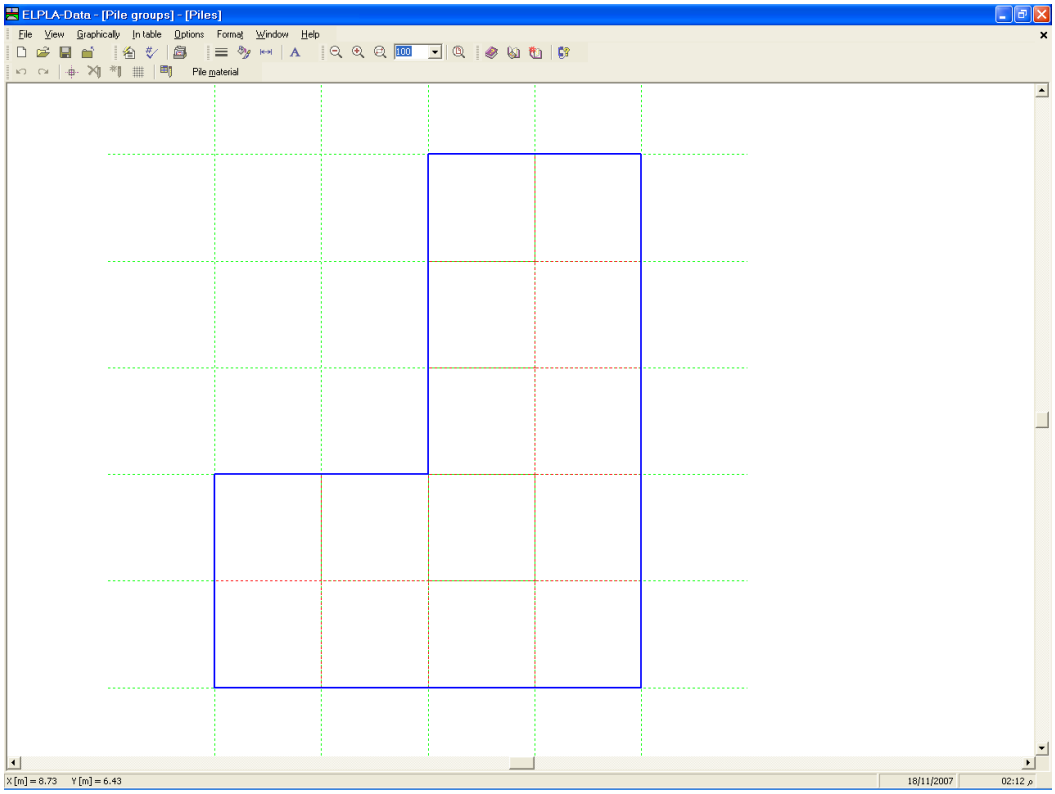


Figure 11-16 "Piles" embedded program

Pile groups

To define pile groups those have the same diameter and length, choose "Pile groups" command from "In Table" menu in Figure 11-16. When the "Pile groups" command is chosen, the following table in Figure 11-17 appears allowing you to define the pile diameter and length.

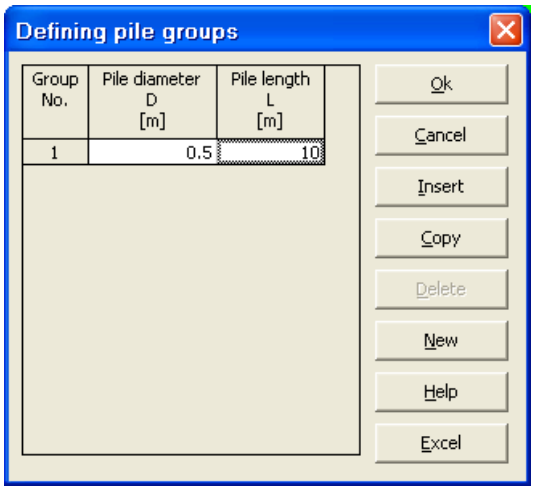


Figure 11-17 Defining pile groups

Pile material

To define pile material, choose "Pile material" command from "In Table" menu in Figure 11-16. When the "Pile material" command is chosen, the following dialog box in Figure 11-18 appears allowing you to define the pile material. Type 0 in the "Unit weight of the pile concrete" edit box to neglect the pile weight.

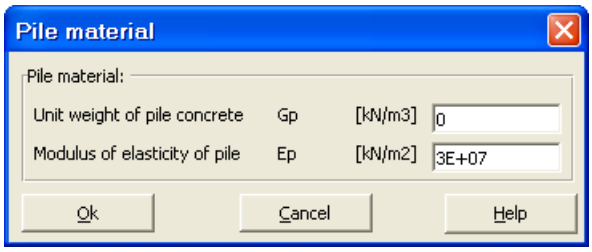


Figure 11-18 "Defining element groups" dialog box

Pile location

To define pile locations on the net

- Choose "Select nodes" command from "Graphically" menu in Figure 11-16
- Select nodes that have piles as shown in Figure 11-19
- Choose "Add piles" command from "Graphically" menu in Figure 11-19. The "Defining pile groups" dialog box in Figure 11-20 appears. In this dialog box click "OK" button

