Example 3

Analysis of system of two circular rafts

Contents   Page

1   Description of the problem

An example of a system of two equal large circular rafts is selected to illustrate some features of *ELPLA* for analyzing system of foundations.

1.1   Loads and dimensions

Each raft has a diameter of 22 [m] and thickness of 0.65 [m]. Loading on each raft consists of 24 column loads, in which 16 column loads have P_1 = 1250 [kN] and 8 column loads have P_2 = 1000 [kN] as shown in Figure 3-1 and Table 3-1. The origin coordinates for raft 1 in the global system are (0.0, 0.0) while for raft 2 they are (0.0, 22.5).
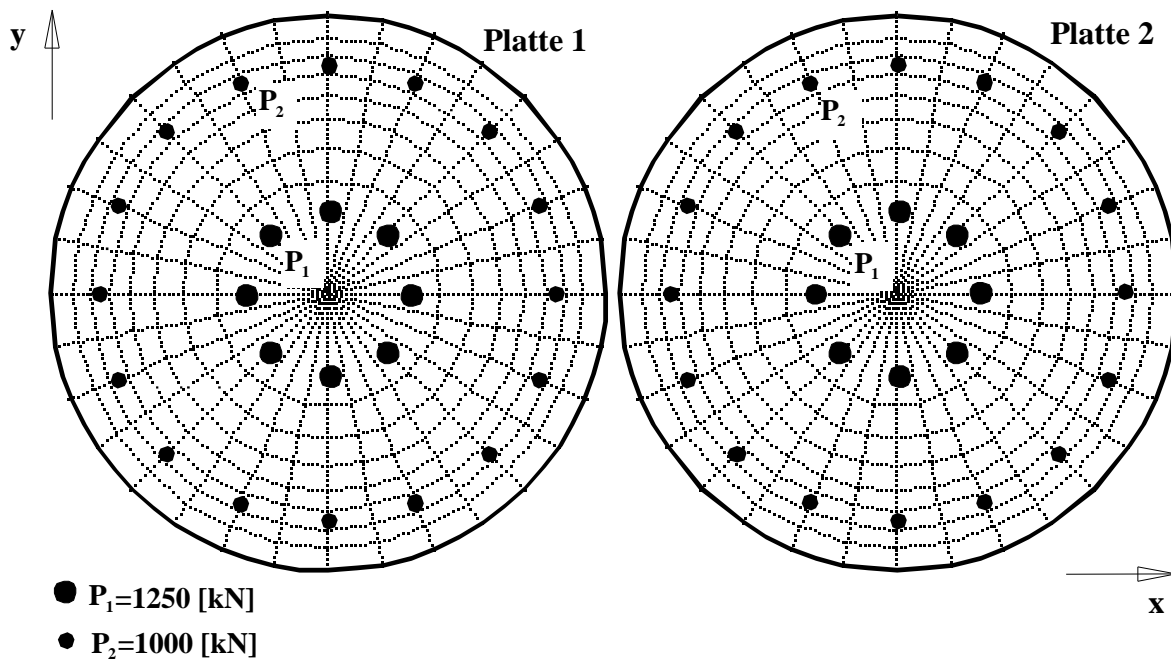

● P_1=1250 [kN]
● P_2=1000 [kN]

Figure 3-1   System of two equal circular rafts

1.2   Raft material

Material of the two rafts is supposed to have the following parameters:

Young's modulus	E_b	$= 2.6 * 10^7$	[kN/m^2]
Poisson's ratio	ν_b	$= 0.15$	[-]
Unit weight of raft material	γ_b	$= 0.0$	[kN/m^3]

Unit weight of the raft material is chosen to be γ_b = 0.0 to neglect the own weight of the raft in the analysis.

1.3 Soil properties

The rafts rest on a silt layer of 15 [m] thickness. The Modulus of compressibility of the silt is $E_s = 9\,500$ [kN/m^2]. *Poisson's* ratio of the soil is taken to be $\nu_s = 0.0$ [-]. The level of foundation under the ground surface is assumed to be $d_f = 0.0$ [m]. The effect of the reloading pressure on the soil and the uplift pressure on the rafts are neglected.

1.4 Method of analysis

It is required to analyze the system of rafts together according to the following soil model and numerical calculation method:

- 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)

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*.

Table 3-1   Point loads *P*

Load No. I [-]	Load value P [kN]	x-position x [m]	y-position y [m]
1	1250	7.75	11
2	1250	14.25	11
3	1250	11	7.75
4	1250	11	14.25
5	1250	8.7	8.7
6	1250	13.3	8.7
7	1250	8.7	13.3
8	1250	13.3	13.3
9	1000	2	11
10	1000	20	11
11	1000	11	2
12	1000	11	20
13	1000	4.64	4.64
14	1000	17.36	4.64
15	1000	4.64	17.36
16	1000	17.36	17.36
17	1000	2.69	7.56
18	1000	7.56	2.69
19	1000	14.44	2.69
20	1000	19.31	7.56
21	1000	2.69	14.44
22	1000	7.56	19.31
23	1000	14.44	19.31
24	1000	19.31	14.44

2   Creating the project of raft 1

In this section the user will learn how to create a project for analyzing system of two rafts. Thus is done by first entering the data of the two rafts individually in the same manner of the previous foundation example and then creating a project for the system of two rafts.

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 3-2. This wizard will guide you through the steps required to create the project. As shown in this Figure, 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 a foundation problem, select "Analysis of slab foundation" then click "Next" button to go to the next form.
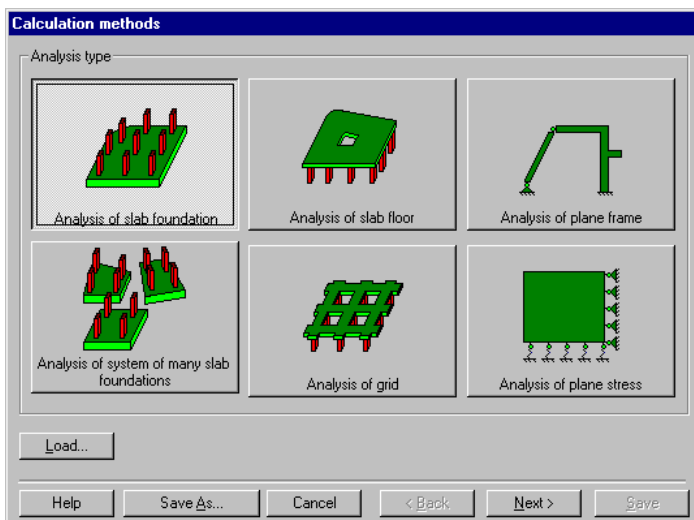

