Example 1

Analysis of an irregular shaped raft on irregular subsoil

Contents Page

1 Description of the problem

A simple example of an irregular shaped raft on irregular subsoil is selected to illustrate some of the essential features of *ELPLA* for analyzing rafts.

1.1 Loads

The raft carries many types of external loads: concentrated loads [kN], uniform load [kN/m^2], line load [kN/m] and moments [kN.m] in both *x*- and *y*-directions as shown in Figure 1-1 and in Table 1-1 to Table 1-5.

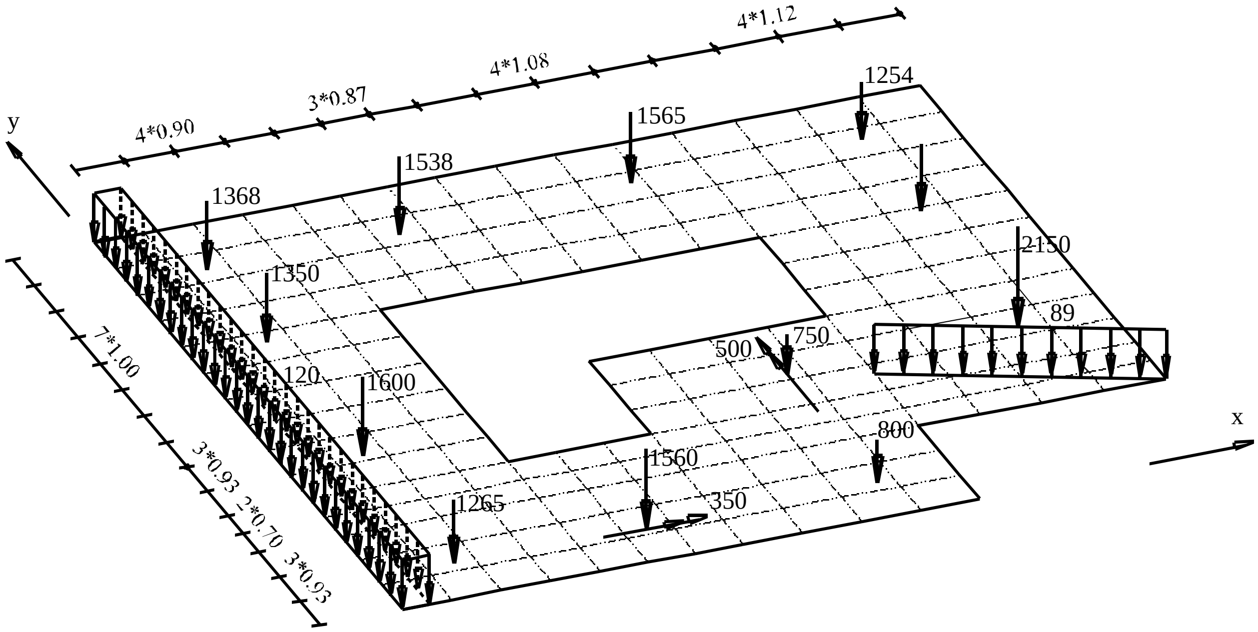


Figure 1-1 Raft dimensions [m] and loads

1.2 Raft material and thickness

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

Young's modulus	E_b	= 2 * 10^7	[kN/m^2]
Poisson's ratio	ν_b	= 0.25	[-]
Unit weight of raft material	γ_b	= 0.0	[kN/m^3]
Raft thickness	*d*	= 0.5	[m]

Unit weight of raft material is chosen to be γ_b = 0.0 [kN/m^3] to neglect the own weight of the raft in the analysis.

1.3 Soil properties

Three boring logs characterize the subsoil under the raft. Each boring has three layers with different soil materials. The Moduli of Elasticity of the three layers for loading are E_{s1} = 9 500 [kN/m^2], E_{s2} = 22 000 [kN/m^2] and E_{s3} = 120 000 [kN/m^2] while for reloading are W_{s1} = 26 000 [kN/m^2], W_{s2} = 52 000 [kN/m^2] and W_{s3} = 220 000 [kN/m^2]. *Poisson's* ratio is 0.3 [-] for all soil layers. Unit weight of the soil above the ground water is γ_s = 19 [kN/m^3] while that under the ground water is γ'_s = 9 [kN/m^3]. The foundation depth d_f = 2.7 [m], the ground water depth under the ground surface is *GW* = 1.5 [m]. Figure 1-2 shows boring logs and locations.

1.4 Method of analysis

It is required to analyze the raft according to the following soil models and numerical calculation methods:

- Layered soil medium - Continuum Model
- Modulus of compressibility method for an elastic raft on layered soil medium
  (Solving system of linear equations by iteration-method 6)

In the analysis the following items will be taken into account:

- The effect of reloading of the soil due to the overburden pressure
- The effect of water pressure on the raft
- The irregularity of the subsoil under the raft using the interpolation method (Figure 1-2)

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 is well documented in the User’s Guide of *ELPLA*.

Table 1-1 Point loads *P*

Load No. I [-]	Load value P [kN]	x-position x [m]	y-position y [m]
1	1265	1.5	1.4
2	1600	1.5	5.5
3	1350	1.5	9.9
4	1368	1.5	12.6
5	1560	5	1.4
6	1538	5	12.6
7	800	9.2	1.4
8	750	9.2	5.5
9	1565	9.2	12.6
10	2150	13.4	5.5
11	1450	13.4	9.9
12	1254	13.4	12.6

Table 1-2 Moment *Mx*

Moment No. [-]	Moment value Mx [kN.m]	x-position x [m]	y-position y [m]
1	350	5	1.4

Table 1-3 Moment *My*

Moment No. [-]	Moment value My [kN.m]	x-position x [m]	y-position y [m]
1	500	9.2	5.5

Table 1-4 Line load *pl*

Load No. I [-]	Load value Pl [kN/m]	Load start x1 [m]	Load start y1 [m]	Load end x2 [m]	Load end y2 [m]
1	89	10.5	4.8	15	2.8

Table 1-5 Distributed load *p*

Load No. I [-]	Load value P [kN/m2]	Load start x1 [m]	Load start y1 [m]	Load end x2 [m]	Load end y2 [m]
1	120	0	0	0.5	13.97

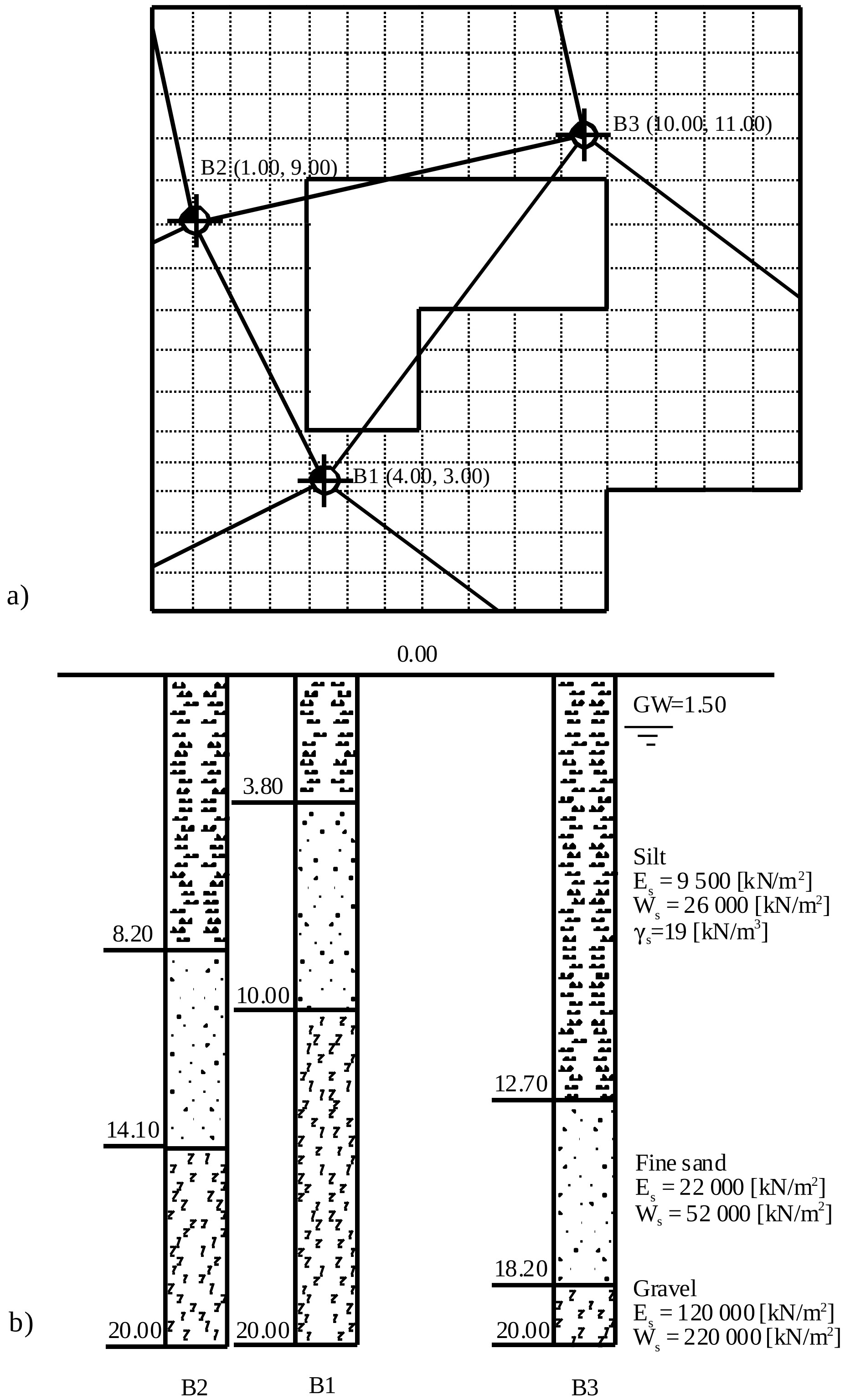


Figure 1-2 a) Boring locations and interpolation zones
b) Boring logs B1 to B3

2 Creating the project

In this section the user will learn how to create a project for analyzing raft foundation. The project 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

Choose "New project" command from the "File" menu of *ELPLA-Data*. The following "Calculation methods" wizard appears, Figure 1-3. 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 1-3).

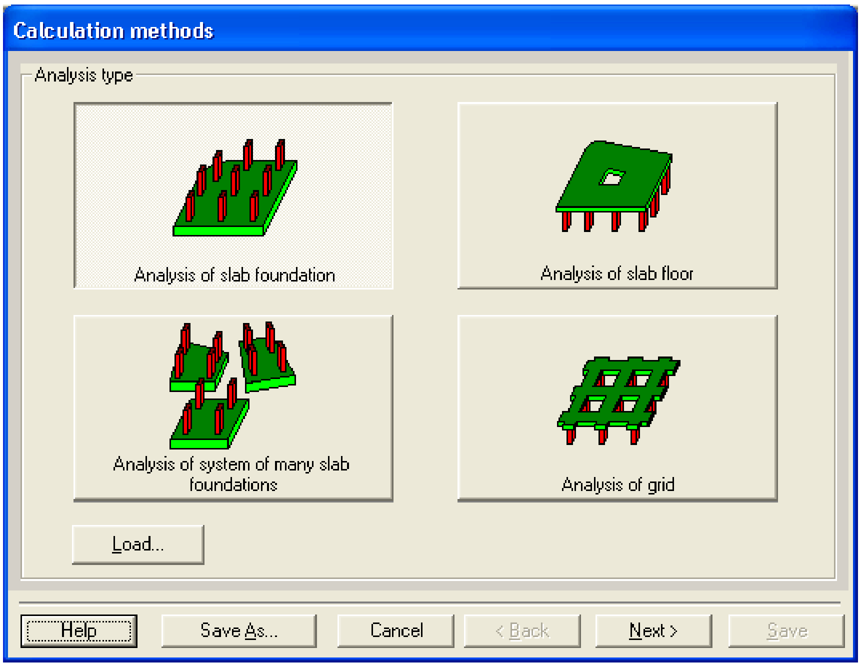


Figure 1-3 "Analysis type" form

In the "Analysis type" form in Figure 1-3 define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a foundation problem, select "Analysis of slab foundation" then click "Next" button to go to the next form. After clicking "Next" button, the "Calculation methods" form appears, Figure 1-4.

To define the calculation method

- Select the calculation method "6-Modulus of Compressibility (Iteration)"
- Select subsoil model "Layered soil model"
- Click "Next" button to go to the next form

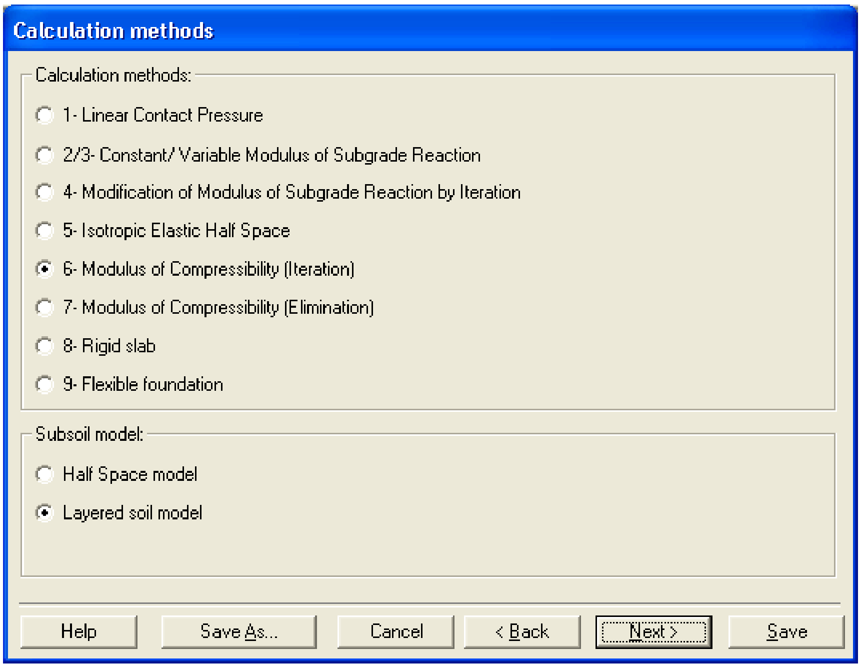


Figure 1-4 "Calculation methods" form

The next form is the "System symmetry" in Figure 1-5. In this form select "Unsymmetrical system" and then click "Next" button to go to the next form.