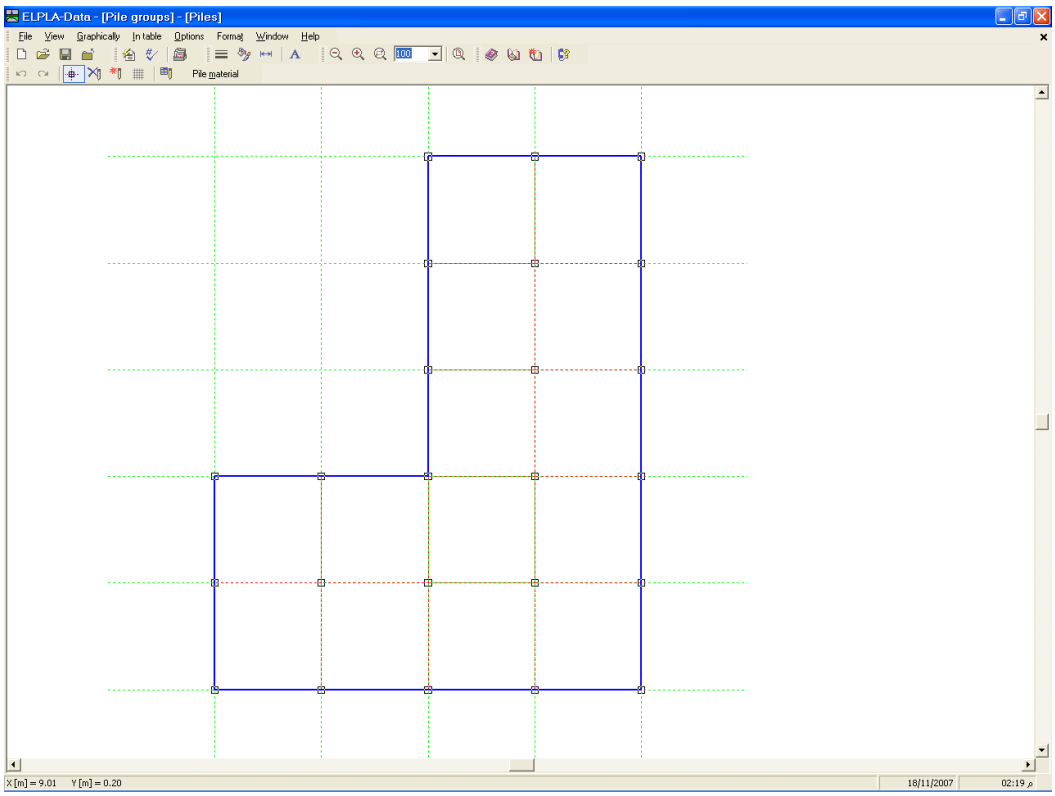


Figure 11-19 Selection of nodes that have piles

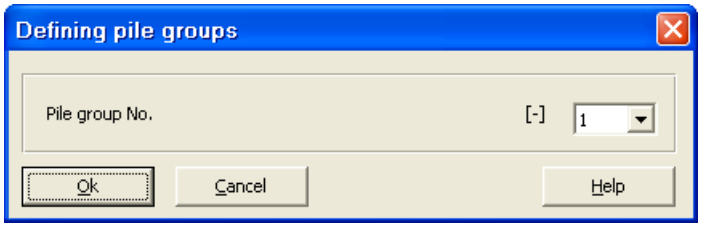


Figure 11-20 "Defining pile groups" dialog box

After you have completed the definition of the piles, the screen should look like the following Figure 11-21.

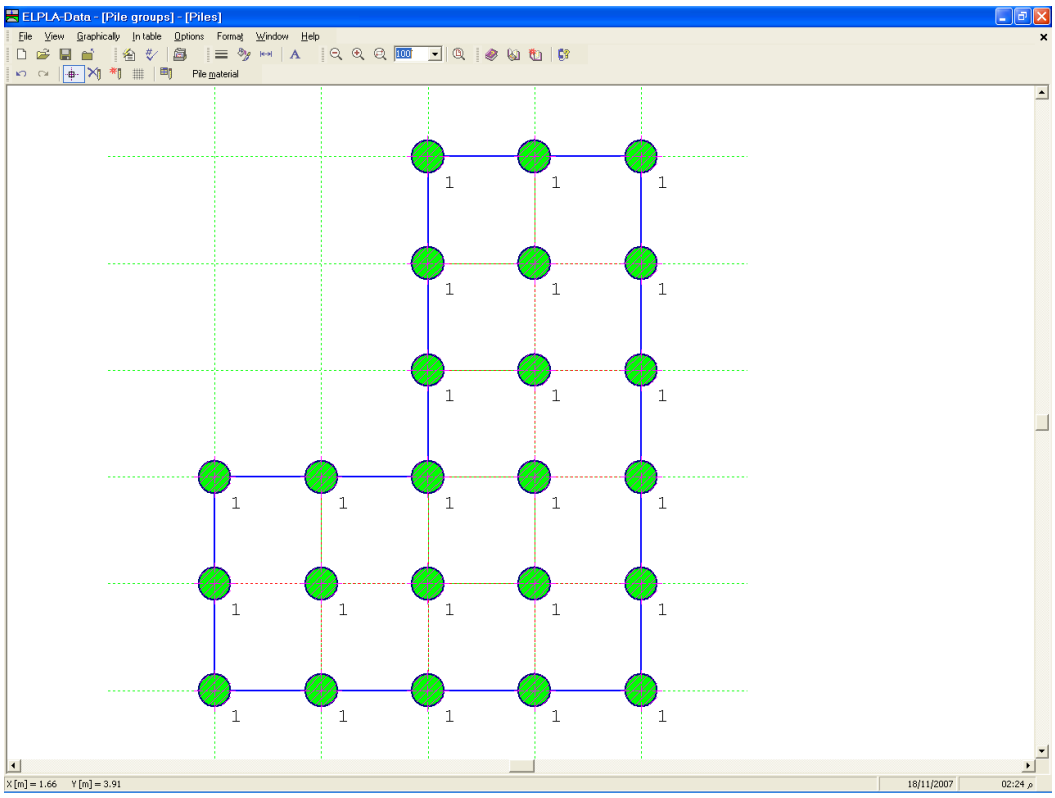


Figure 11-21 Piles on the screen

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

- Choose "Save piles" command from "File" menu in Figure 11-21 to save the data of piles
- Choose "Close piles" command from "File" menu in Figure 11-21 to close the *ELPLA-Piles* sub program and to return to the main window of *ELPLA-Data*

Note that the sign "√" 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 11-22 appears with a default-boring log.

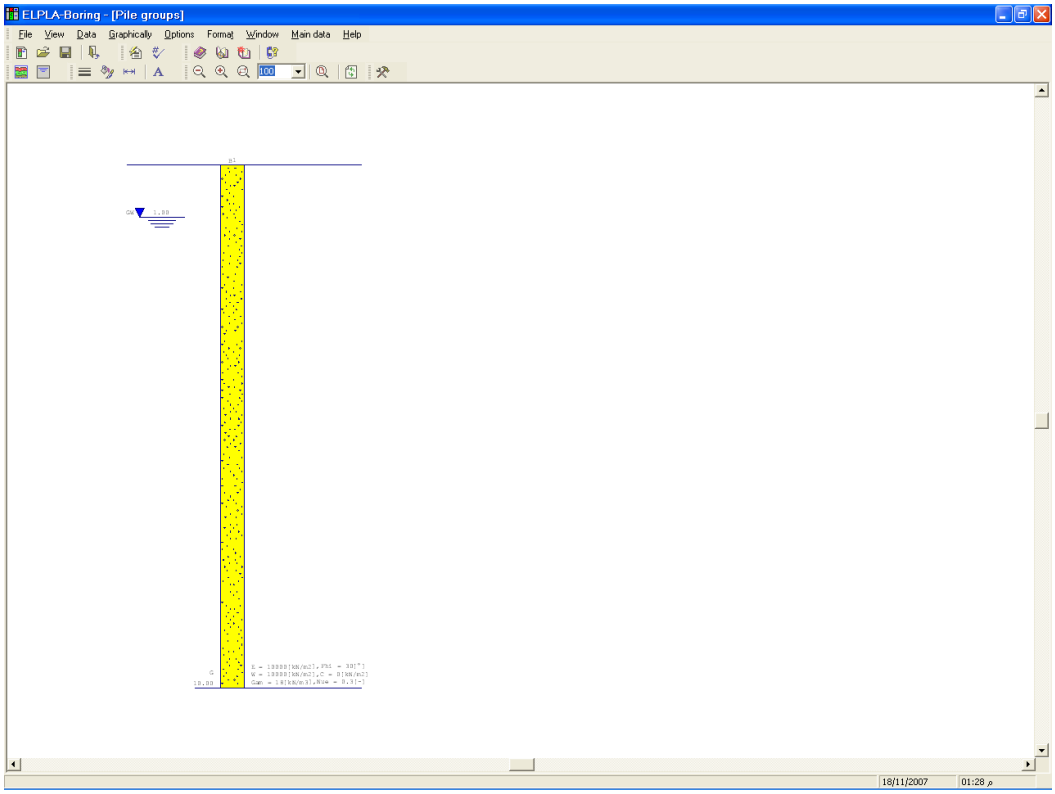


Figure 11-22 *ELPLA-Boring* sub program with a default-boring log

Modifying data of boring log graphically

In *ELPLA*, the boring log can be defined or modified graphically, which makes the definition of the boring log very easy. By double-clicking the left mouse button on a specified screen position, the user can define or modify the soil data and input parameters graphically.