Figure 3-2   "Calculation method" wizard with "Analysis type" form

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

To define the calculation method

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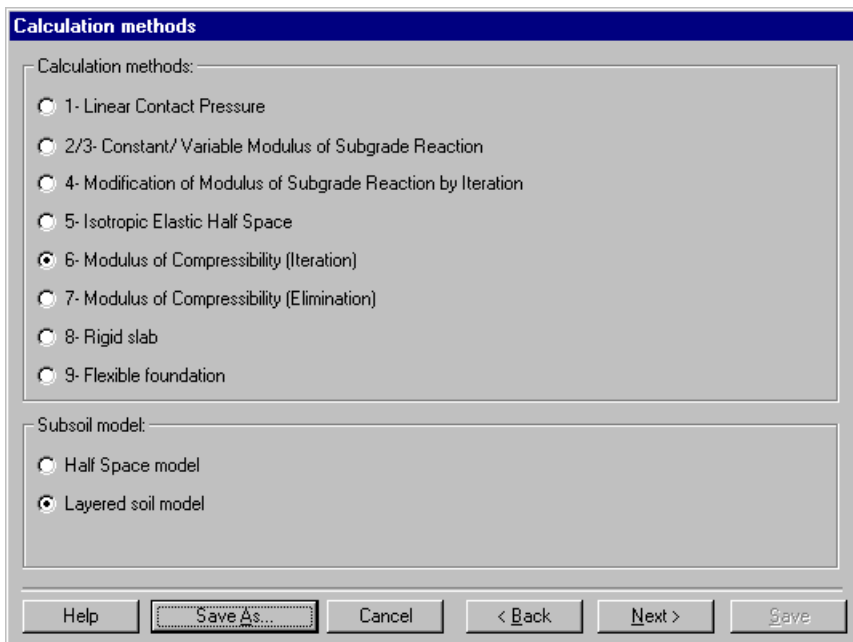- Click "Next" button to go to the next form

Figure 3-3   "Calculation methods" form

The next form is the "System symmetry" (Figure 3-4). In this form

-   Choose "Unsymmetrical system"
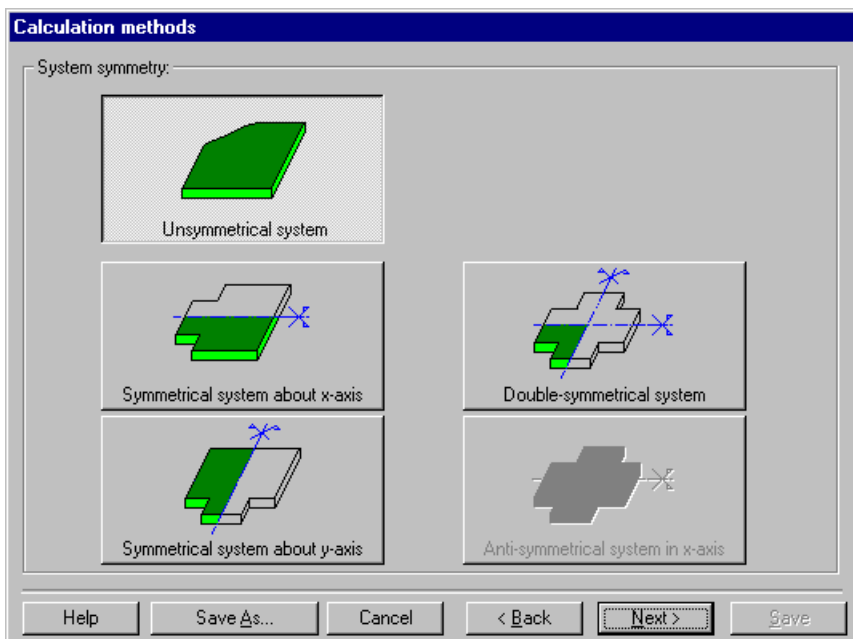-   Click "Next" button

Figure 3-4   "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 3-5. In this list *ELPLA*
displays some of the available options corresponding to the used numerical model, which differ
from model to other. Since no option will be considered in the analysis, click the "Save" button.
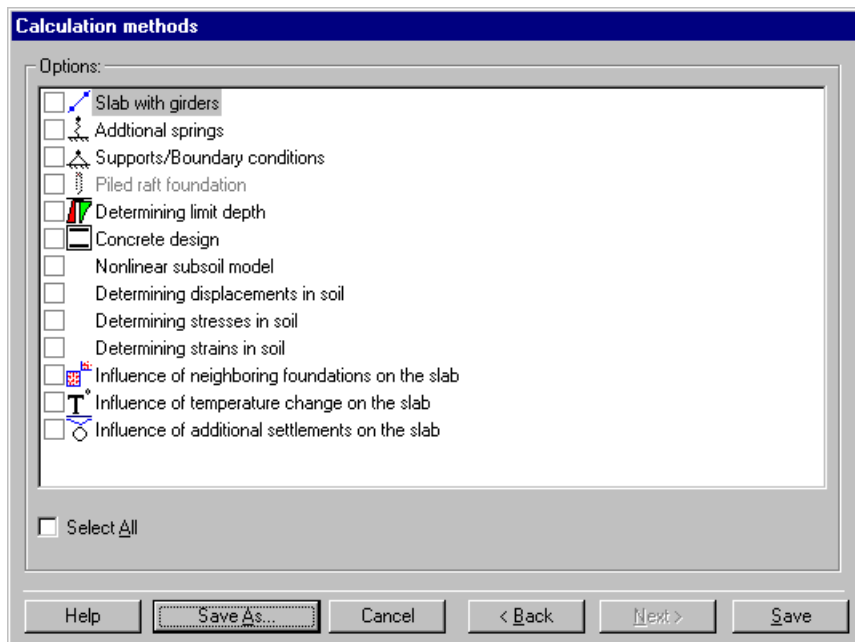
Figure 3-5   "Options" list

After clicking "Save" button the "Save as" dialog box appears, Figure 3-6.