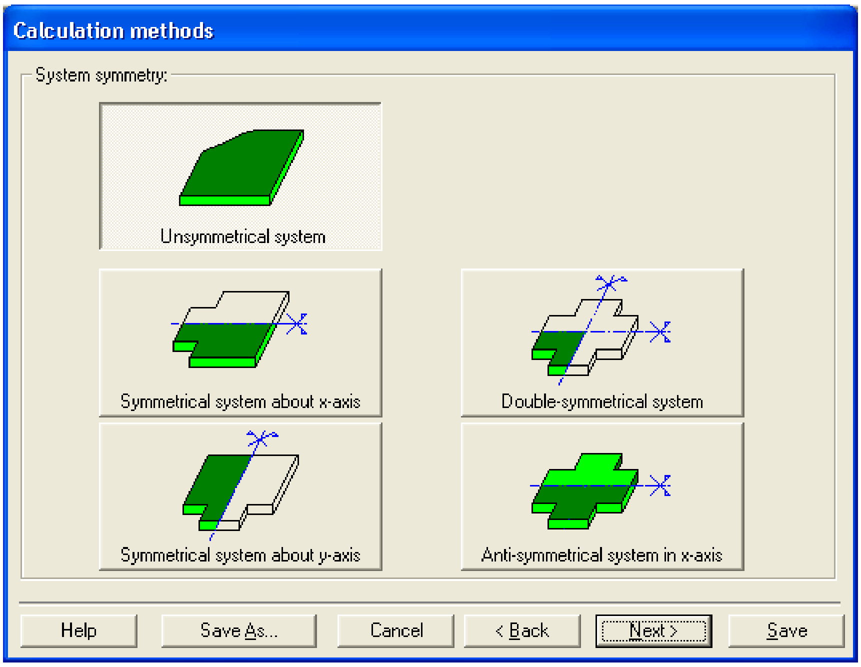


Figure 1-5 "System symmetry" form

The last form in the wizard is the "Options" form, Figure 1-6. 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.

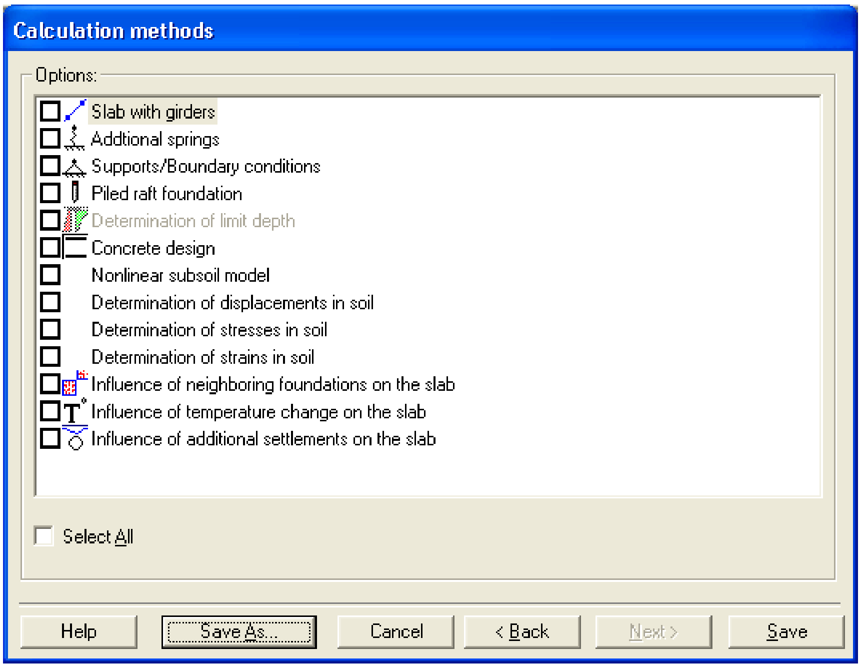


Figure 1-6 "Options" form

After clicking "Save" button, the "Save as" dialog box appears, Figure 1-7. In this dialog box type a file name for the current project in "File name" edit box. For example type "Example". *ELPLA* will use automatically this file name in all reading and writing processes.

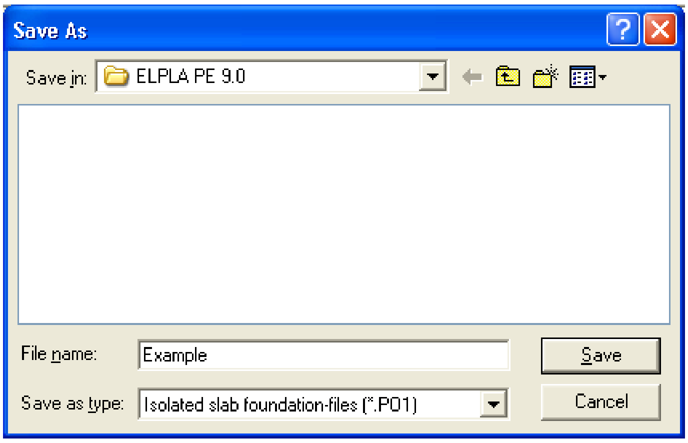


Figure 1-7 "Save as" dialog box

Click "Save" button to complete the definition of the calculation method and the file name of the project. *ELPLA* will activate the "Data" menu. Also the file name of the current project [Example] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, see Figure 1-8.

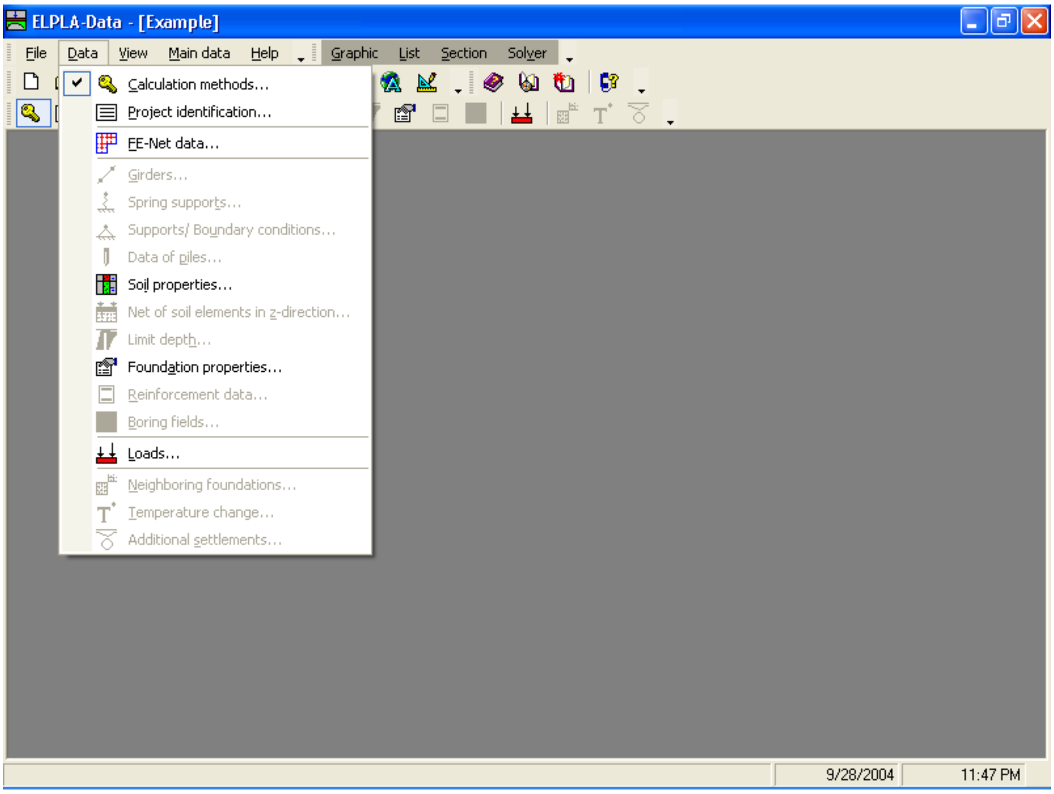


Figure 1-8 *ELPLA-Data* after defining the calculation method

In the "Data" menu of *ELPLA-Data*, 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 1-8). *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.

2.2 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 "Data" menu. The dialog box in Figure 1-9 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box:
  "Analysis of an irregular raft on irregular subsoil"
- Type the date of the project in the "Date" edit box
- Type the word "Example" in the "Project" edit box
- Click "Save" button

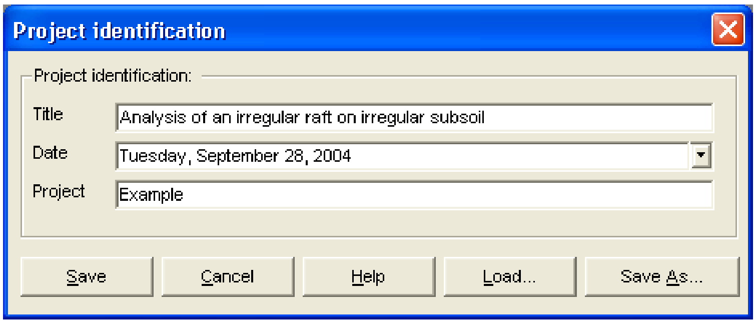


Figure 1-9 "Project identification" dialog box

2.3 FE-Net data

For the given problem, the raft has irregular shape and is divided into 15 * 15 elements. Element size in both *x*- and *y*-directions is variable as shown in Figure 1-1. *ELPLA* has different procedures for defining the FE-Net. The easy procedure to define the FE-Net of this raft is generating a mesh for the entire area first and then removing the unnecessary nodes to get the foundation shape.

To define the FE-Net for this raft, choose "FE-Net data" command from the "Data" menu. "FE-Net generation" wizard appears as shown in Figure 1-10. This wizard will guide you through the steps required to generate a FE-Net. As shown in Figure 1-10, 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.

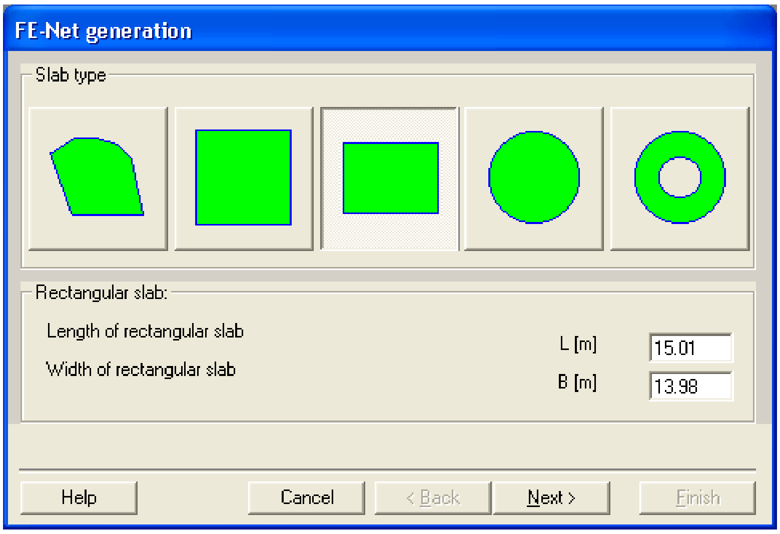


Figure 1-10 "FE-Net generation" wizard with "Slab type" form

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 1-11. *ELPLA* can deal with various types of generations with triangle and/ or rectangular elements. Choose the first type of rectangular elements, then click "Next" button.

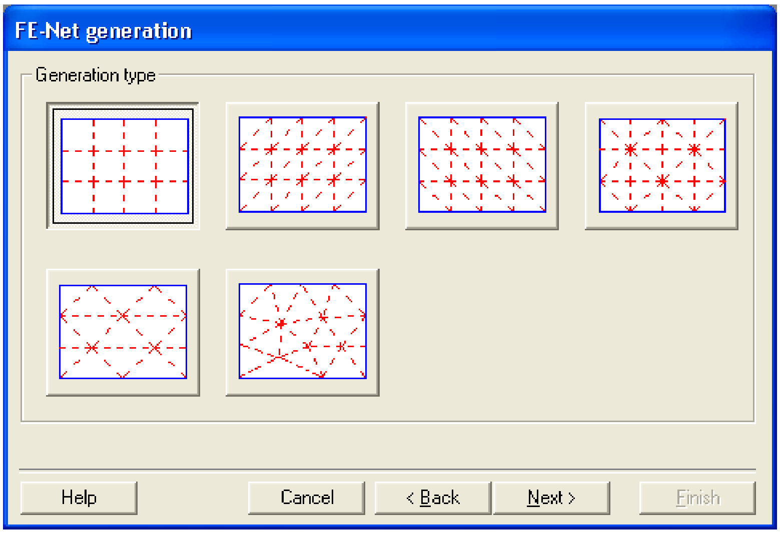


Figure 1-11 "Generation type" form

The next form of the "FE-Net generation" wizard is the "Grid definition" form. In this form, the default values of constant element size appear, Figure 1-12. To edit the variable grid spacing in *x*-direction, do the following steps in "Grid in *x*-direction" frame:

- Uncheck the "Constant grid spacing" check box. "Grid spacing" button will be activated
- Click the button "Grid spacing". "Grid spacing in *x*-direction" list box appears, Figure 1-13. In this list box the spacing width can be entered individually for every space
- Enter the grid spacing in *x*-direction in this list box
- Click "OK" button to return to "Net of finite elements" dialog box

To edit the variable grid spacing in *y*-direction, repeat the previous steps. After that, click "Finish" button in "FE-Net generation" wizard to see the FE-Net on the screen.