To enter the geotechnical data of the soil layer

- Double-click on the geotechnical data of the soil layer. The corresponding dialog box appears allowing you to modify the geotechnical data of the soil layer, Figure 11-23
- In the dialog group box "Geotechnical data of the layer" in Figure 11-23, define the geotechnical data of the soil layer as follows:

E_s	= 10 000	[kN/m^2]
W_s	= 10 000	[kN/m^2]
ν_s	= 0.3	[-]

The values *Es* and *Ws* are the same because the effect of reloading on the soil is not required. The unit weight of the soil is used to determine the overburden pressure q_v [kN/m^2] due to the removed soil, which is equal to γ_s * d_f. In the current example d_f = 0.0, which means the unit weight of the soil is not required. However, the unit weight of the soil is entered by the default value. Also, 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. Next, click "OK" button.

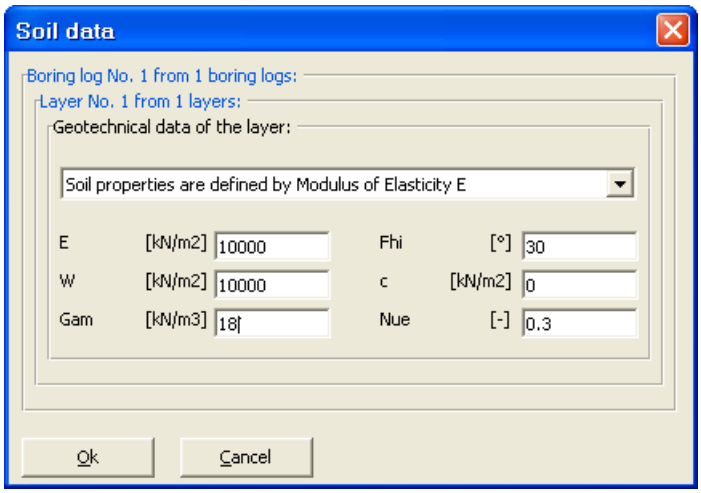


Figure 11-23 "Geotechnical data of the soil layer" dialog box

To define the soil type and color for the layer

- Double-click on the soil symbol of the soil layer. The corresponding dialog box appears allowing you to modify the soil symbols of that layer, Figure 11-24
- Select "U, Silt" as the soil type in the "Main soil type 1" combo box in the dialog group box "Soil and rock symbols" in Figure 11-24. The color of the silt and a short text "U" according to the German specification code DIN 4023 will be automatically created
- Click "OK" button

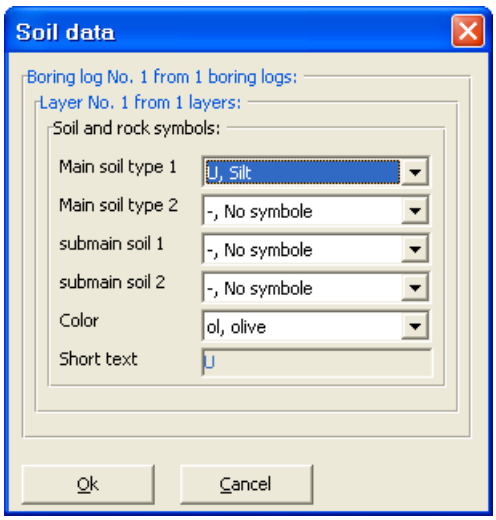


Figure 11-24 "Soil and rock symbols" dialog box

To modify a layer depth

- Double-click on the layer depth. The corresponding edit box appears allowing you to modify the layer depth under the ground surface, Figure 11-25
- Type 30 in the "Layer depth under the ground surface" edit box
- Click "OK" button

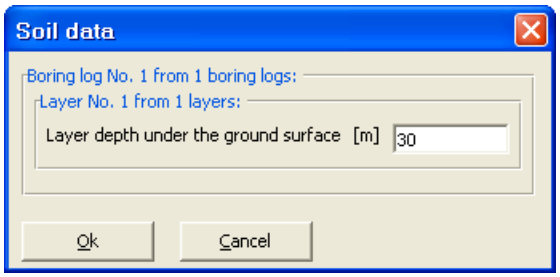


Figure 11-25 "Layer depth under the ground surface" edit box

To modify the groundwater depth under the ground surface

- Double-click on the groundwater level. The corresponding edit box appears allowing you to modify the groundwater depth under the ground surface, Figure 11-26. To neglect the uplift pressure on the raft, groundwater level is chosen at anywhere under the raft basement
- Type 30 in the "Groundwater depth under the ground surface" edit box
- Click "OK" button

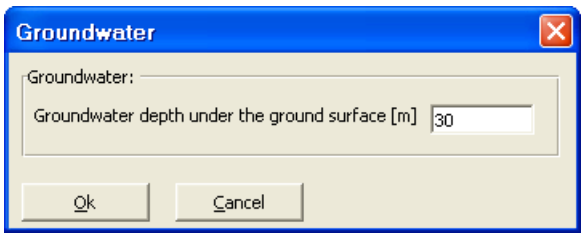


Figure 11-26 "Groundwater depth under the ground surface" edit box

To modify the label of a boring log

- Double-click on the label of the boring log. The corresponding edit box appears allowing you to modify the label of the boring log, Figure 11-27
- Type B1 in the edit box of Figure 11-27
- Press "Enter" key to consider the text

Figure 11-27 "Label of the boring log" edit box

After the user has completed the definition of all soil properties and parameters, the screen should look like the following Figure 11-28.