In this dialog box

-    Type a file name for the current project in the "File name" edit box. For example type "Raft 1". *ELPLA* will use automatically this file name in all reading and writing processes
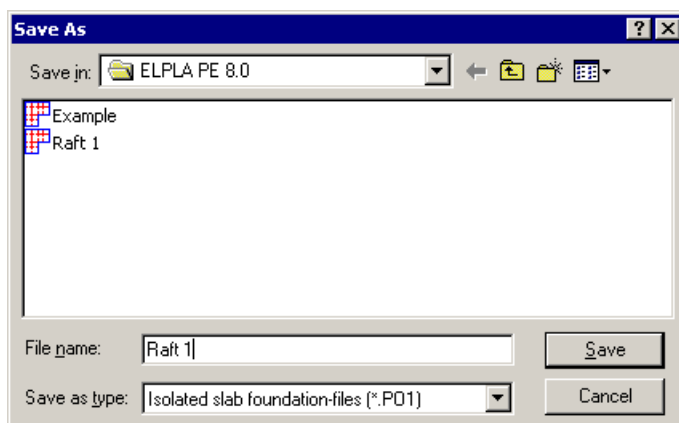-   Click "Save" button

Figure 3-6   "Save as" dialog box

ELPLA will activate the "Data" menu. Also the file name of the current project [Raft1] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 3-7.

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 3-7). *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.
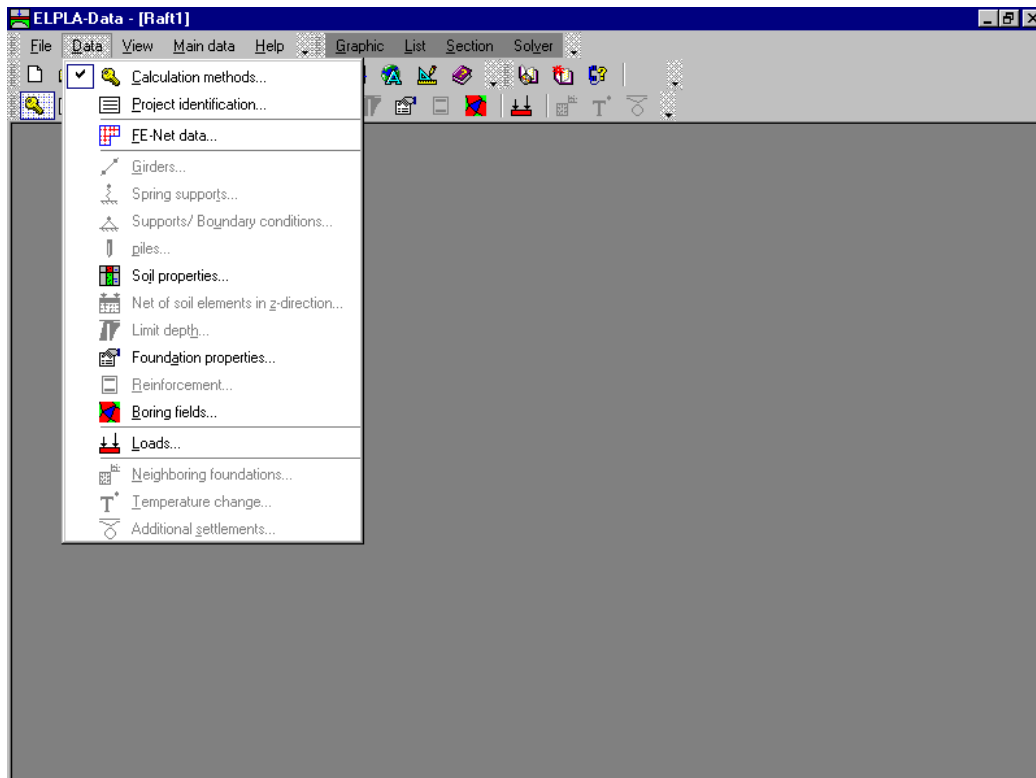
Figure 3-7   *ELPLA-Data* after defining the calculation method

2.2   Project identification

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data* window. The dialog box in Figure 3-8 appears.

In this dialog box

-   Type the following line to describe the problem in the "Title" edit box:
    "Analysis of system of two circular rafts"
-   Type the date of the project in the "Date" edit box
-   Type "Raft 1" in the "Project" edit box
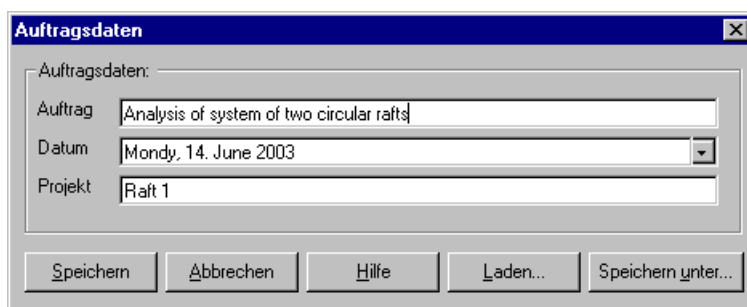-   Click "Save" button

Figure 3-8   "Project identification" dialog box

2.3   FE-Net data

Choose "FE-Net data" command from "Data" menu of *ELPLA-Data* window. The "FE-Net generation" wizard appears (Figure 3-9). This wizard will guide you through the steps required to generate the FE-Net. As shown in this Figure the first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These are used to generate standard nets that have irregular shapes. For the given problem the raft has a circular shape.

To generate the FE-Net

-    In the "Slab type" options choose the circular slab option
-    Type 22 in the "Diameter of circular slab" edit box
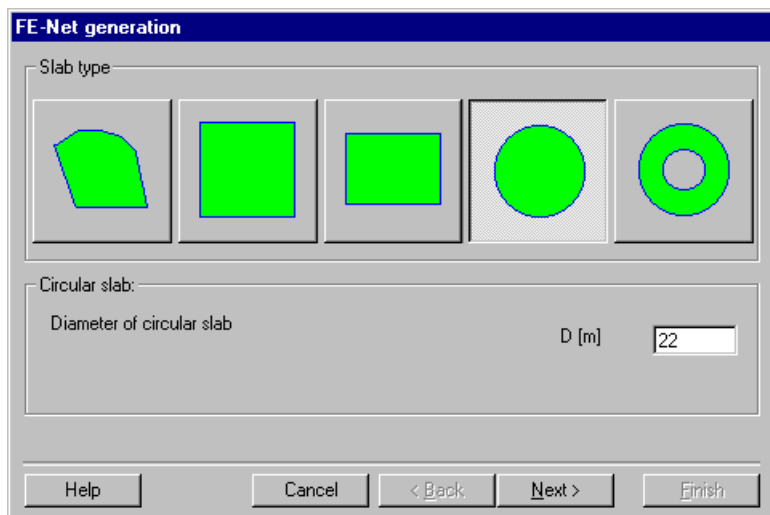-   Click "Next" button to go to the next form