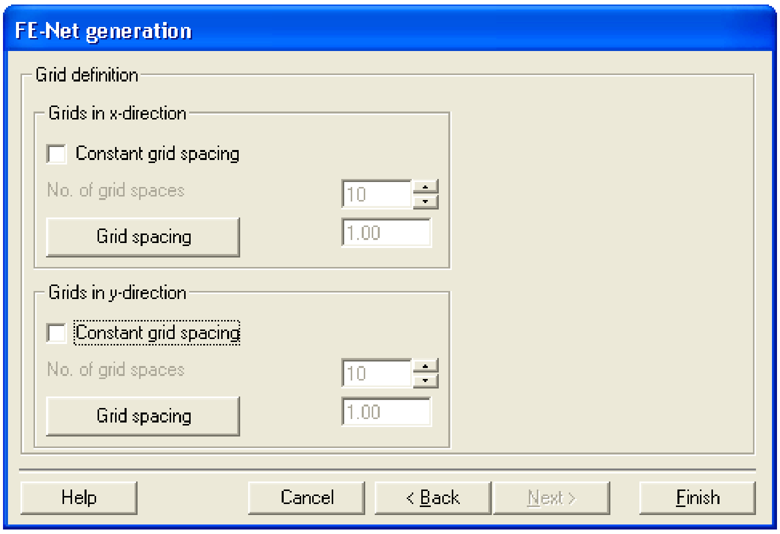


Figure 1-12 "Grid definition" form

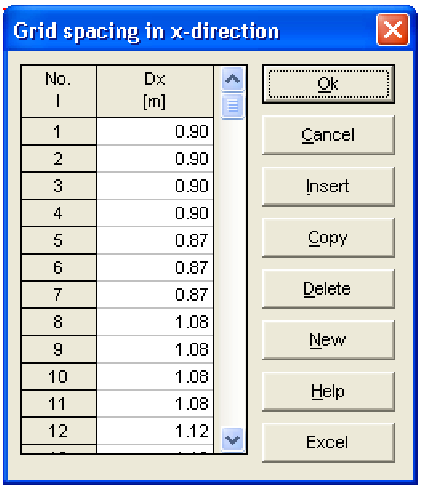


Figure 1-13 "Grid spacing in *x*-direction" list box

Deleting nodes from the FE-Net

To select the unnecessary nodes which 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 1-14. To remove the selected nodes, choose "Remove nodes" command from the "Graphically" menu. The action of this command is indicated in Figure 1-15. To leave the graphic mode, press "Esc" key.

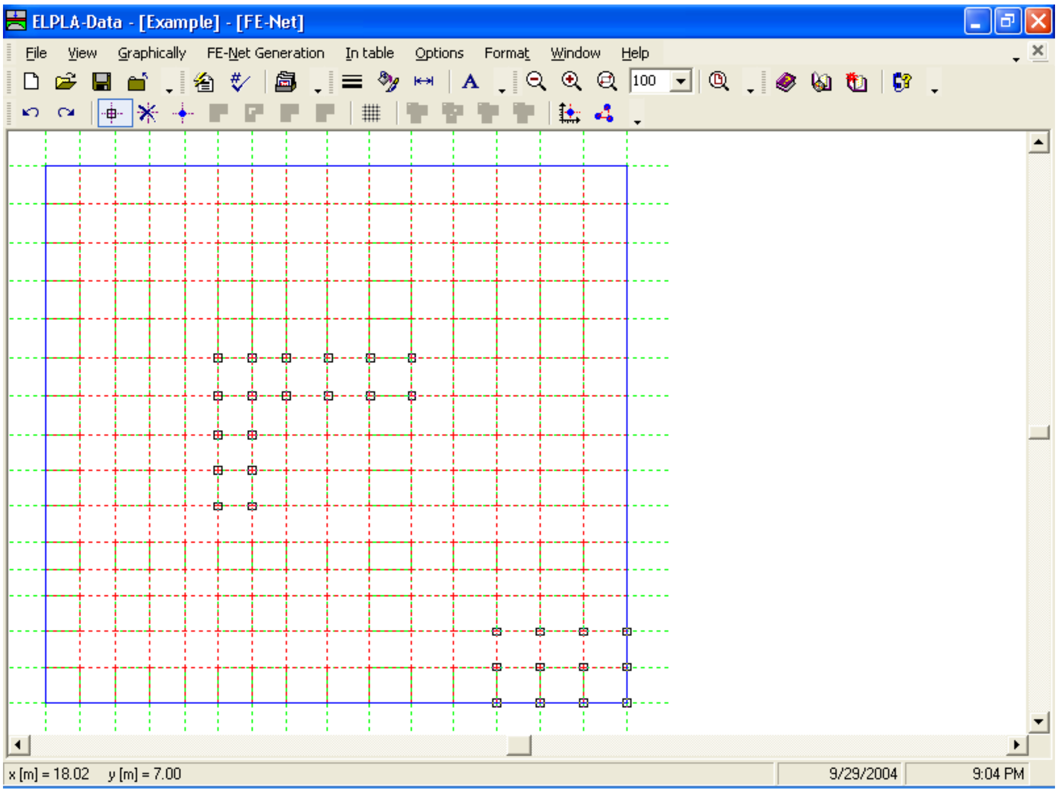


Figure 1-14 Generated FE-Net after selecting the unnecessary nodes

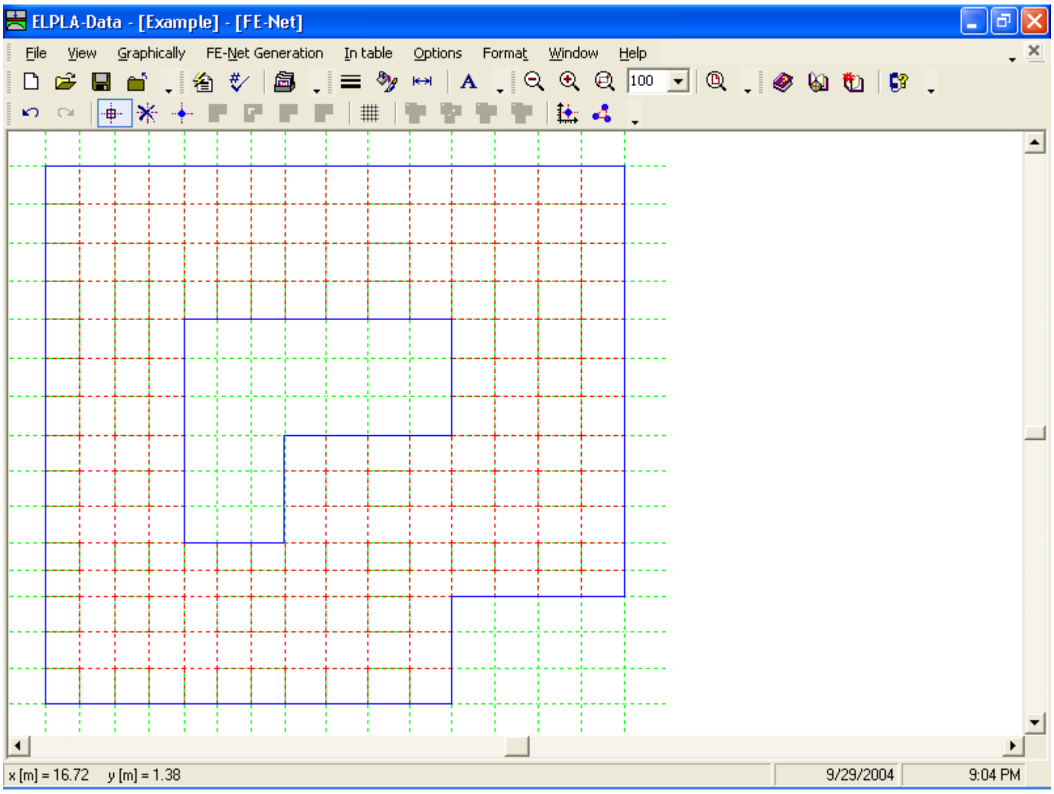


Figure 1-15 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save FE-Net" command from "File" menu in Figure 1-15 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 1-15 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 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 Layered Soil Model. This model requires that the subsoil has to be defined by boring logs. In the example, three boring logs at different locations define the soil under the raft. Each boring log has multi-layers with different soil materials. The geotechnical data for each layer are unit weight of the soil γ_s and modulus of Elasticity for loading E_s and reloading W_s.

To define the soil properties, choose "Soil properties" command from "Data" menu. The following sub program in Figure 1-16 appears with a default-boring log.

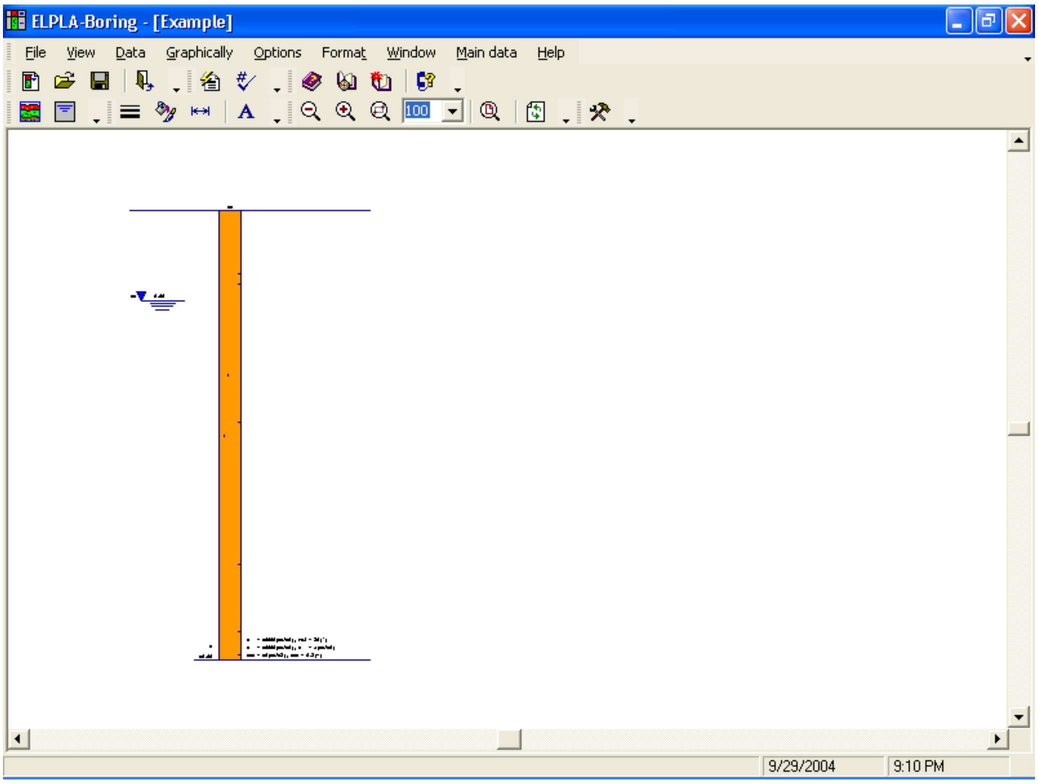


Figure 1-16 *ELPLA-Boring* with a default-boring log

In Figure 1-16, 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 three boring logs of the current example

- Choose "Soil data" command from "Data" menu in the window of Figure 1-16. The following dialog box in Figure 1-17 with default-boring log data appears

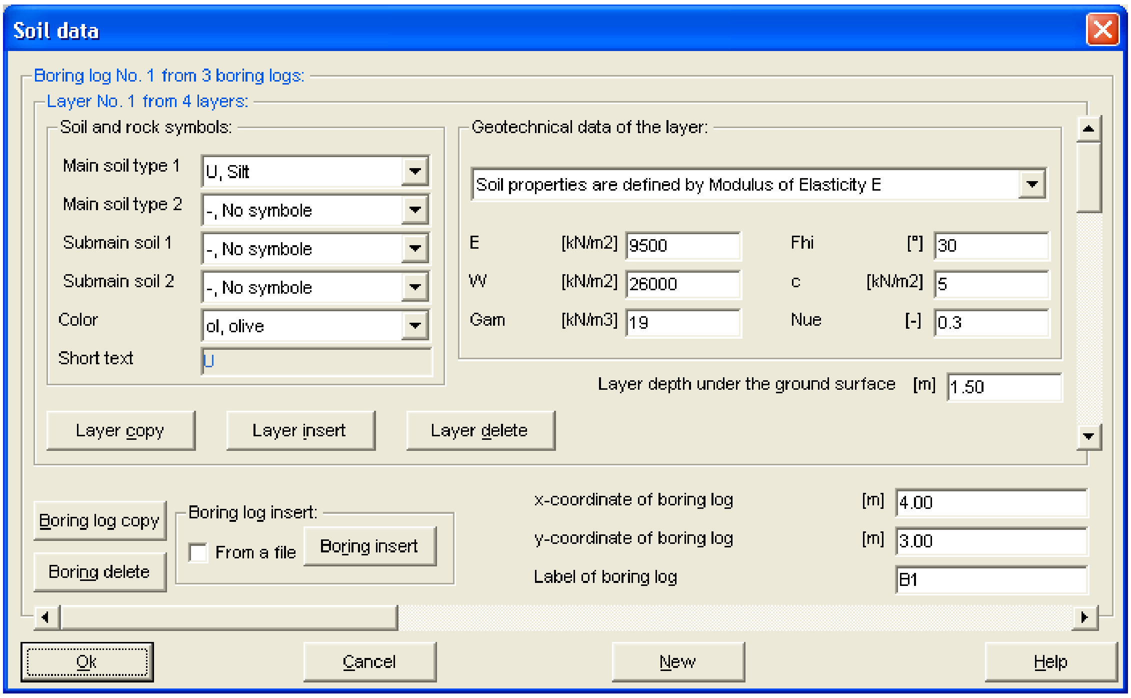


Figure 1-17 "Soil data" dialog box with default-boring log data

In the "Geotechnical data of the layer" dialog group box in Figure 1-17, define the geotechnical data of the first soil layer of the first boring log as follows:

E_s	= 9 500	[kN/m^2]
W_s	= 26 000	[kN/m^2]
Gam	= 19	[kN/m^3]

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.