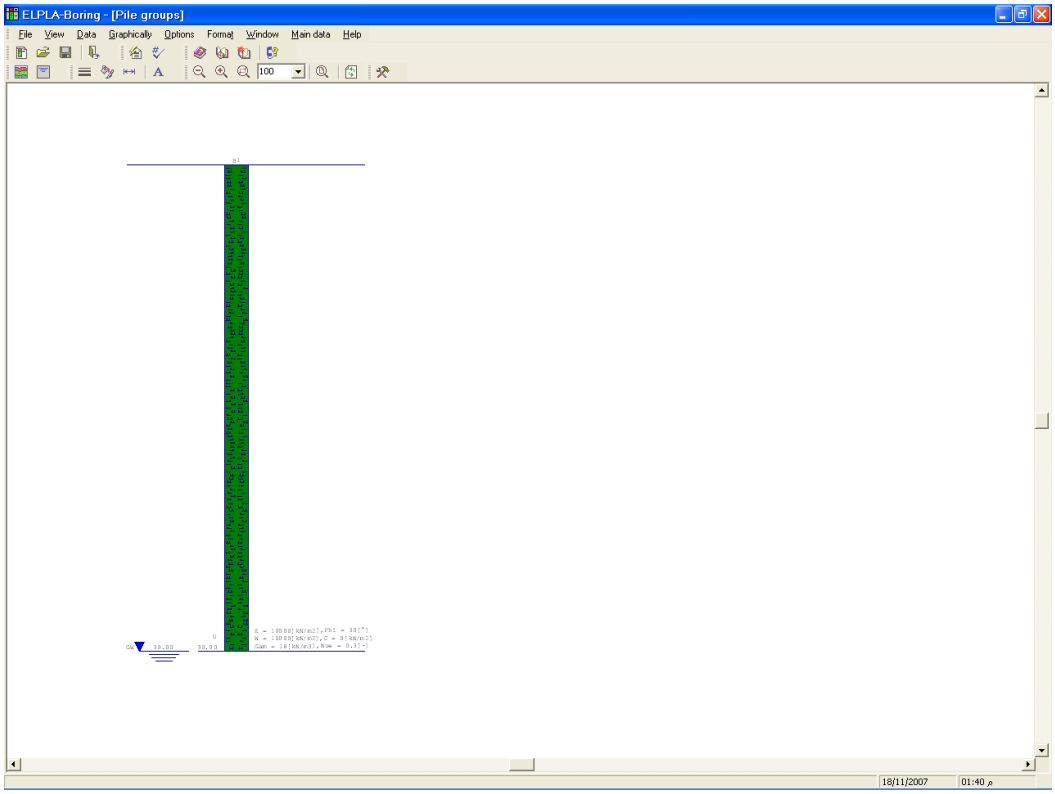


Figure 11-28 Boring log on the screen

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

- Choose "Save boring logs" command from "File" menu in Figure 11-28 to save the data of boring log
- Choose "Close boring logs" command from "File" menu in Figure 11-28 to close the *ELPLA-Boring* sub program and to return to the main window of *ELPLA-Data*

Note that the sign "√" 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 11-29 appears with default foundation properties. In the current example, it is required to define only the unit weight of the foundation. 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.

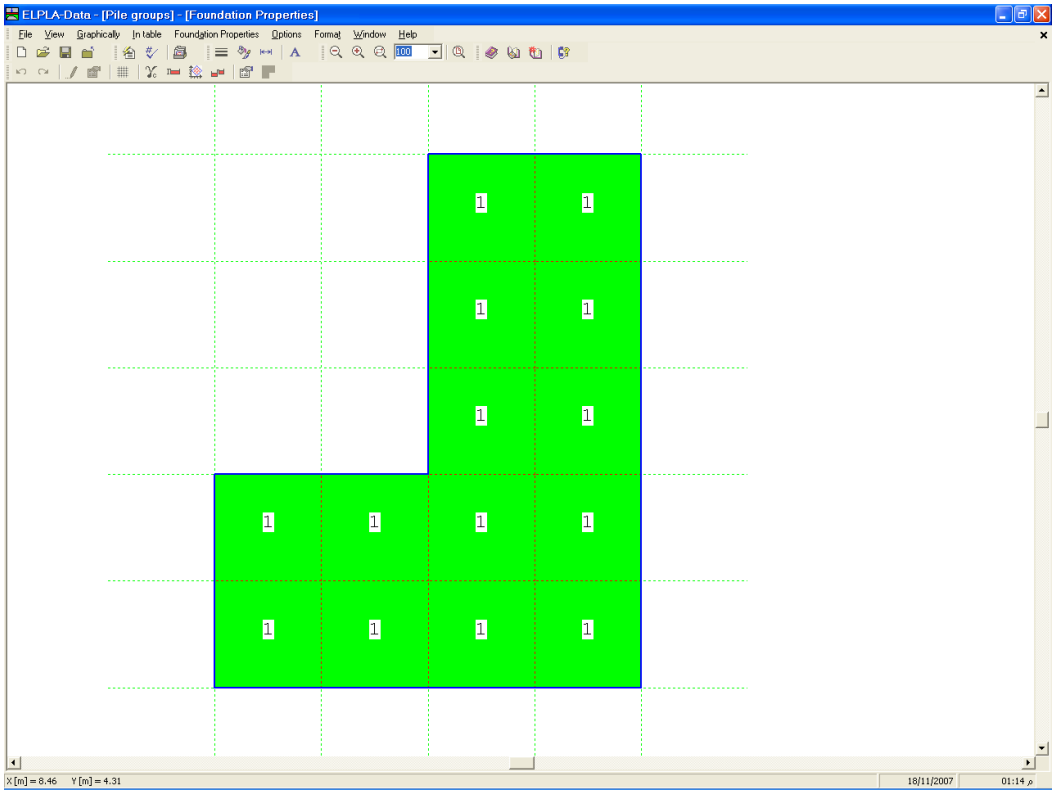


Figure 11-29 "Foundation properties" embedded program

To enter the unit weight of the foundation

- Choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 11-29. The following dialog box in Figure 11-30 with a default unit weight of 25 [kN/m^3] appears. Type 0 to neglect the foundation weight in the edit box "Unit weight of the foundation"
- Click "OK" button

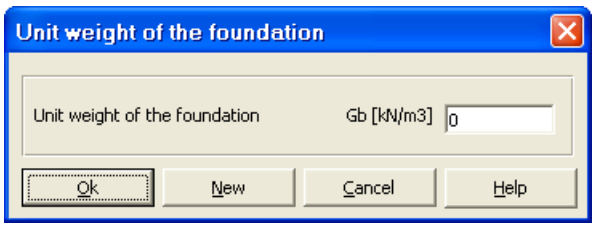


Figure 11-30 "Unit weight of the foundation" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 11-29 to save the foundation properties
- Choose "Close foundation properties" command to close the "Foundation properties" embedded program and to return to the main window of *ELPLA-Data*

Note that the sign "√" 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 11-31 appears.