Figure 3-9   "Slab type" form

The following "Generation type" form appears, Figure 3-10. *ELPLA* can deal with various types of generations with triangle and/ or rectangular elements. Choose the combination of triangle and rectangular elements option as the generation type. Click "Next" button to go to the next form.
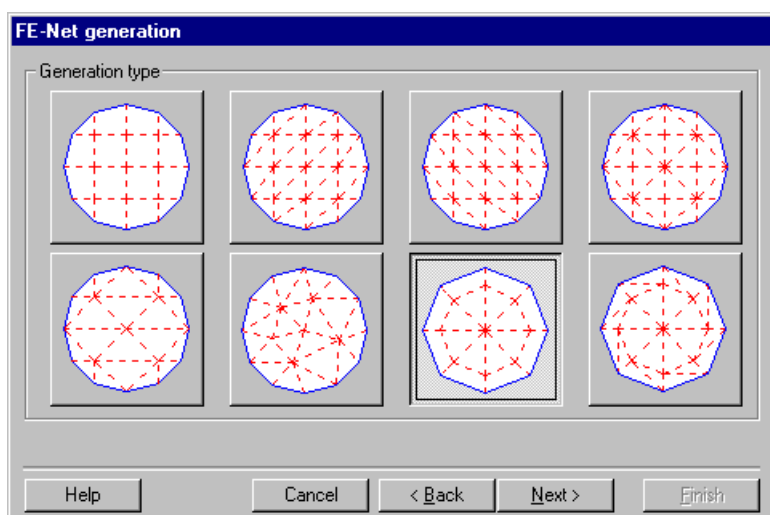

Figure 3-10   "Generation type" page

The next form of the "FE-Net generation" wizard is the "Generation parameters" form. In this form the default values of generation parameters appear, Figure 3-11. For the given problem mesh optimization options are not required.

In this form

- In the *x*-direction frame type 8 in the "No. of circular divisions" edit box
- Uncheck the "Smoothing mesh" option
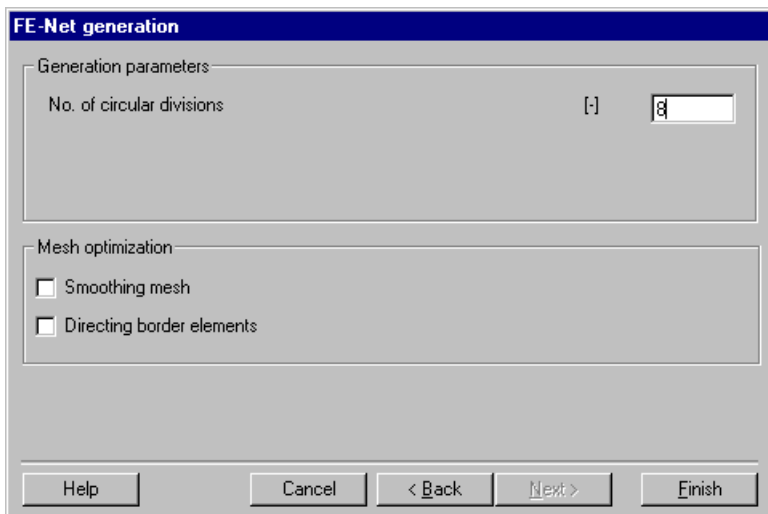- Uncheck the "Directing border elements" option

Figure 3-11   "Generation parameters" form

Click "Finish" button in the "Generation parameters" form. *ELPLA* will generate a suitable FE-Net for the circular raft of 22 [m] diameter with a combination of triangle and rectangular elements, which have equal areas. The following embedded program in Figure 3-12 appears with the generated net.
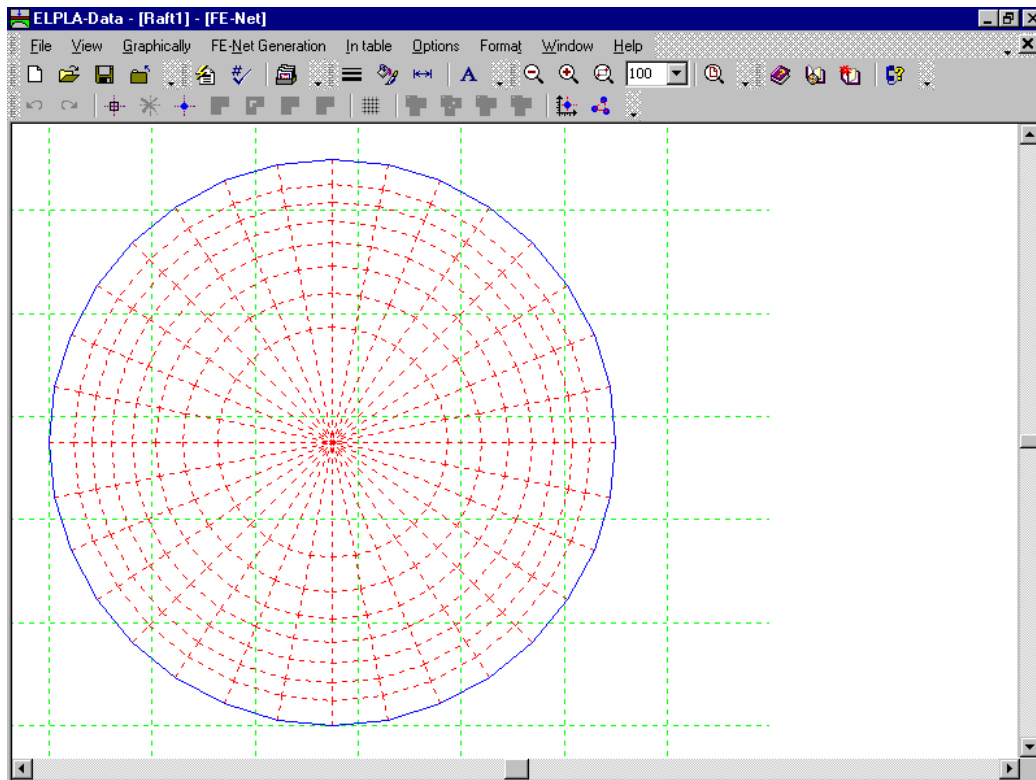
Figure 3-12   FE-Net on the screen

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

- Choose "Save FE-Net" command from "File" menu in Figure 3-12 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 3-12 to close "FE-Net" embedded program and to return to the main window of *ELPLA-Data*