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 for all boring logs is taken to be 1.5 [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 in Figure 1-17. 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 of the first boring log

- Click "Layer copy" button in Figure 1-17. 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 19 [kN/m^3] to 9 [kN/m^3]
- Change the value of the layer depth under the ground surface from 1.5 [m] to 3.8 [m]

To enter the fine sand and gravel layers

- Click twice "Layer insert" button in Figure 1-17, two layers 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 1-17, define the geotechnical data of the fine sand layer as follows:

Es	= 22 000	[kN/m^2]	Phi	= 30	[°]
Ws	= 52 000	[kN/m^2]	*c*	= 5	[kN/m^2]
Gam	= 9	[kN/m^3]			

- Select "fs, Fine sand" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box
- Type 10 in "Layer depth under the ground surface" edit box
- Use the vertical scrollbar to move to the fourth soil layer
- Type the following data for the gravel layer:

Es	= 22 000	[kN/m^2]	Phi	= 30	[°]
Ws	= 52 000	[kN/m^2]	*c*	= 5	[kN/m^2]
Gam	= 9	[kN/m^3]			

- Select "G, Gravel" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box
- Type 20 in the "Layer depth under the ground surface" edit box

Note that 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 $\gamma_s * d_f$. 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/m^3].

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

To enter the boring coordinates and the label

- Type 4 for *x*-coordinate in "*x*-coordinate of boring log [m]" edit box
- Type 3 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

Now all data and parameters for the first boring log have been entered. The next step is to enter the data of the other two boring logs. As the three boring logs contain the same soil layers, data of the other two boring logs are created by first copying the data of the first boring log and then modifying boring logs individually. Only layer depths, boring coordinates and labels are required to be modified.

To create the other two boring logs, click twice "Boring log copy" button in Figure 1-17. Two boring logs with the same data and parameters of the first boring log will be copied.

Modifying data of boring logs

Modifying boring coordinates is carried out only numerically while modifying the other data of boring logs may be carried out either numerically or graphically. In this example all data will be modified numerically.

To modify the boring coordinates and labels

- Use the horizontal scrollbar to switch to the second boring log. Boring log No. will be typed automatically at the upper-left corner of the main dialog box of boring logs as a head title
- Type 1 as *x*-coordinate in "*x*-coordinate of boring log [m]" edit box in Figure 1-17
- Type 9 as *y*-coordinate in "*y*-coordinate of boring log [m]" edit box in Figure 1-17
- Type B2 as a label name for the second boring in "Label of boring log" dialog box in Figure 1-17
- Use the vertical scrollbar to move from a layer to another. Then modify the layer depth under the ground surface for each layer
- In "Layer depth under the ground surface" edit box in Figure 1-17 type the following values for layer depths:

Layer depth under the ground surface (2nd layer)	8.2	[m]
Layer depth under the ground surface (3rd layer)	14.1	[m]
Layer depth under the ground surface (4th layer)	20	[m]

Repeat the previous steps to modify the boring data for the third boring log. Data, which are required to be modified for the third boring log, are:

x-coordinate of boring log	10	[m]
y-coordinate of boring log	11	[m]
Label of boring log	B3	
Layer depth under the ground surface (2nd layer)	12.7	[m]
Layer depth under the ground surface (3rd layer)	18.2	[m]
Layer depth under the ground surface (4th layer)	20	[m]

Now, after finishing the creation of boring logs, click "OK" button in "Soil data" dialog box in Figure 1-17 to see the defined boring logs on the screen where the user can control or modify the input data and parameters. As a default plot parameter, *ELPLA* displays only the first boring log on the screen (Figure 1-18).

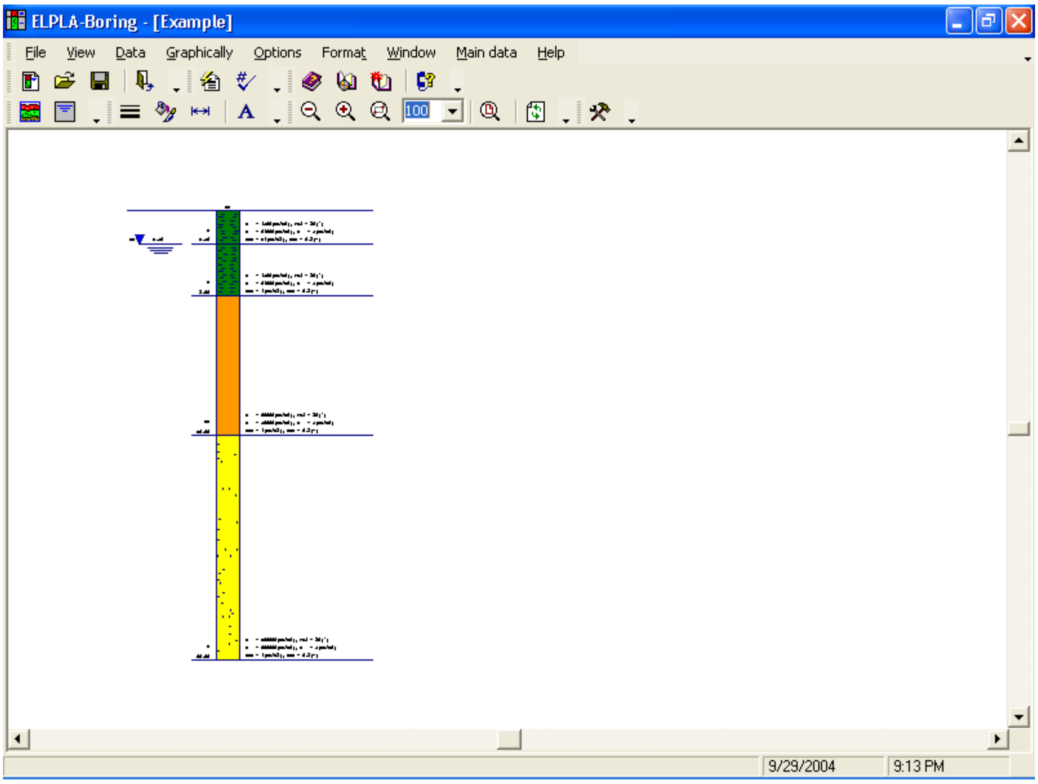


Figure 1-18 First boring log on the screen

To display all boring logs or specified boring logs on the screen, choose "Drawing boring logs" command from "Graphically" menu in Figure 1-18. The following list box in Figure 1-19 appears.

To select the boring logs you want to display

- Select the boring log that is required to be displayed from the list of the available boring logs in Figure 1-19
- Click "Boring insert" button. Double clicking on the required boring log in the list of the available boring logs gives the same action. Removing a boring log from the drawing list is carried out by double clicking on that boring log in the list of the selected boring logs
- Click "OK" button in Figure 1-19. The selected boring logs appear on the screen to control or modify the boring data graphically, Figure 1-20

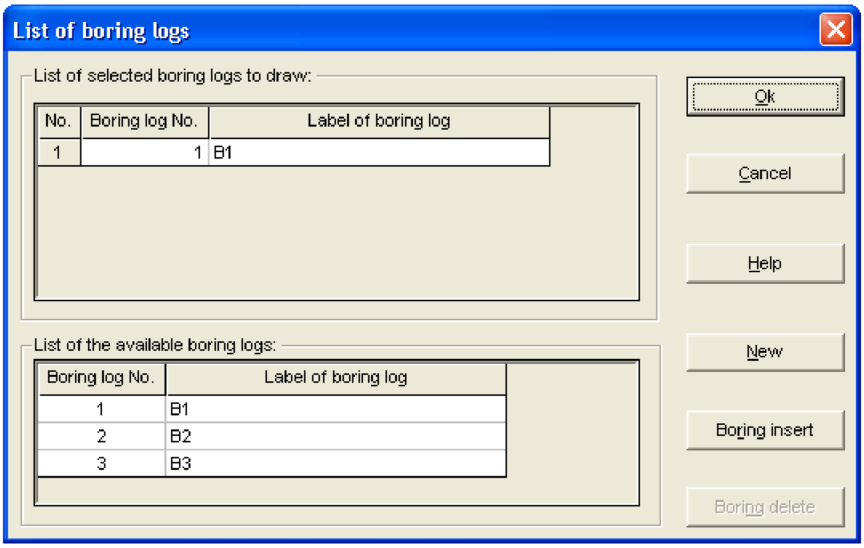


Figure 1-19 "List of boring logs" list box

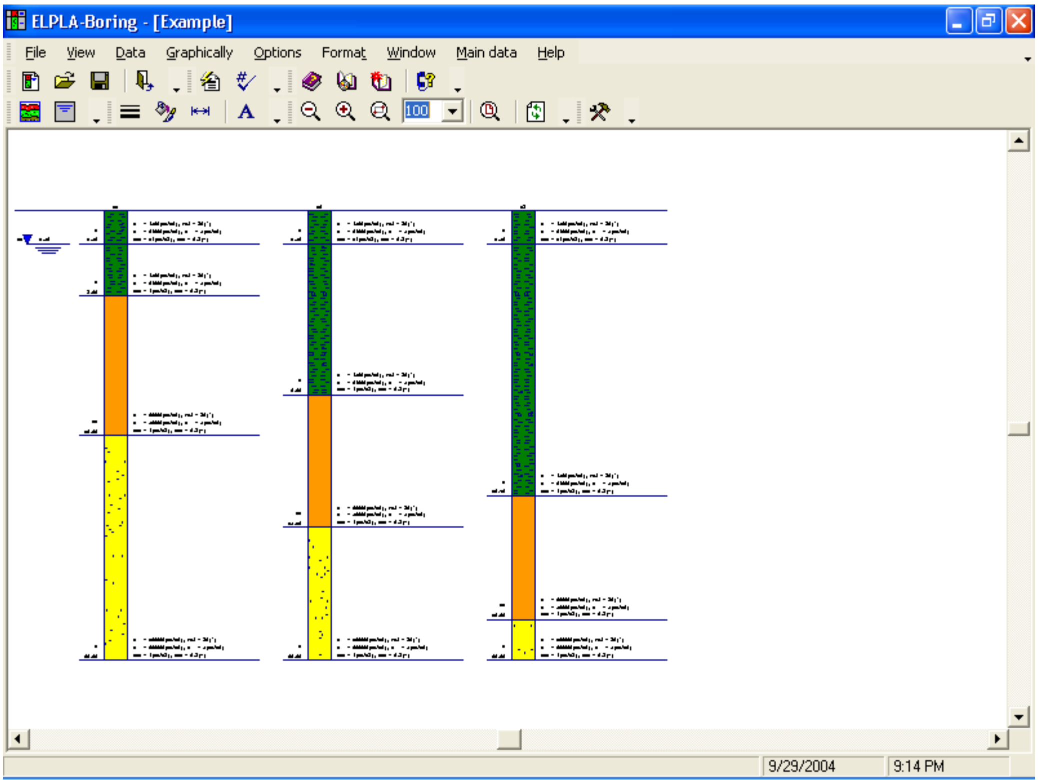


Figure 1-20 Boring logs on the screen

To enter the main soil data for all layers, choose "Main soil data" command from "Data" menu in Figure 1-20. The following dialog box in Figure 1-21 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 1-21 enter the settlement reduction factor α [-] and the groundwater depth under the ground surface G_w [m]. Then click "OK" button.

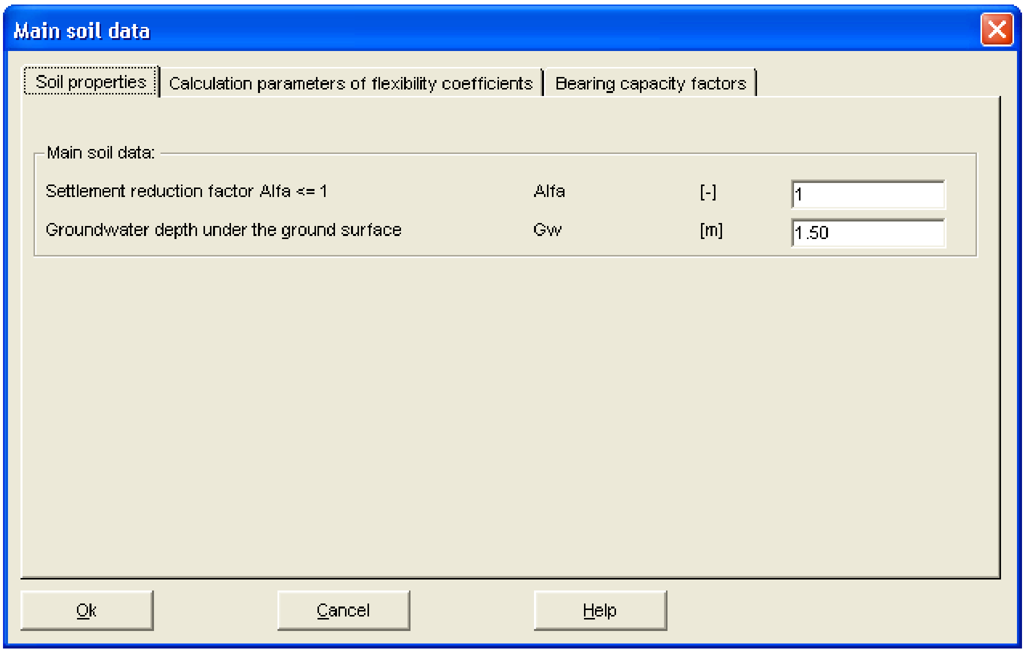


Figure 1-21 "Main soil data" dialog box

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

- Choose "Save boring logs" command from "File" menu in Figure 1-20 to save the data of boring logs
- Choose "Close boring logs" command from "File" menu in Figure 1-20 to close the sub program *ELPLA-Boring* and 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*.