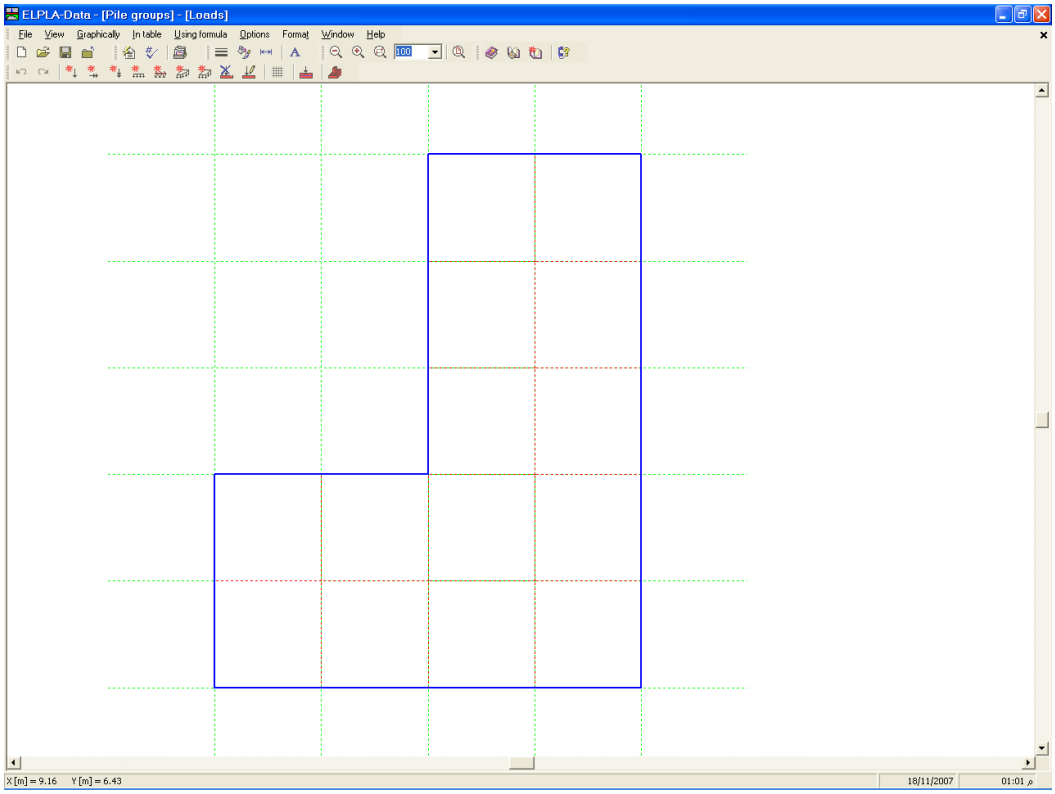


Figure 11-31 "Loads" embedded program

To enter the concentrated load choose "Point loads" command from "Graphically" menu in the window of Figure 11-31. When "Point loads" command is chosen, the cursor is changed from an arrow to a cross hair. Then, the load can be defined by clicking on the screen (grid). When the screen is clicked, the following dialog box in Figure 11-32 appears with the load value and coordinates.

In this dialog box

- Type 8000 in the "Load" edit box
- Type 5.2 in the "*x*-position" edit box
- Type 5.2 in the "*y*-position" edit box
- Click "OK" button

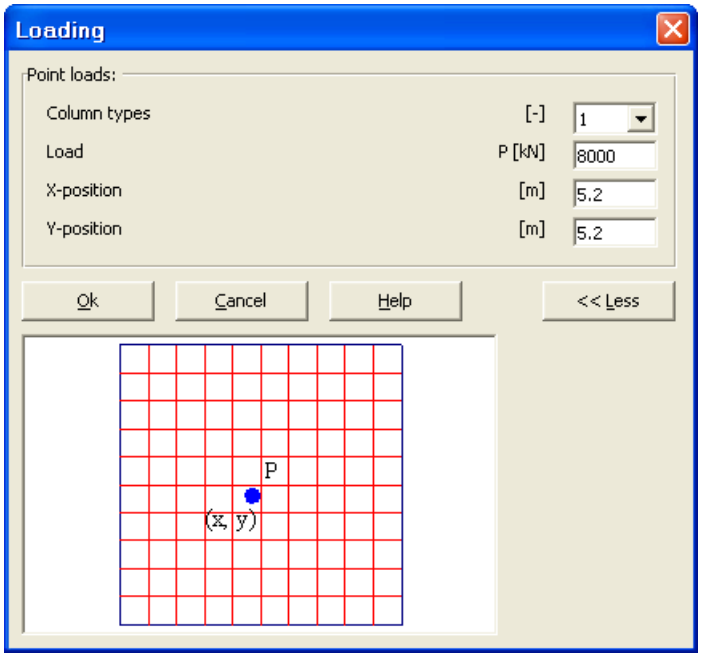


Figure 11-32 "Point loads *P*" dialog box

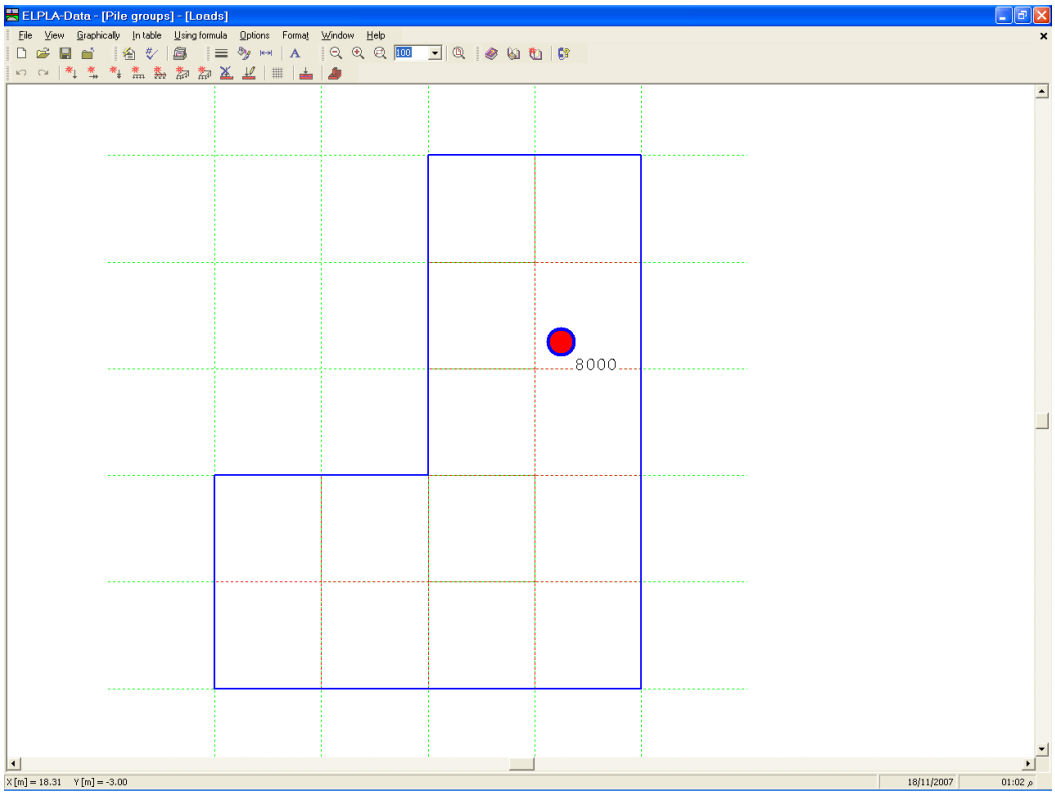


Figure 11-33 Loads on the screen

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

- Choose "Save loads" command from "File" menu in Figure 11-33 to save the load data
- Choose "Close loads" command from "File" menu in Figure 11-33 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

Note that the sign "√" is typed automatically beside the "Loads" command in the "Data" menu of *ELPLA-Data*.

Creating the project of the pile groups 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 11-34.