2.4   Soil properties

To define the soil properties, choose "Soil properties" command from "Data" menu of *ELPLA-Data* window. The following sub program in Figure 3-13 appears with a default-boring log.
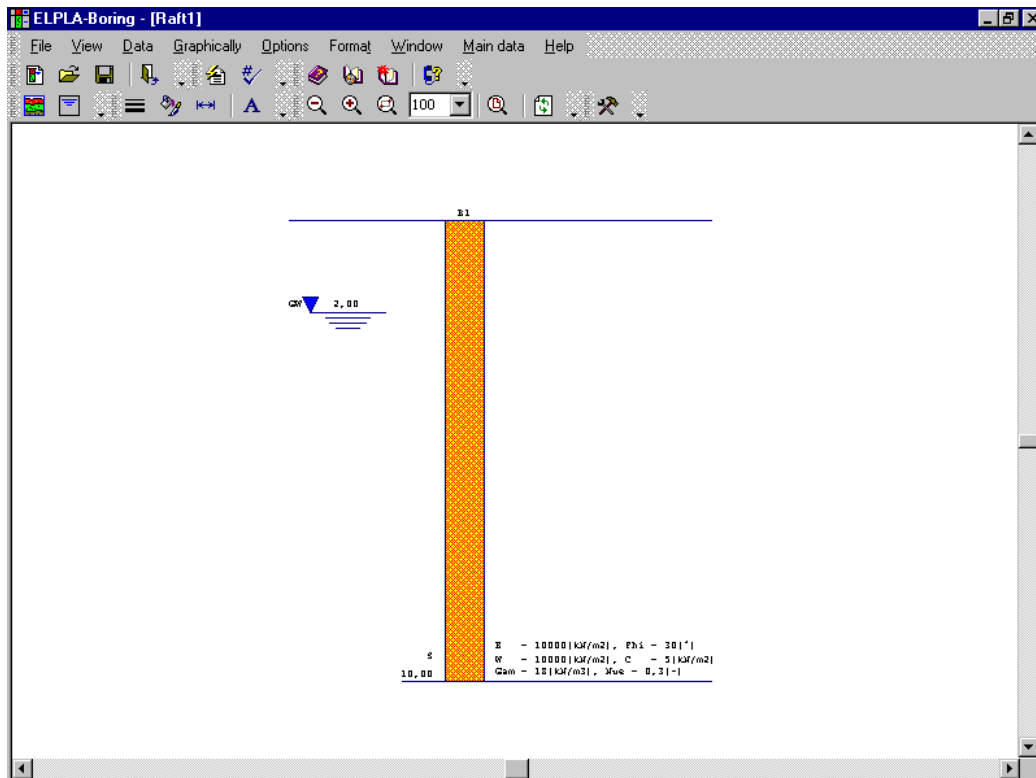
Figure 3-13   *ELPLA-Boring* sub program with a default-boring log

Modifying data of boring log graphically

In *ELPLA* it is also possible to define or modify the boring log 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 to modify the geotechnical data of the soil layer appears (Figure 3-14)
-   In the dialog group box "Geotechnical data of the layer" in Figure 3-14 define the geotechnical data of the soil layer as follows:

E_s = 9500 [kN/m^2]
W_s = 9500 [kN/m^2]

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 $\gamma_s * d_f$. 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 under the foundation depth 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.
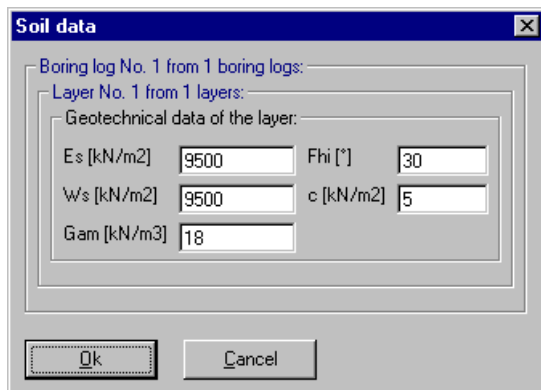
Figure 3-14   "Geotechnical data of the soil layer" dialog box

To define the soil type and color for the layer

- Double-click on the soil symbol of the soil layer. The corresponding dialog box to modify the soil symbols of that layer appears (Figure 3-15)
- 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 3-15. 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
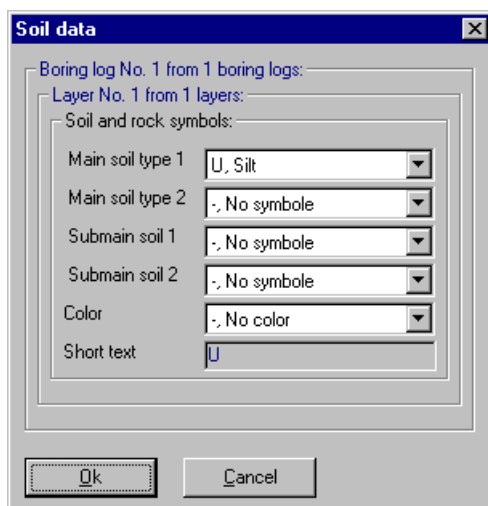
Figure 3-15   "Soil and rock symbols" dialog box

To modify the groundwater depth under the ground surface

- Double-click on the groundwater level. The corresponding edit box to modify the groundwater depth under the ground surface appears, Figure 3-16. To neglect the uplift pressure on the raft, groundwater level is chosen at anywhere under the raft basement
- Type 15 in the "Groundwater depth under the ground surface" edit box
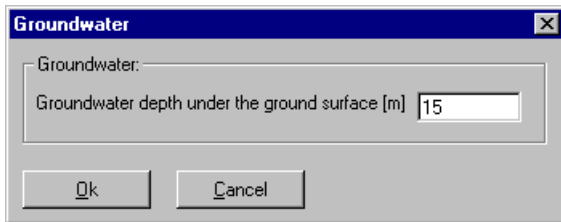- Click "OK" button

Figure 3-16   "Groundwater depth under the ground surface" edit box

To modify a layer depth

- Double-click on the layer depth. The corresponding dialog edit box to modify the layer depth under the ground surface appears, Figure 3-17
- Type 15 in the "Layer depth under the ground surface" edit box
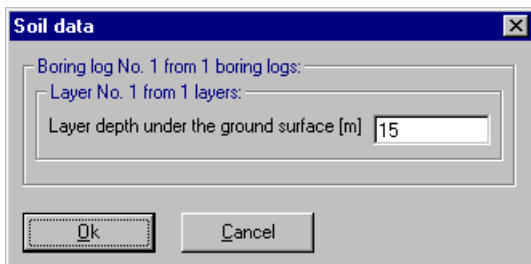- Click "OK" button

Figure 3-17   "Layer 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 to modify the label of the boring log appears, Figure 3-18
- Type B1 in the edit box of Figure 3-18
- Press "Enter" key to consider the text