2.5 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu. The following embedded program in Figure 1-22 appears with default foundation properties. The data of foundation properties for the current example, which are required to be defined, are raft material, raft thickness and foundation depth. Any other data corresponding to foundation properties in the program menus are not required in this example. Therefore, the user can take these data from the default foundation properties.

To enter the raft material and thickness, choose "Element groups" command from "In Table" menu. The following list box in Figure 1-23 appears. In this list box, enter E-Modulus of the raft, *Poisson's* ratio of the raft and raft thickness. Then click "OK" button to go to the next step.

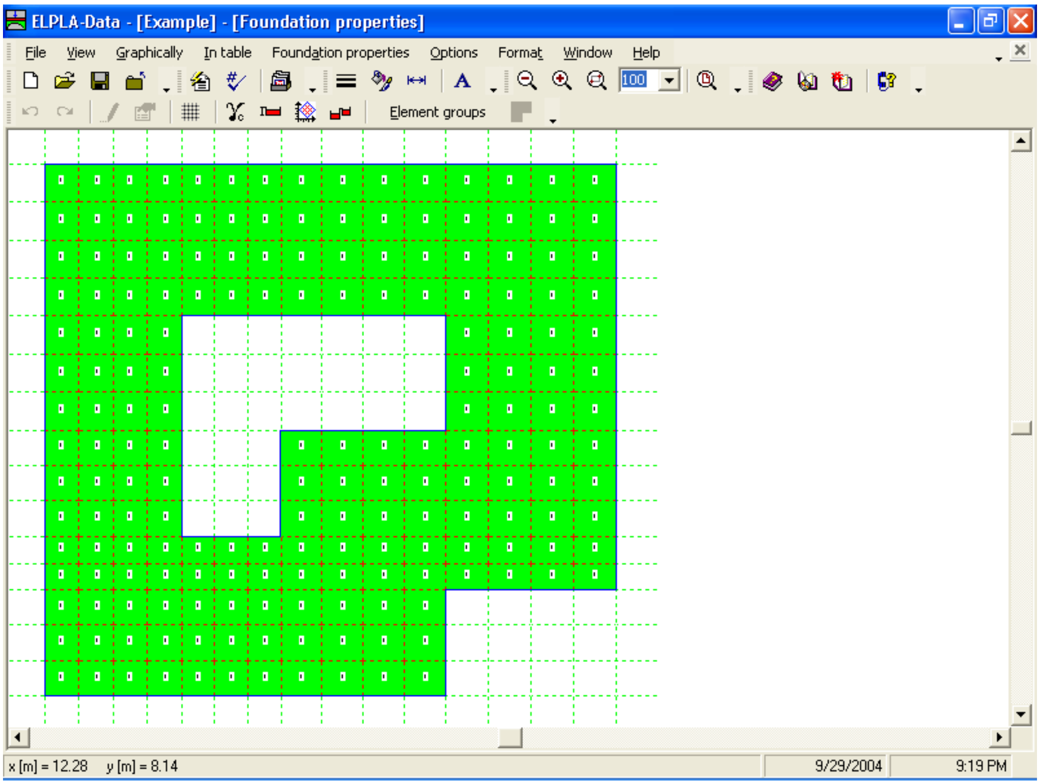


Bild 1-22 "Foundation properties" embedded program

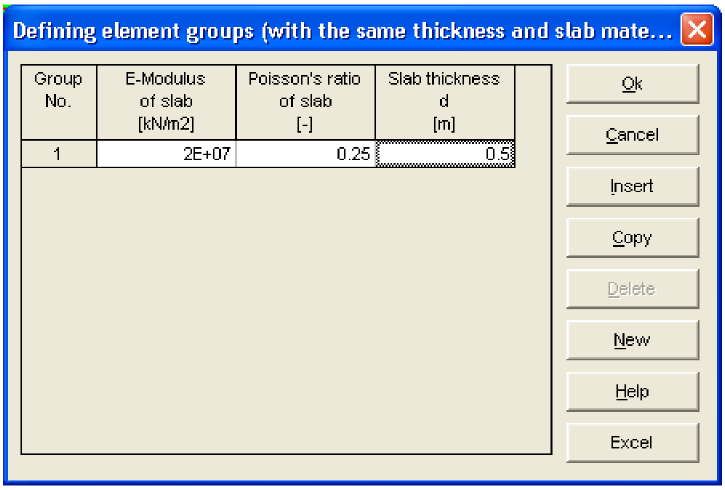


Figure 1-23 "Defining element groups" list box

To enter the unit weight of the raft, choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 1-22. The following dialog box in Figure 1-24 with a default unit weight of 25 [kN/m^3] appears. To neglect the self-weight of the raft in the analysis, type 0 in the "Unit weight of the foundation" edit box. Click "OK" button.

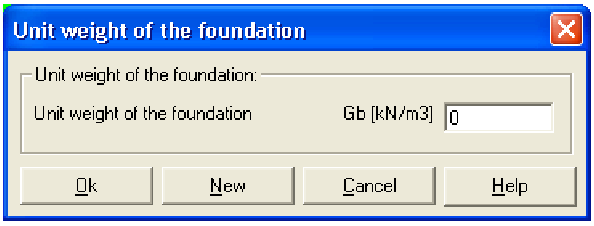


Figure 1-24 "Unit weight of the foundation" dialog box

To enter the foundation depth under the ground surface, choose "Foundation depth" command from "Foundation properties" menu in the window of Figure 1-22. The following dialog box in Figure 1-25 appears to define the foundation depth under the ground surface. Type 2.7 in the "Foundation depth under the ground surface" edit box. Then click "OK" button.

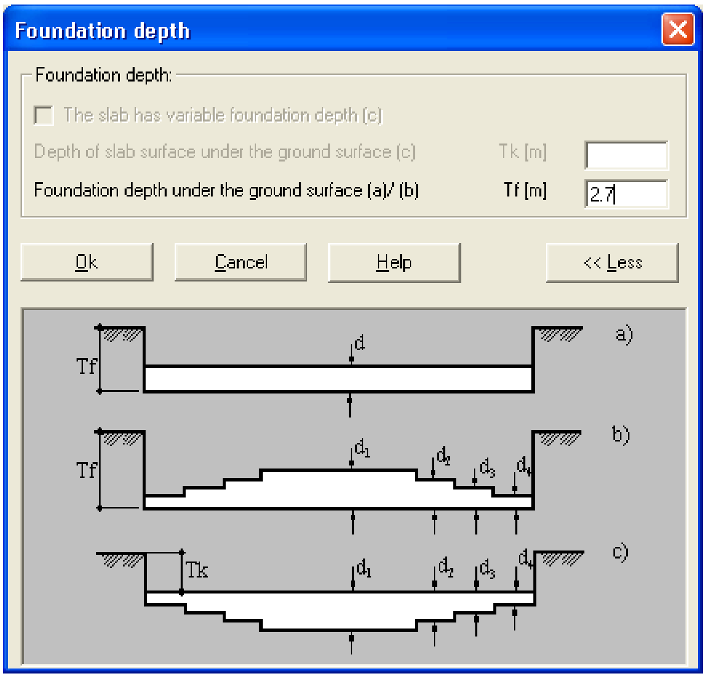


Figure 1-25 "Foundation depth" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 1-22 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 1-22 to close the "Foundation properties" embedded program and return to the main window of *ELPLA-Data*

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

2.6 Boring fields

If the subsoil under the raft is defined by two boring logs or more such as in the current example, the irregularity of the subsoil must be taken into account. "Boring fields" command let the user define which method is to be used to consider the irregularity of the subsoil. In the current example, the Interpolation Method will be used.

To consider the Interpolation Method in the analysis, choose "Boring fields" command from "Data" menu. The following embedded program in Figure 1-26 appears with a default method. *ELPLA* considers that the Interpolation Method is the default one, which takes into account the irregularity of the subsoil. In most cases *ELPLA* defines the interpolation zone types I, II, III automatically such as in this example. But in the case of extreme boring arrangements, the user must define these zones.

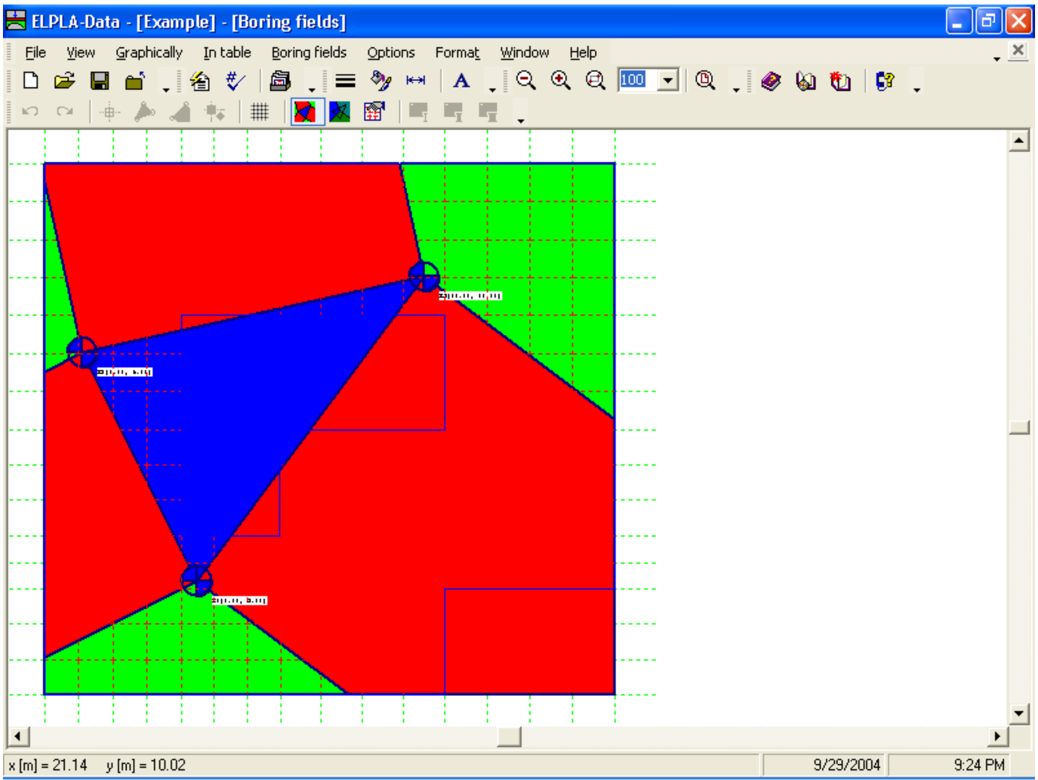


Figure 1-26 "Boring fields" embedded program

You do not need to change anything. Now do the following two steps:

- Choose "Save boring fields" command from "File" menu in Figure 1-26 to save the data of boring fields
- Choose "Close boring fields" command from "File" menu in Figure 1-26 to close the "Boring fields" embedded program and return to the main window of *ELPLA-Data*

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

2.7 Loads

In *ELPLA*, loads on the raft such as point loads, line loads, uniform loads or moments may be applied to the net of the finite elements at any position independently on the node position. The coordinates of the loads are related to the lower-left corner of the raft (local coordinates).

To define the loads, choose "Loads" command from "Data" menu. The following embedded program in Figure 1-27 appears.

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Graphically" menu in Figure 1-27. In this example the user will learn the definition of loads numerically.