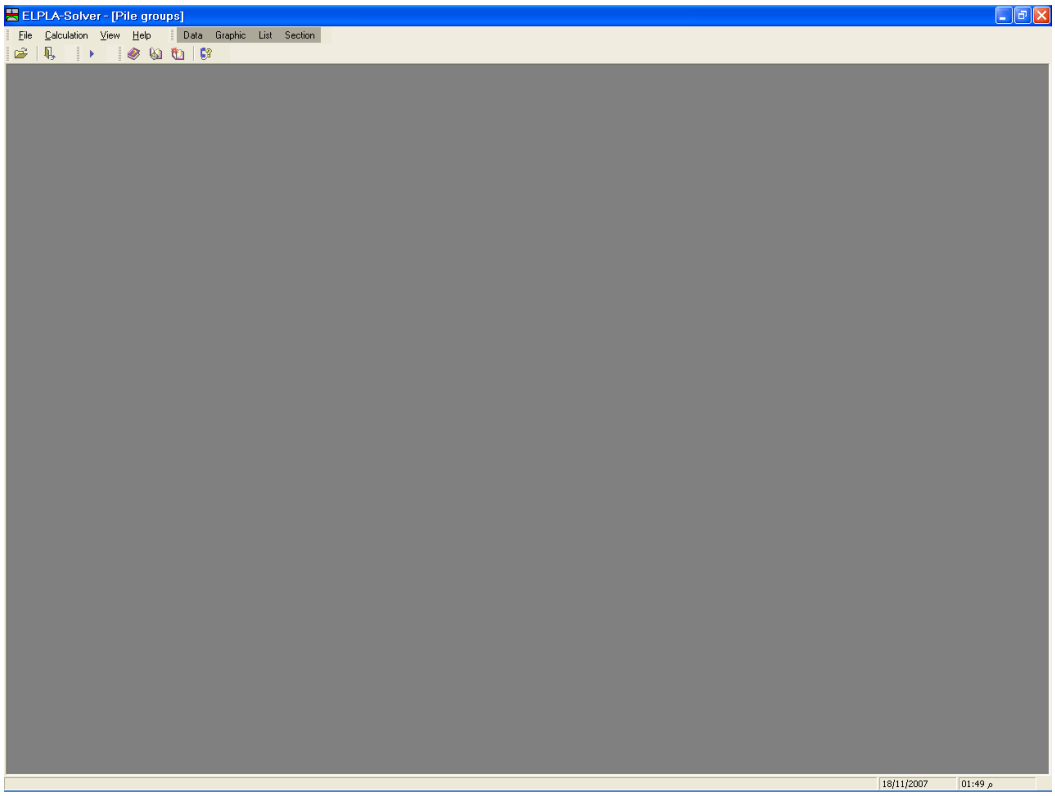


Figure 11-34 Opening screen of the sub program *ELPLA-Solver*

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
- Analysis of rigid pile groups
- 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 11-35 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.

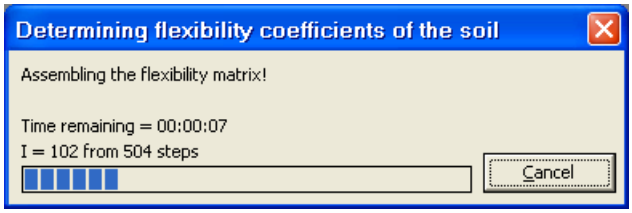


Figure 11-35 Analysis progress menu

Once the analysis is complete, a check menu of the solution appears, Figure 11-36. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

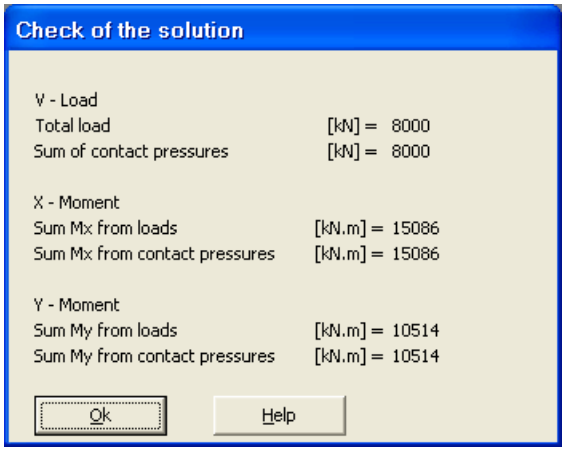


Figure 11-36 Menu "Check of the solution"

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 11-37. *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.

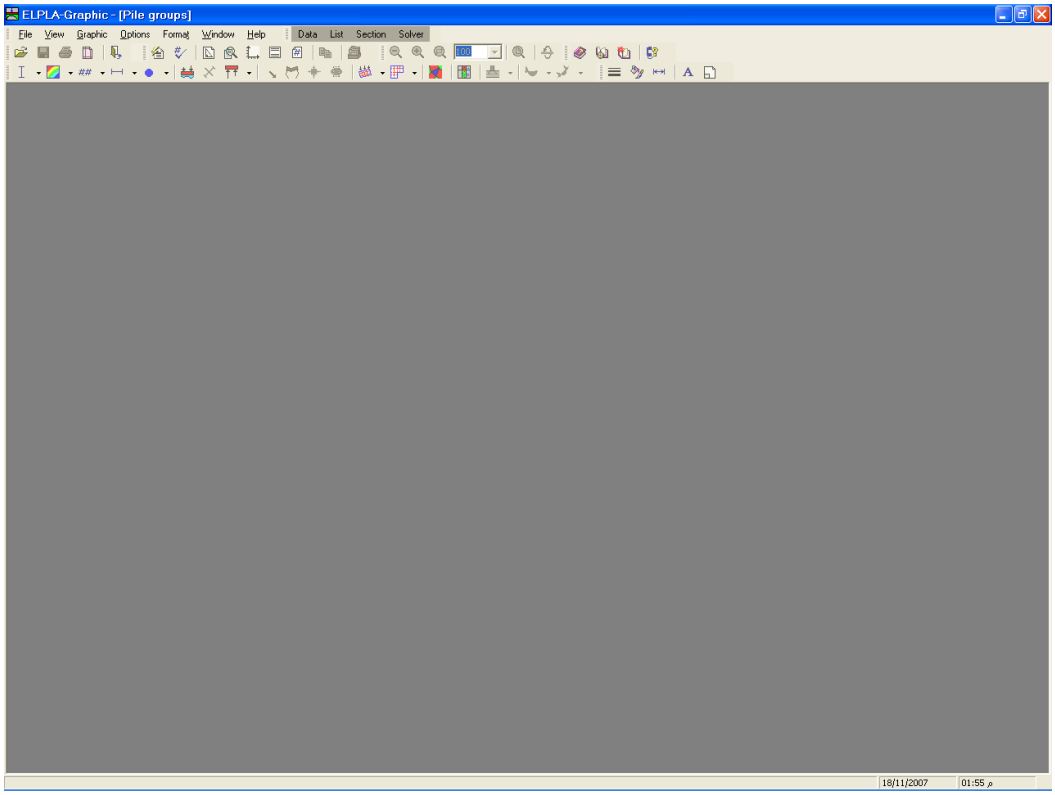


Figure 11-37 Opening screen of the sub program *ELPLA-Graphic*

To view the results for pile groups as circular diagrams

- Choose "Results as Circular Diagrams" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 11-38 appears
- In "Results as circular diagrams" option box, select "Settlements *s*" as a sample for the results to be displayed
- Click "OK" button

Settlements are now displayed as circular diagrams for piles as shown in Figure 11-39.