Figure 3-18   "Label of the boring log" edit box

To enter the main soil data for the layer

- Choose "Main soil data" from "Data" menu in Figure 3-13. The following dialog group box in Figure 3-19 appears
- In this dialog group box enter the settlement reduction factor α [-], *Poisson's* ratio of the soil v_s [-] and the groundwater depth under the ground surface G_w [m] as indicated in Figure 3-19. Note that the groundwater depth under the ground surface was typed in the corresponding edit box because this value has been already defined graphically
- Click "OK" button in the "Main soil data" dialog group box in Figure 3-19
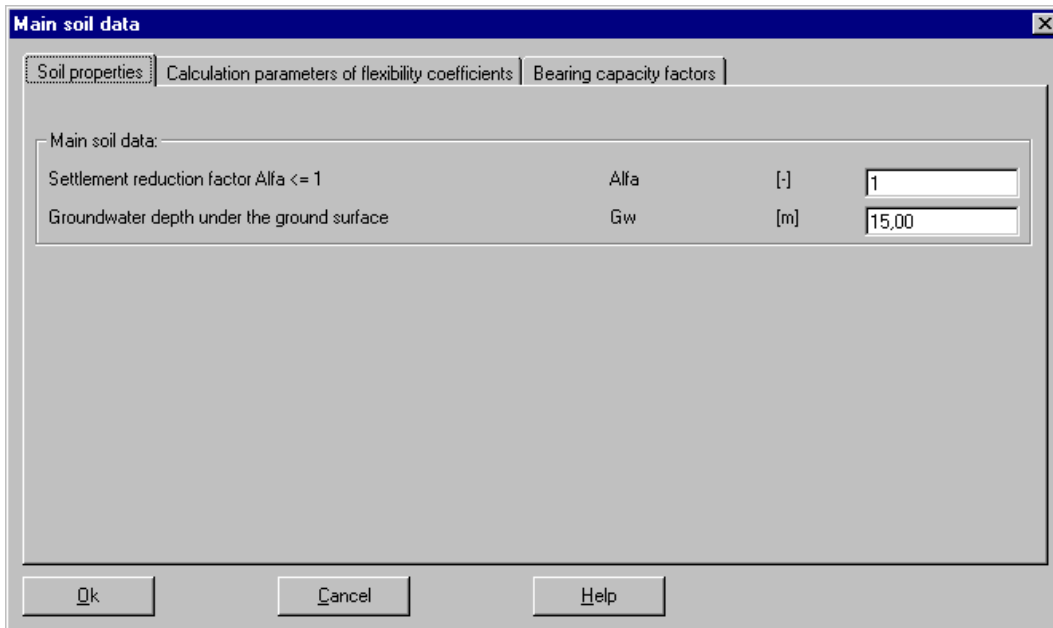
Figure 3-19   "Main soil data" dialog group box

After the user has completed the definition of all soil properties and parameters, the screen should look like the following Figure 3-20.
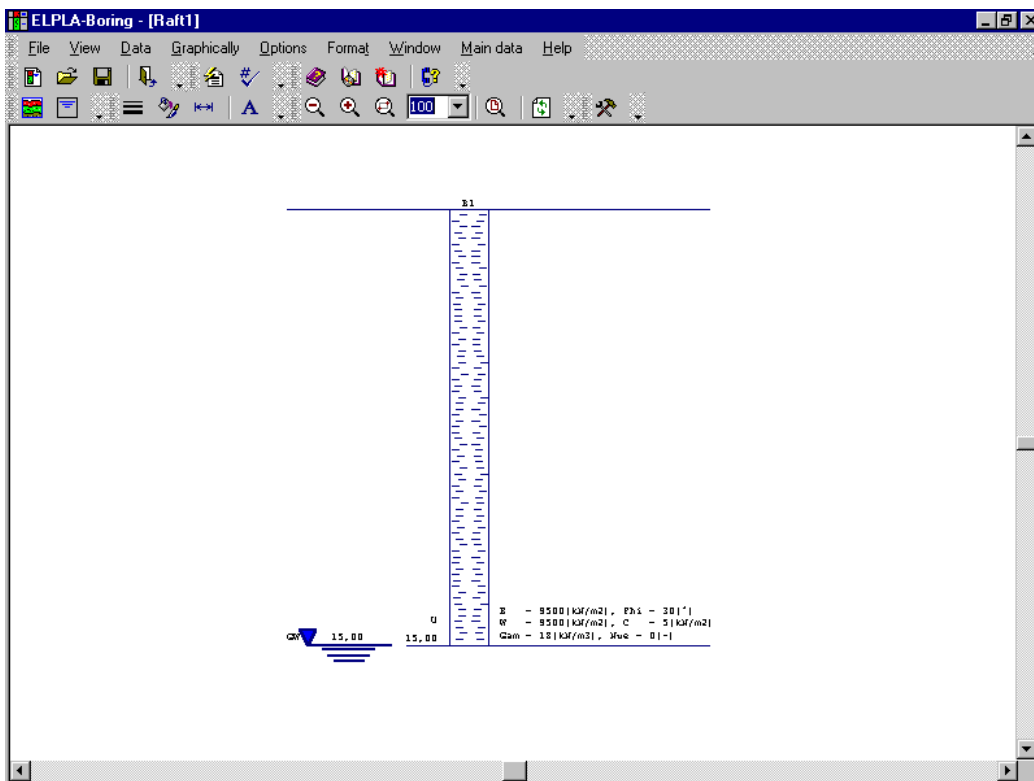

Figure 3-20   Boring log on the screen

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

- Choose "Save boring logs" command from "File" menu in Figure 3-20 to save the data of the boring log
- Choose "Close boring logs" command from "File" menu in Figure 3-20 to close *ELPLA-Boring* and to return to the main window of *ELPLA-Data*