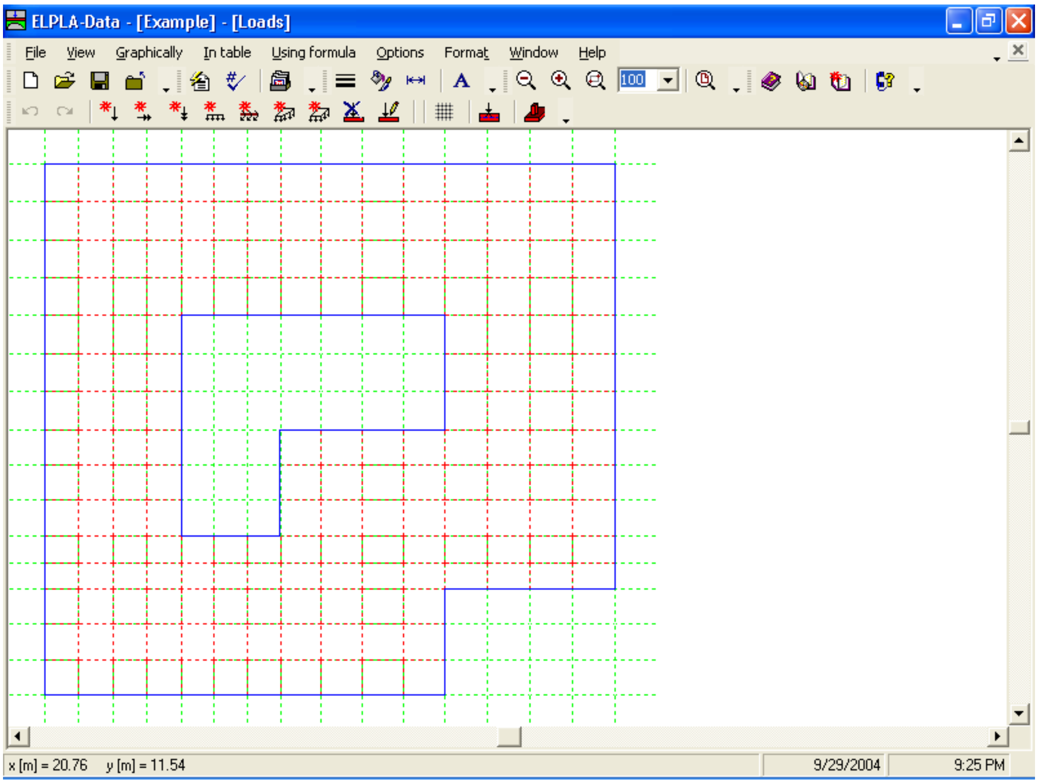
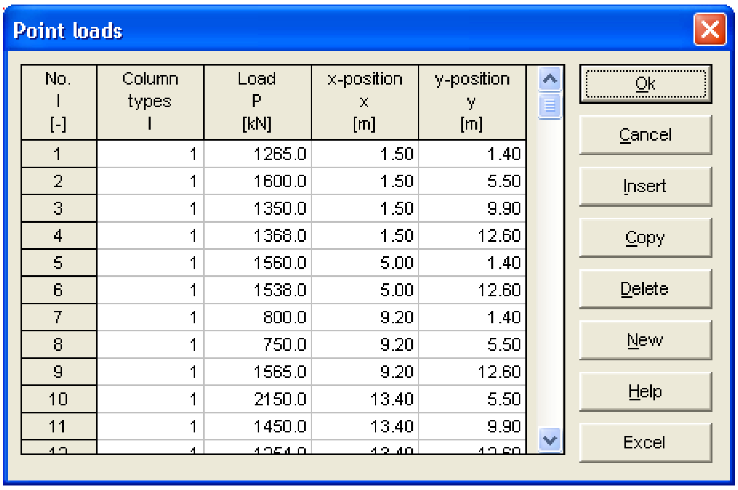


Figure 1-27 "Loads" embedded program

To enter point loads

- Choose "Point loads" command from "In Table" menu in the window of Figure 1-27. The following list box in Figure 1-28 appears. *ELPLA* can distribute concentrated loads under columns. In this example, data corresponding to column dimensions are not required. Therefore, the user can take these data from the default column dimensions and consider all loads have column type 1
- Enter the external point loads *P* [kN] and their corresponding coordinates (*x*, *y*) in the list box of Figure 1-28 by typing the value in the corresponding cell and pressing "Enter" key
- The coordinates of the point load are related to the lower-left corner of the raft (local coordinates)
- Click "OK" button

Repeat the previous steps for moments *Mx*, moments *My*, line loads and distributed loads using "Moments *Mx*", "Moments *My*", "Line loads" and "Distributed loads" commands from "In Table" menu respectively. After that the screen should look like the following Figure 1-29.



No. I [-]	Column types I	Load P [kN]	x-position x [m]	y-position y [m]
1	1	1265.0	1.50	1.40
2	1	1600.0	1.50	5.50
3	1	1350.0	1.50	9.90
4	1	1368.0	1.50	12.60
5	1	1560.0	5.00	1.40
6	1	1538.0	5.00	12.60
7	1	800.0	9.20	1.40
8	1	750.0	9.20	5.50
9	1	1565.0	9.20	12.60
10	1	2150.0	13.40	5.50
11	1	1450.0	13.40	9.90

Figure 1-28 "Point loads *P*" List box

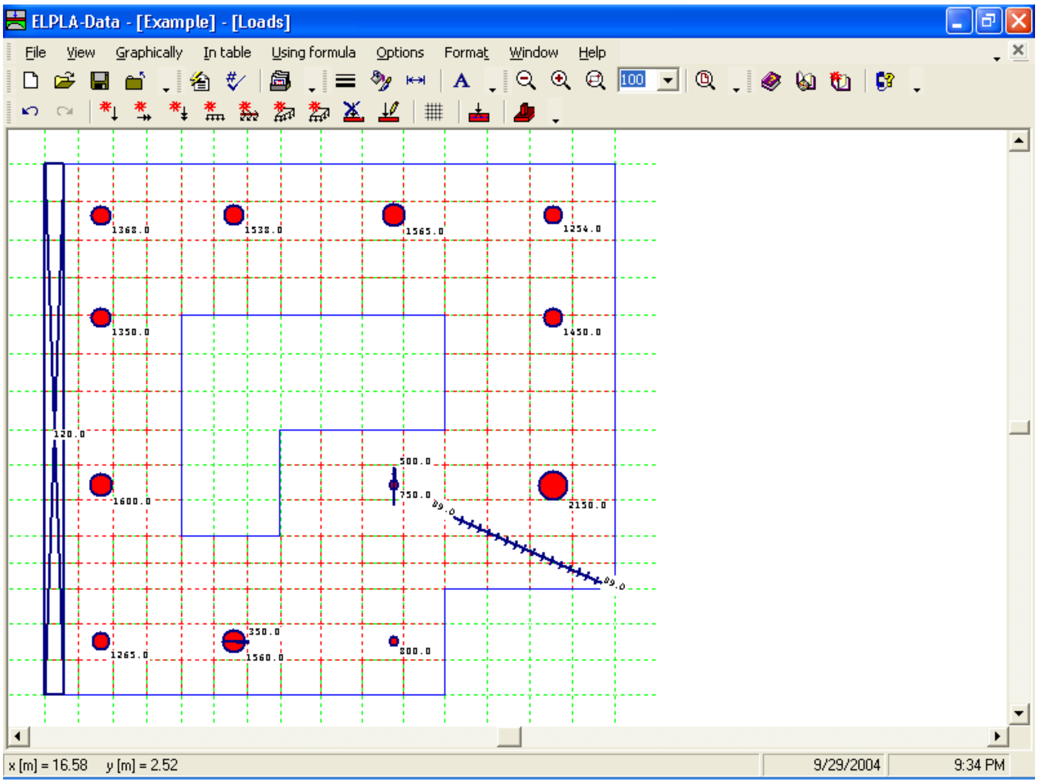


Figure 1-29 Loads on the screen

To save the load data, choose "Save loads" command from "File" menu. Then choose "Close loads" command from this menu to close "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 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 which you have just defined, leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar at the upper-right corner of *ELPLA-Data*. *ELPLA-Solver* window appears, Figure 1-30. This window belongs to *ELPLA-Solver*. Like *ELPLA-Data*, on the upper-right corner of *ELPLA-Solver* window appears the menu bar of the sub programs, which are used for switching between individual sub programs of *ELPLA* package. On the upper-left corner of this window appears the menu bar of *ELPLA-Solver*, which is used for analyzing the problem.

In Figure 1-30, the "Calculation" menu is active. This menu contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For the current example, the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Iteration process
- Determining deformation, internal forces, contact pressures

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

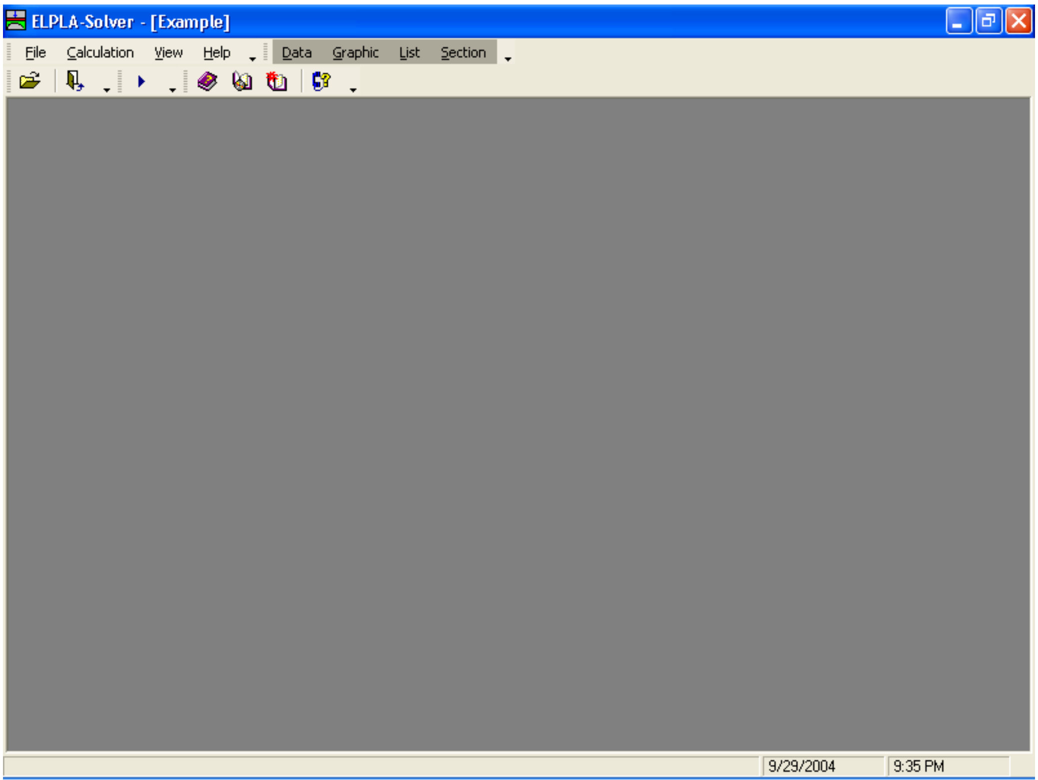


Figure 1-30 Opening screen of *ELPLA-Solver*

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 following "Iteration parameters" option box in Figure 1-31 appears
- In "Iteration parameters" option box, select which option is ending the iteration process. For this example, choose an accuracy of 0.0001 [m] to end the iteration process
- Click "OK" button

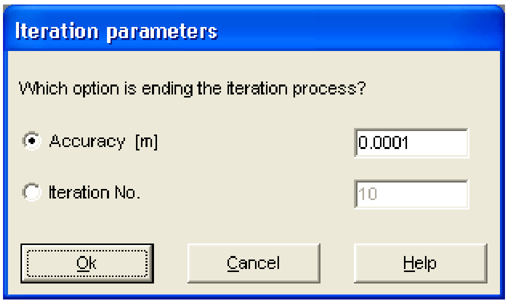


Figure 1-31 "Iteration parameters" option box

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 1-32 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.

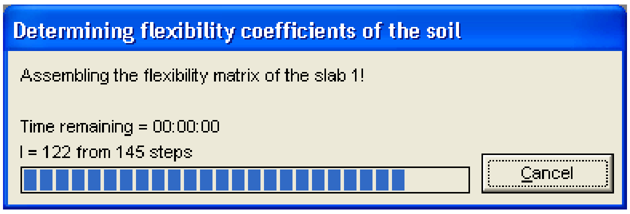


Figure 1-32 Analysis progress menu

Iteration process

Information about the convergence progress of the computations is displayed in the "Iteration process" list box in Figure 1-33 during the iteration process.

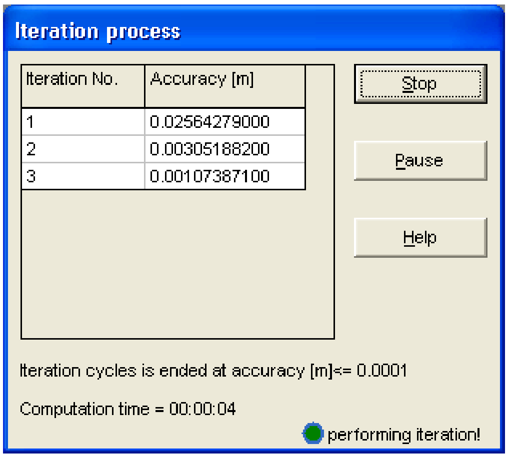


Figure 1-33 "Iteration process" list box

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 1-34. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

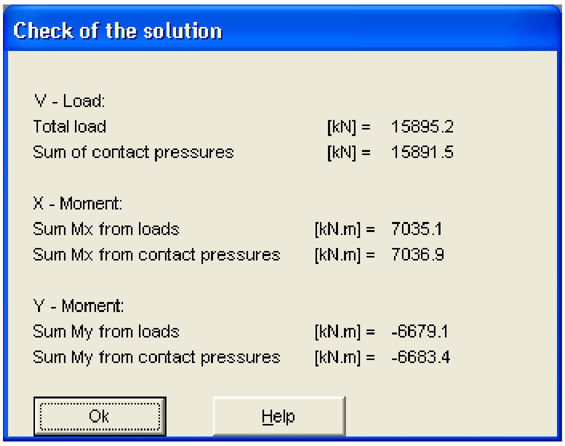


Figure 1-34 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

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 data and result graphics

To view graphically the data and results of a problem that has already been defined and analyzed, switch 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 1-35. This window belongs to the sub program *ELPLA-Graphic*. Like in other sub programs of *ELPLA*, on the upper-right corner of *ELPLA-Graphic* window appears the menu bar, which is used for switching between individual sub programs of the *ELPLA* package. On the upper-left corner of this window appears the menu bar of *ELPLA-Graphic*, which is used for displaying the data and results.