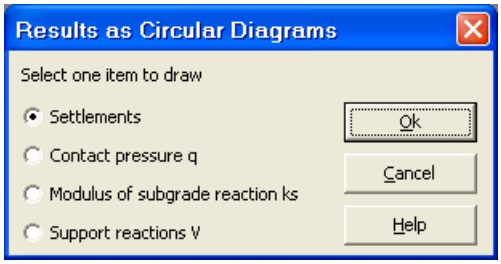


Figure 11-38 "Results as circular diagrams" option box

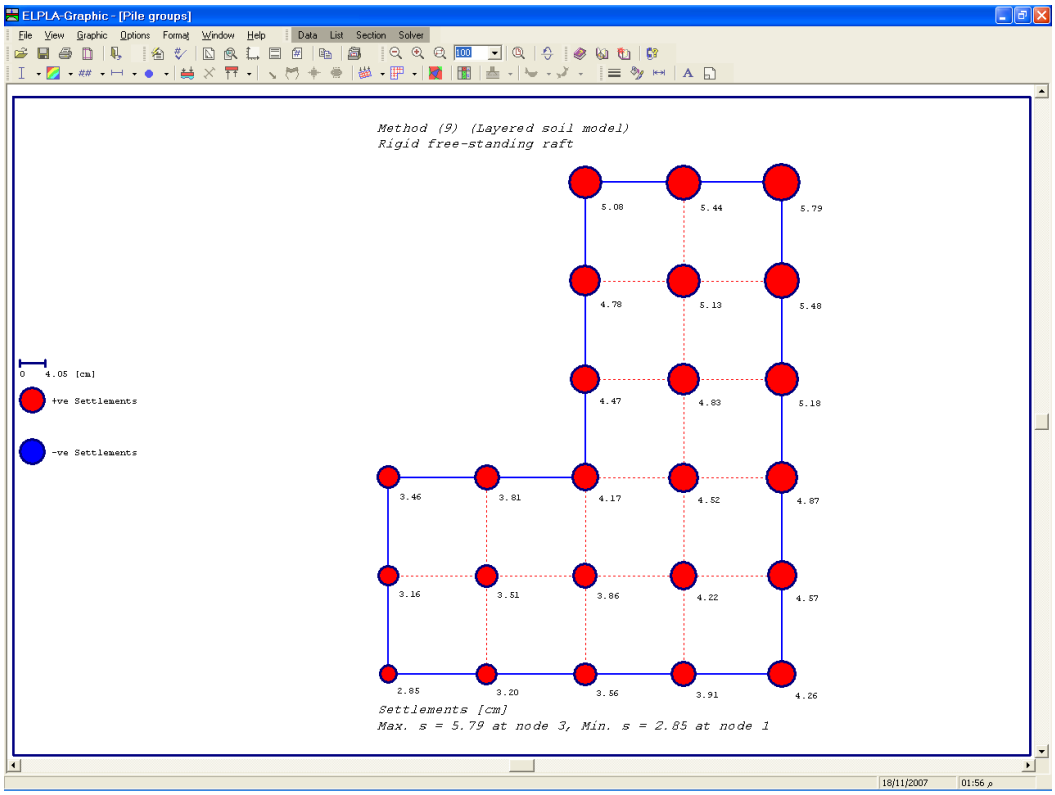


Figure 11-39 Settlements as circular diagrams for piles

4.2 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 11-40.

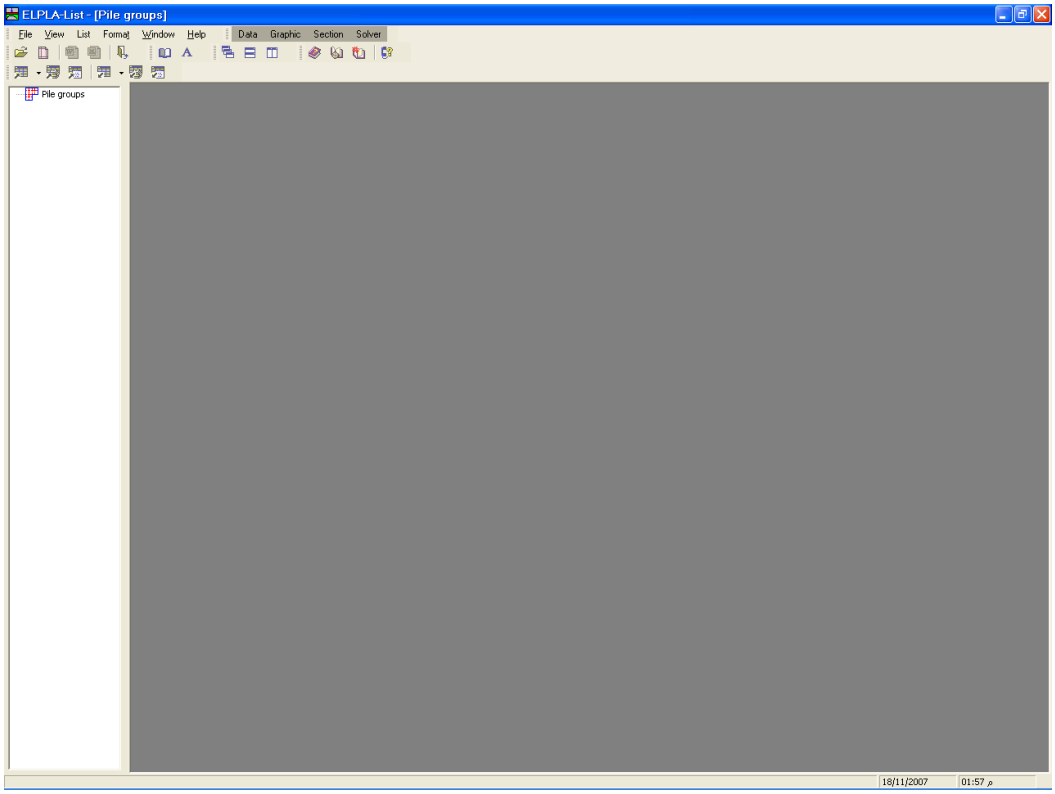


Figure 11-40 Opening screen of the sub program *ELPLA-List*

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 a table

- Choose "Display tables of results" command from "List" menu. The following option box in Figure 11-41 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 11-42)
- Choose "Send to Word" from "File" menu if you wish to export the text to the MS Word application, Figure 11-43