2.5   Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu of *ELPLA-Data* window. The following embedded program in Figure 3-21 appears with default foundation properties. The data of foundation properties for the current example, which are required to define, are raft material and raft 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.
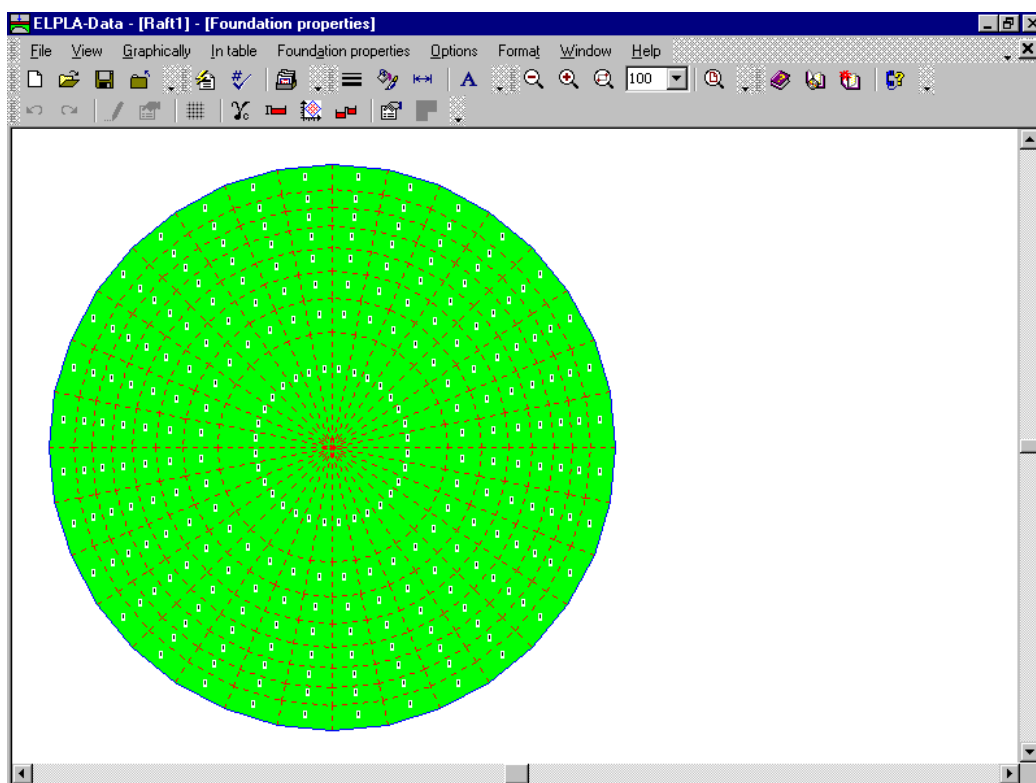

Figure 3-21   "Foundation properties" embedded program

To enter the raft material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 3-21. The following list box in Figure 3-22 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 3-22 enter E-Modulus of the raft, *Poisson's* ratio of the raft and raft thickness
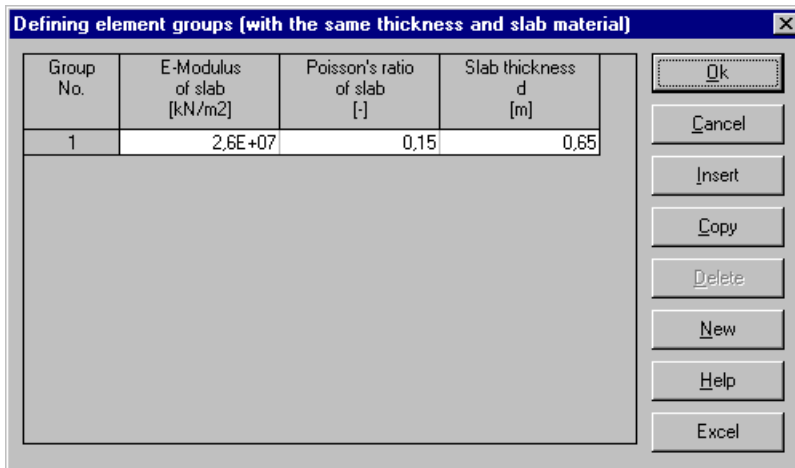- Click "OK" button

Figure 3-22   "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 3-21. The following dialog box in Figure 3-23 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 edit box "Unit weight of the foundation"
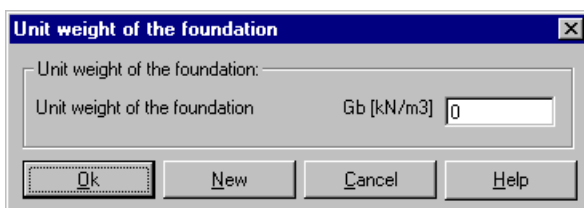- Click "OK" button

Figure 3-23   "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 3-21 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 3-21 to close the "Foundation properties" embedded program and to return to the main window of *ELPLA-Data*

2.6   Loads

To define the loads, choose "Loads" command from "Data" menu of *ELPLA-Data* window. The following embedded program in Figure 3-24 appears.
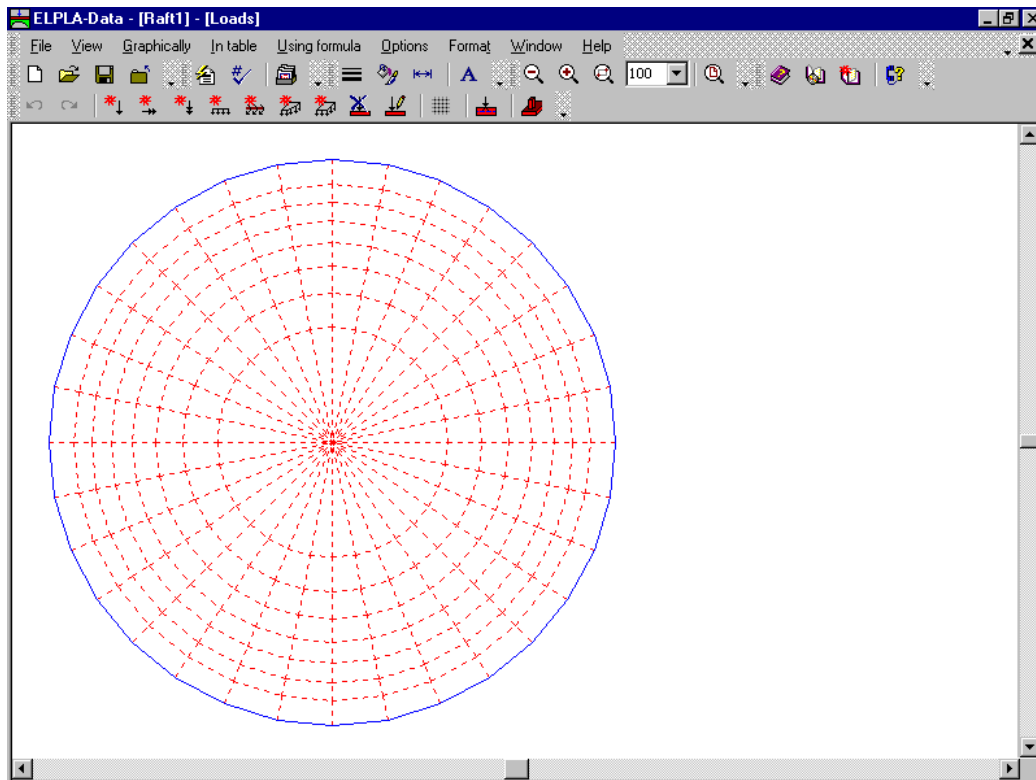
Figure 3-24   "Loads" embedded program