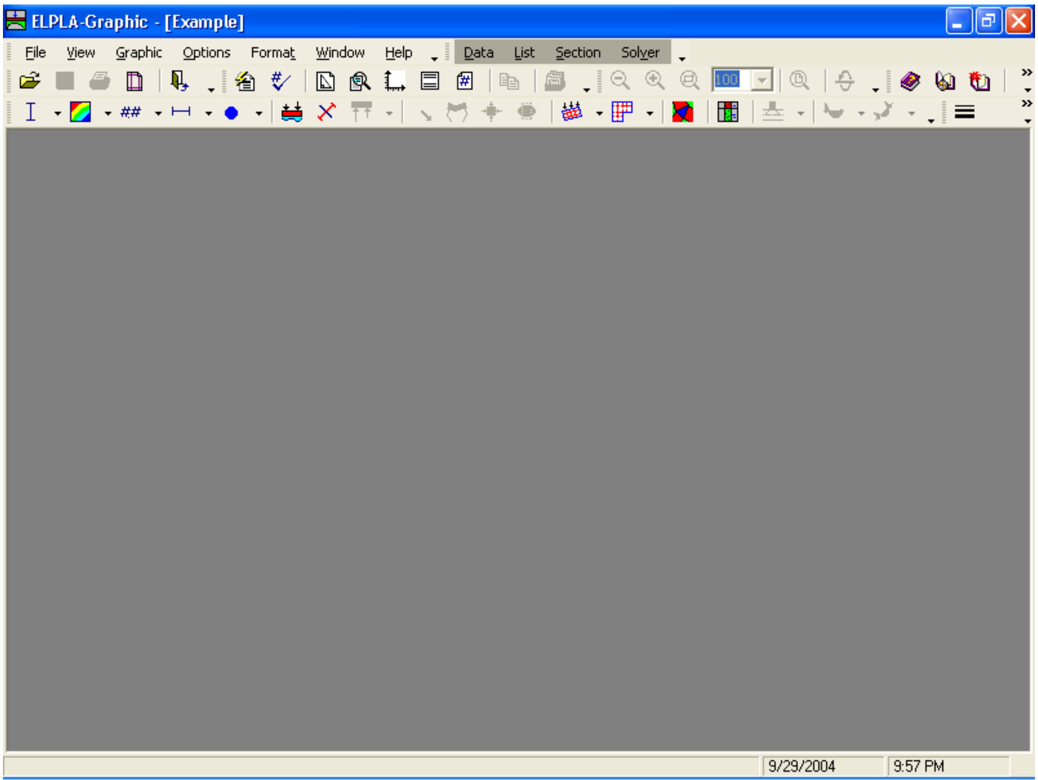


Figure 1-35 Opening screen of the sub program *ELPLA-Graphic*

The "Graphic" menu 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:

- Results in isometric view
- Results as contour lines
- Result values in the plan
- Distribution of results in the plan
- Results as circular diagrams
- Slab deformation
- Principal moments as streaks
- Data in isometric view
- Data in the plan
- Boring locations
- Boring logs

Only the first command of the "Graphic" 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 view the results in isometric view

- Choose "Results in isometric view" command from "Graphic" menu. The following option box in Figure 1-36 appears
- In the "Results in isometric view" option box, select "Contact pressures *q*" as an example for the results to be displayed
- Click "OK" button

The contact pressures are now displayed in an isometric view as shown in Figure 1-37.

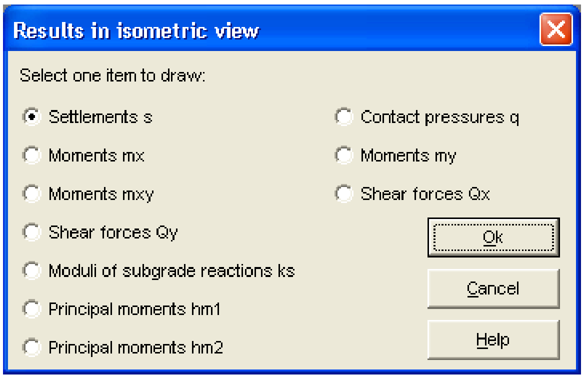


Figure 1-36 "Results in isometric view" option box

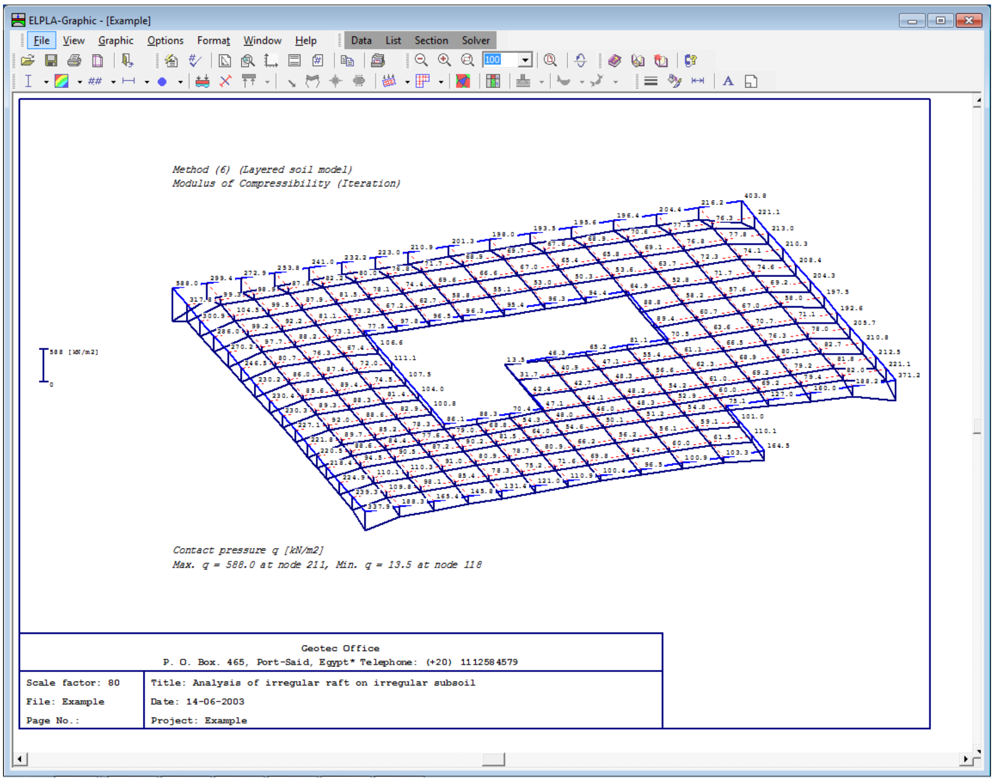


Figure 1-37 Contact pressures in an isometric view

4.2 Plot a diagram of the results at a specified section

To plot a diagram of the results, switch 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* window appears, Figure 1-38. The function of *ELPLA-Section* is plotting and printing the results in diagrams. *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.

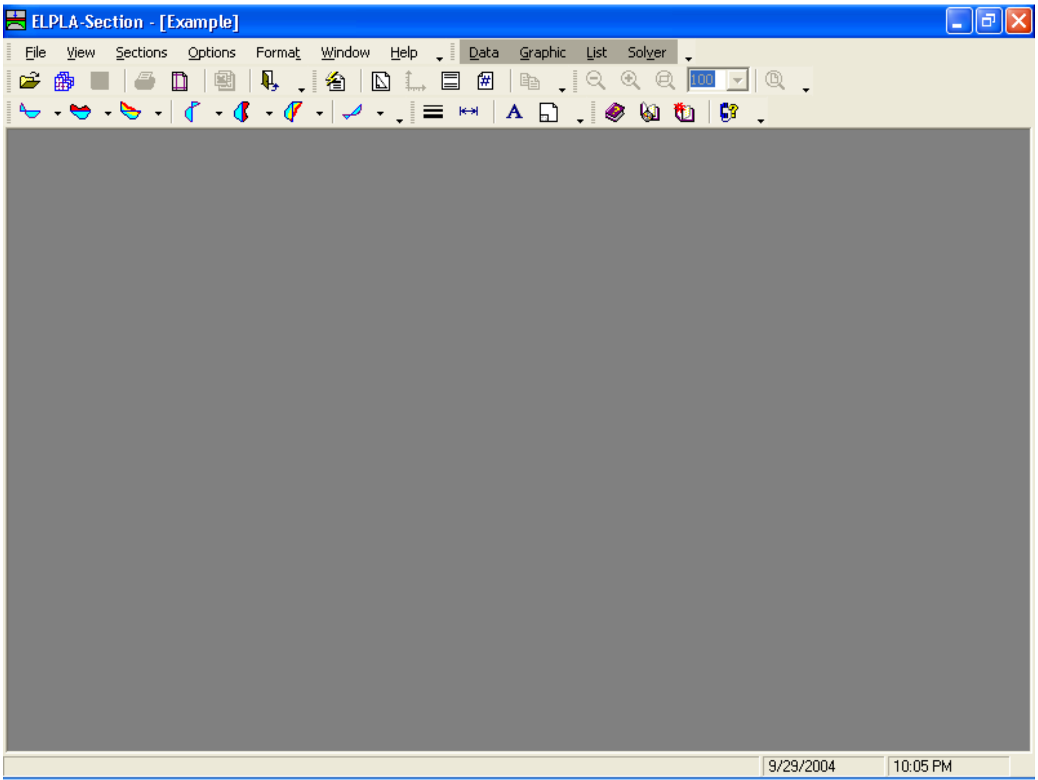


Figure 1-38 Opening screen of the sub program *ELPLA-Section*

The "Section" menu in *ELPLA-Section* contains the commands of drawing the diagrams. The commands for presenting the results in diagrams are:

- Section in *x*-direction
- Max./ Min. values in *x*-direction
- Overlapping in *x*-direction
- Arbitrary section
- Section in *y*-direction
- Max./ Min. values in *y*-direction
- Overlapping in *y*-direction

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 1-39 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

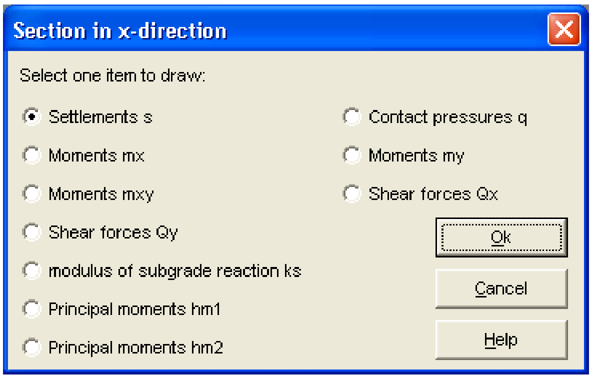


Figure 1-39 "Section in *x*-direction" option box

The following dialog box in Figure 1-40 appears to specify the section in *x*-direction. In this dialog box, click "OK" button to plot the default section. The settlements are now plotted in a diagram as shown in Figure 1-41.

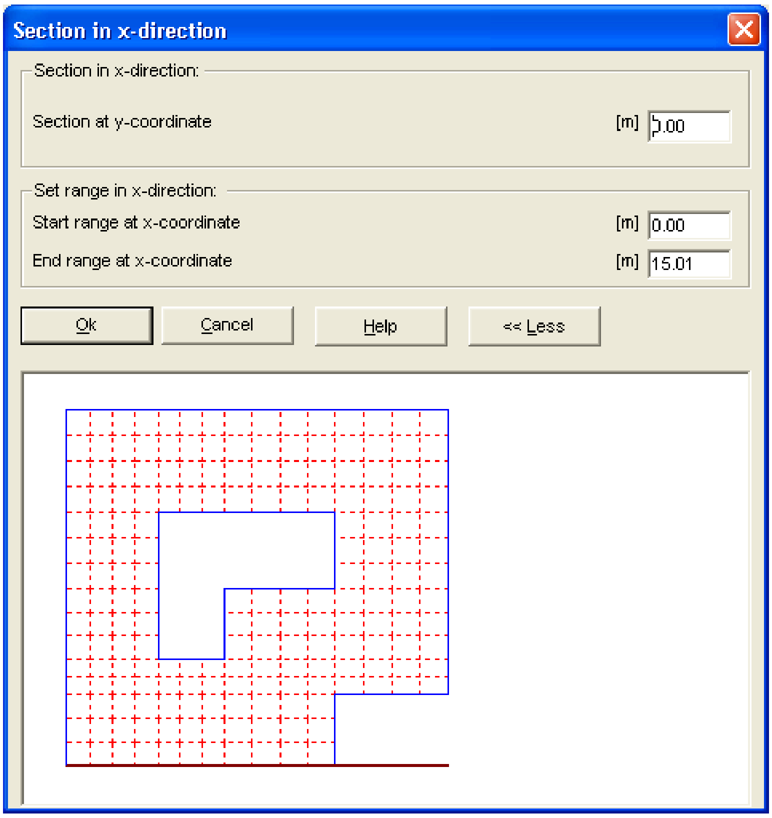


Figure 1-40 "Section in *x*-direction" dialog box with a default section

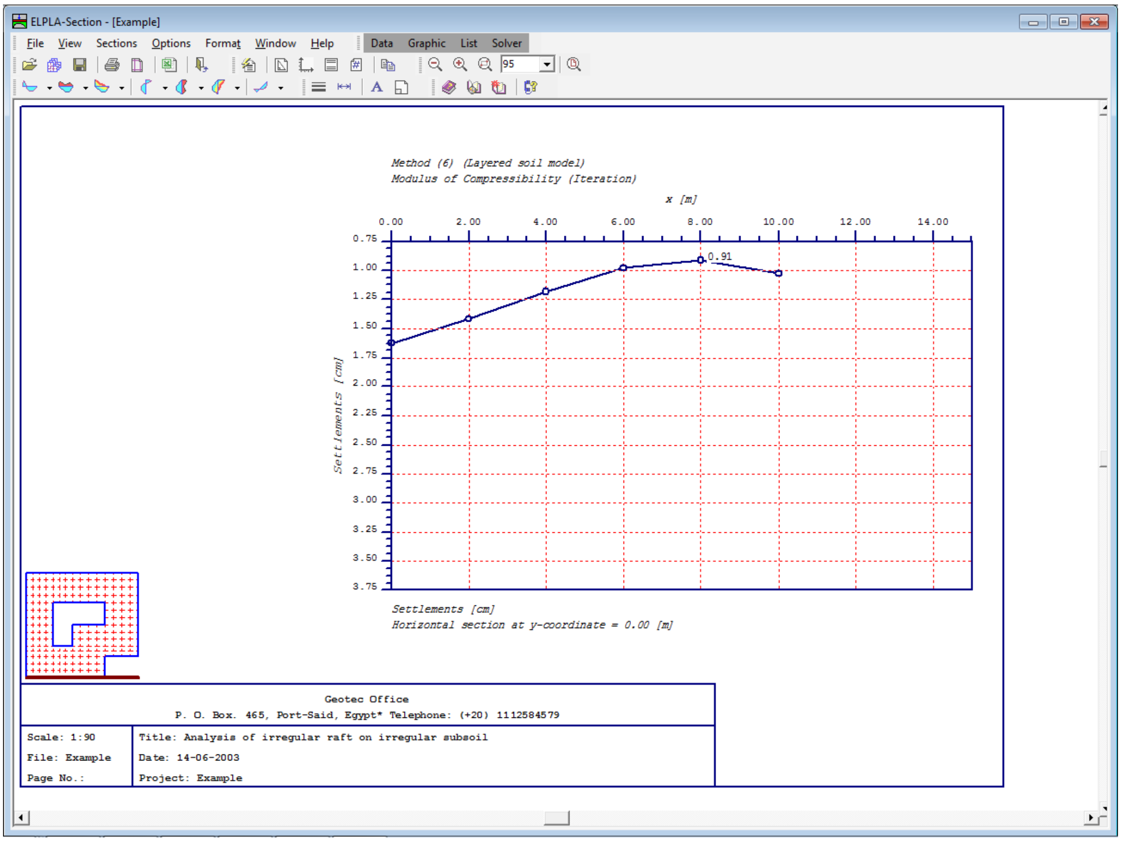


Figure 1-41 Diagram of settlements in *x*-direction

4.3 Print the drawing

To print the drawing

- Choose "Print" command from "File" menu in *ELPLA-Graphic* or *ELPLA-Section* Window. The following "Print" dialog box in Figure 1-42 appears
- In the "Print" dialog box, define the printer properties and drawing copies
- Click "OK" button

Only the objects currently displayed on the drawing are printed.