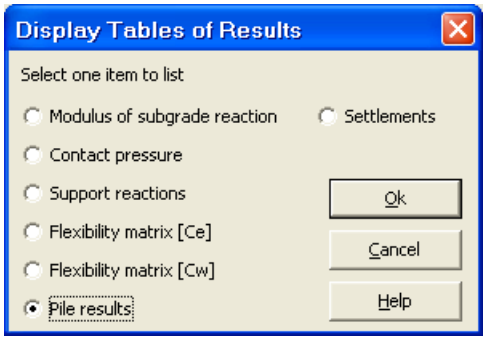


Figure 11-41 "Display tables of results" option box

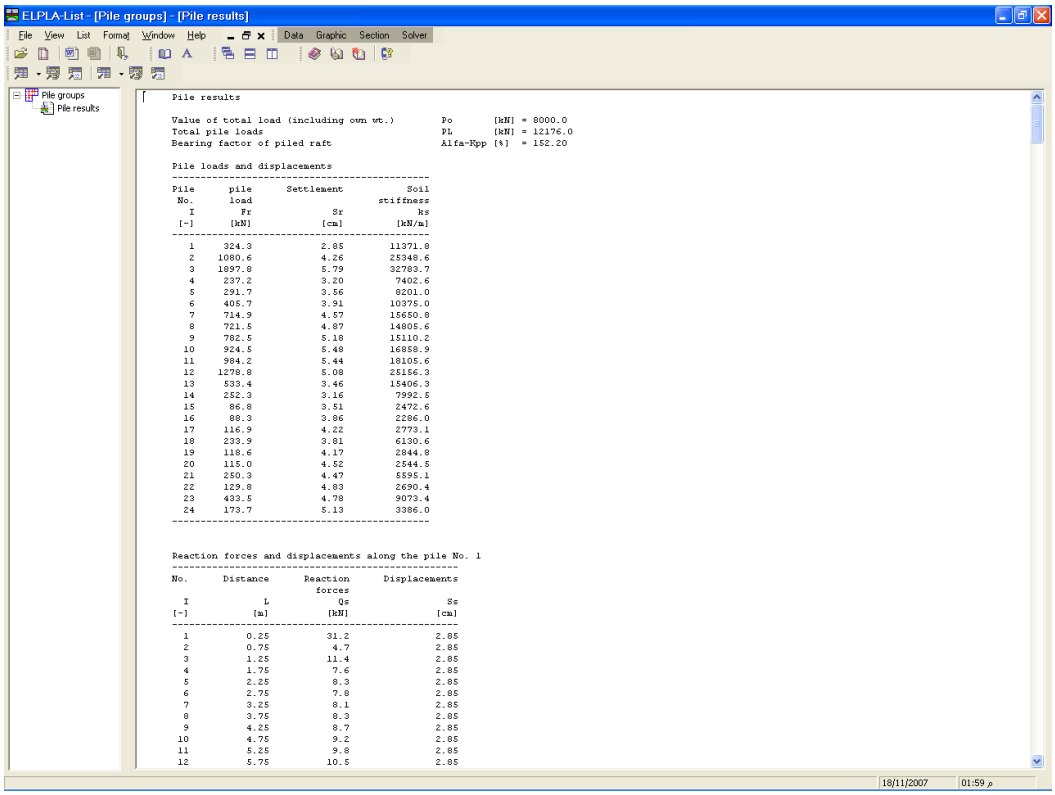
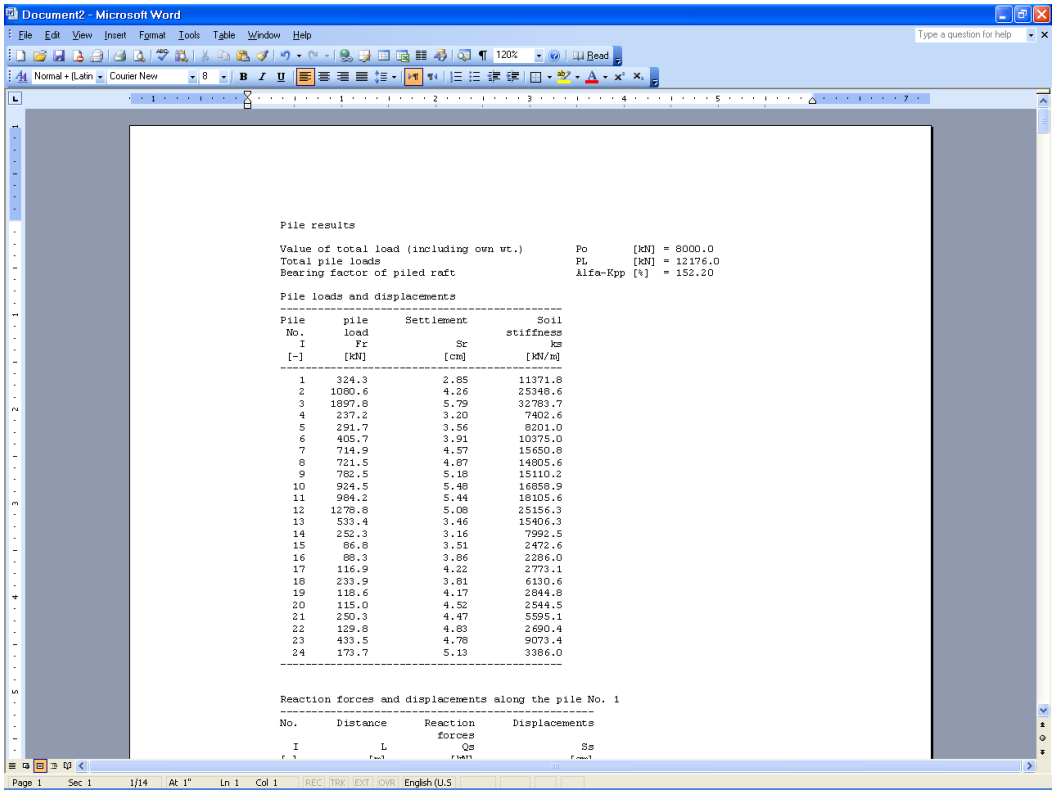


Figure 11-42 List of settlement results



Pile results

Value of total load (including own wt.) Po [kN] = 8000.0
Total pile loads PL [kN] = 12176.0
Bearing factor of piled raft Alfa-Kpp [%] = 152.20

Pile loads and displacements

Pile No. I [-]	pile load Fr [kN]	Settlement Sr [cm]	Soil stiffness ks [kN/m]
1	324.3	2.85	11371.8
2	1080.6	4.26	25348.6
3	1897.8	5.79	32783.7
4	237.2	3.20	7402.6
5	291.7	3.56	8201.0
6	405.7	3.91	10375.0
7	714.9	4.57	15650.8
8	721.5	4.87	14805.6
9	782.5	5.18	15110.2
10	924.5	5.48	16858.9
11	984.2	5.44	18105.6
12	1278.8	5.08	25156.3
13	533.4	3.46	15406.3
14	252.3	3.16	7992.5
15	86.8	3.51	2472.6
16	88.3	3.86	2286.0
17	116.9	4.22	2773.1
18	233.9	3.81	6130.6
19	118.6	4.17	2844.8
20	115.0	4.52	2544.5
21	250.3	4.47	5595.1
22	129.8	4.83	2690.4
23	433.5	4.78	9073.4
24	173.7	5.13	3386.0

Reaction forces and displacements along the pile No. 1

No. I	Distance L [m]	Reaction forces Qs [kN]	Displacements Ss [cm]

Figure 11-43 Exported results in MS Word

5 Index