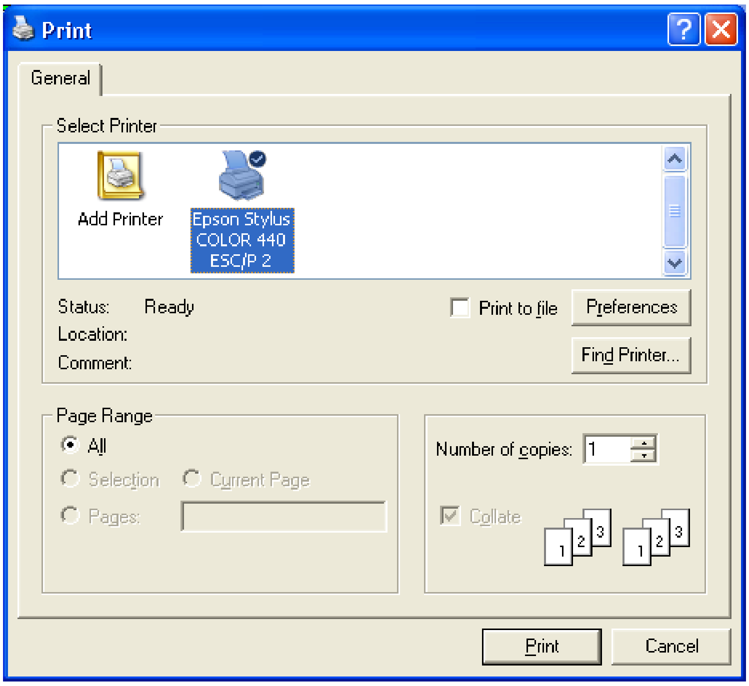


Figure 1-42 "Print" dialog box

4.4 Listing data and results 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 1-43. The function of *ELPLA-List* is listing and printing data and results in tables. 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.

The "List" menu in *ELPLA-List* contains the commands of listing data and results. The commands for listing data and results in tables are:

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

Only the first 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*.

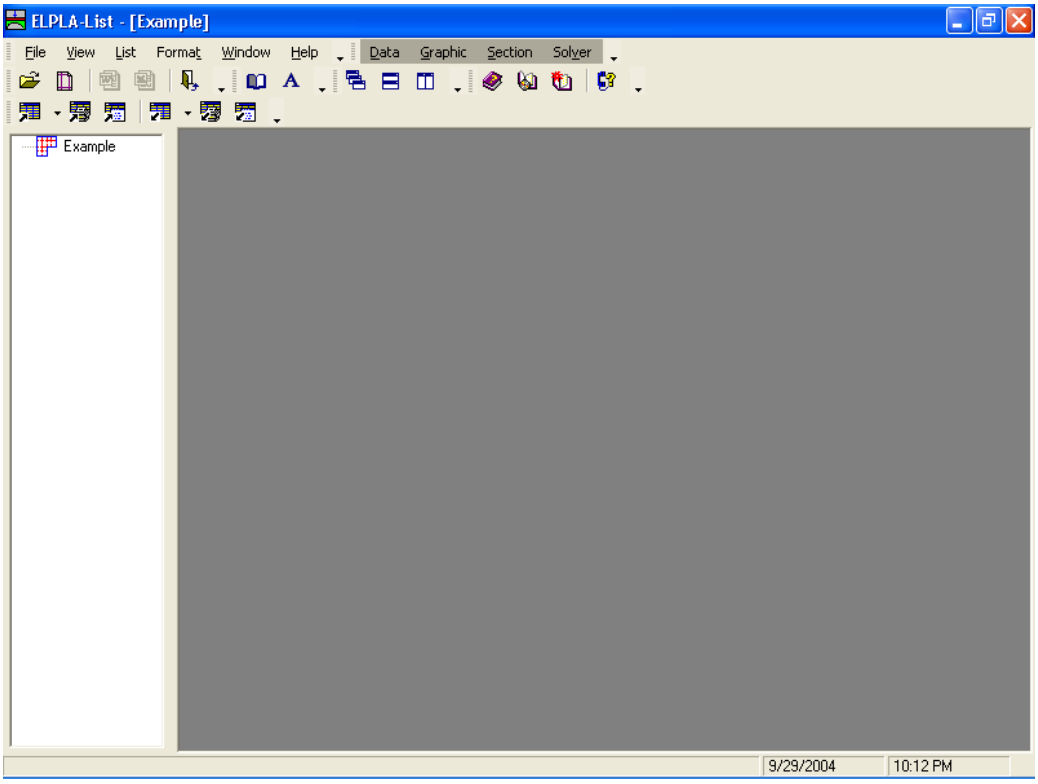


Figure 1-43 Opening screen of the sub program *ELPLA-List*

To list data in a table

- Choose "Display tables of data" command from "List" menu. The following option box in Figure 1-44 appears
- In the "Display tables of data" option box, select "Loading" as an example for the data to be listed in a table
- Click "OK" button. The loading data are now listed on the screen (Figure 1-45)
- Choose "Send to Word" from "File" menu if you wish to export the table to a MS Word application, Figure 1-46

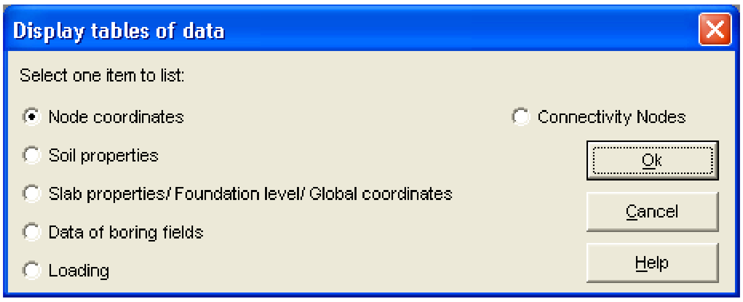
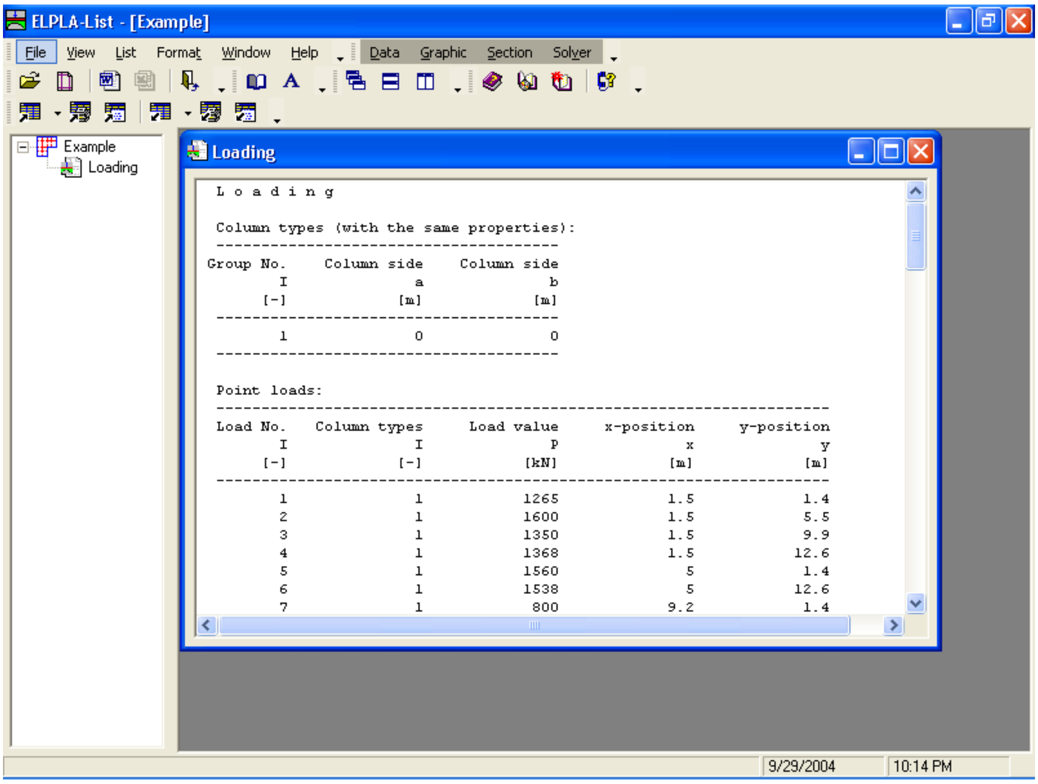


Figure 1-44 "Display tables of data" option box



L o a d i n g

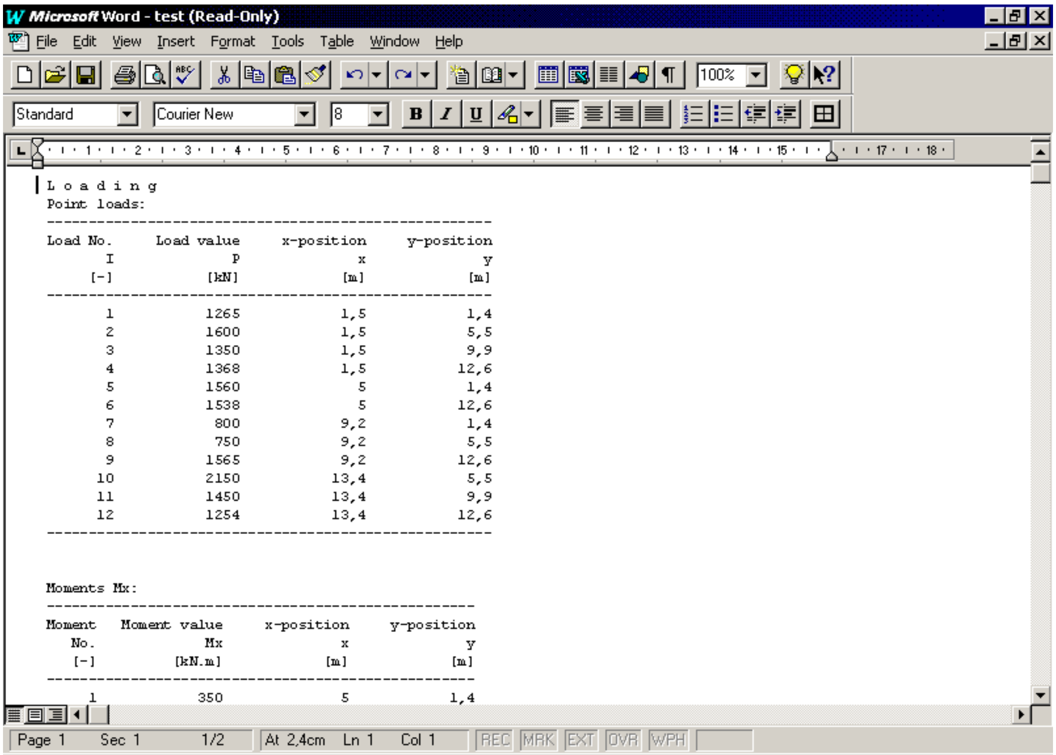
Column types (with the same properties):

Group No. I [-]	Column side a [m]	Column side b [m]
1	0	0

Point loads:

Load No. I [-]	Column types I [-]	Load value P [kN]	x-position x [m]	y-position y [m]
1	1	1265	1.5	1.4
2	1	1600	1.5	5.5
3	1	1350	1.5	9.9
4	1	1368	1.5	12.6
5	1	1560	5	1.4
6	1	1538	5	12.6
7	1	800	9.2	1.4

Figure 1-45 List of loading data



L o a d i n g

Point loads:

Load No. I [-]	Load value P [kN]	x-position x [m]	y-position y [m]
1	1265	1,5	1,4
2	1600	1,5	5,5
3	1350	1,5	9,9
4	1368	1,5	12,6
5	1560	5	1,4
6	1538	5	12,6
7	800	9,2	1,4
8	750	9,2	5,5
9	1565	9,2	12,6
10	2150	13,4	5,5
11	1450	13,4	9,9
12	1254	13,4	12,6

Moments Mx:

Moment No. [-]	Moment value Mx [kN.m]	x-position x [m]	y-position y [m]
1	350	5	1,4

Figure 1-46 Exported data in MS Word

5 Index