To enter loads

- Choose "Point loads" command from "In Table" menu in the window of Figure 3-24. The following list box in Figure 3-25 appears
- Enter the external point loads P [kN] and their corresponding coordinates (x, y) in the list box of Figure 3-25. This is done by typing the value in the corresponding cell and then press "Enter" key. The coordinates of the point load are related to the lower-left corner of the raft (local coordinates)
- Click "OK" button

Figure 3-25   "Point loads P" list box

After you have completed the definition of all load data, the screen should look like the following Figure 3-26.
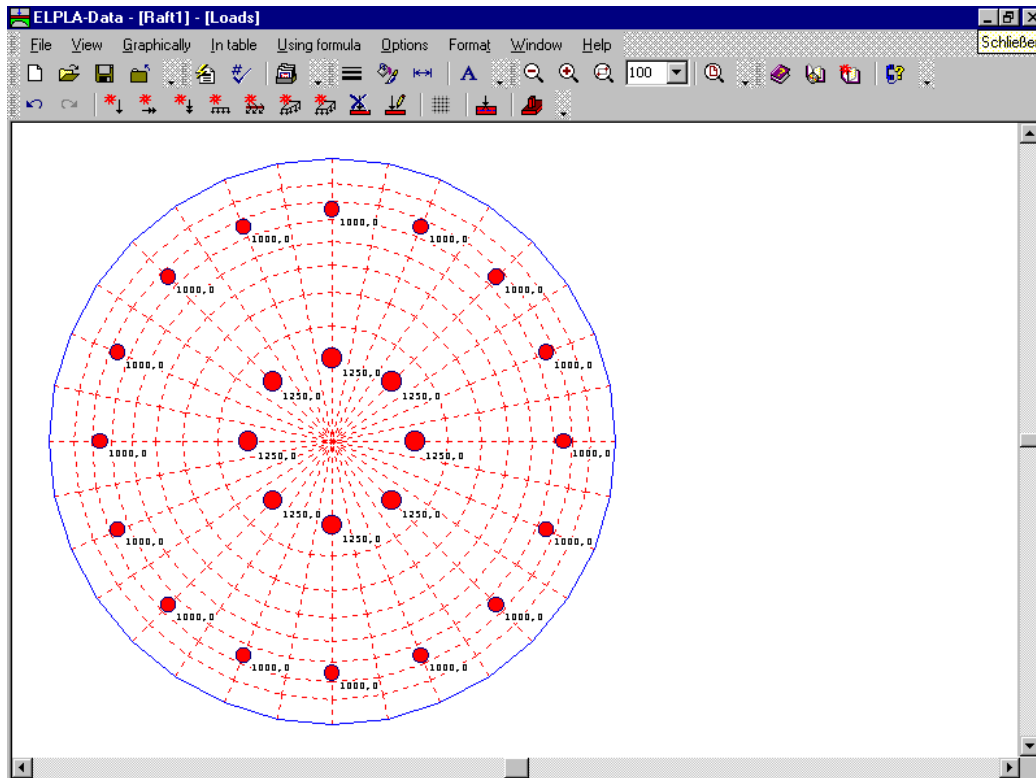

Figure 3-26   Loads on the screen

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

- Choose "Save loads" command from "File menu in Figure 3-26 to save the load data
- Choose "Close loads" command from "File" menu in Figure 3-26 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

The project creation of raft 1 is now complete.

3   Creating the project of raft 2

The data of the two rafts are quite similar except the origin coordinates of the global system, which are chosen to be (0, 0) and (0, 22.5) for rafts 1 and 2 respectively. Project identification is entered here so that the user can distinguish between the two projects. The data of raft 2 are created by first saving the data of raft 1 under a new file name and then modifying the project identification and origin coordinates.

To save the data under a new file name

- Choose "Save project as" command from "File" menu of *ELPLA-Data* window. The following "Save as" dialog box appears, Figure 3-27

In this dialog box

-   Type a file name for the project of raft 2 in the file name edit box, for example "Raft 2"
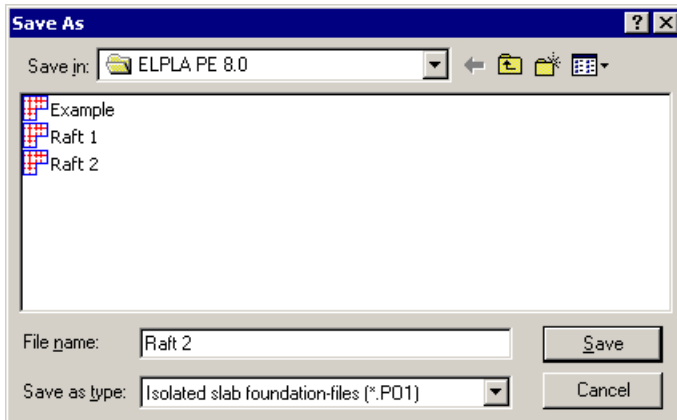-   Click "Save" button

Figure 3-27   "Save as" dialog box

3.1   Modifying the project identification

To modify the project identification of raft 2 choose "Project identification" command from "Data" menu of *ELPLA-Data* window. The dialog box in Figure 3-28 appears.

In this dialog box

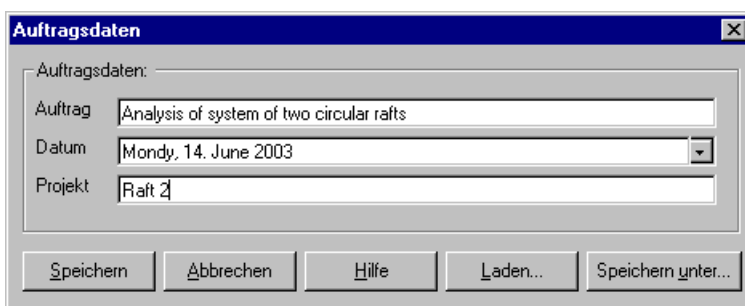-   Type "Raft 2" in the "Project" edit box
-   Click "Save" button

Figure 3-28   "Project identification" dialog box

3.2   Modifying origin coordinates

To modify the origin coordinates of raft 2, choose "Foundation properties" command from "Data" menu of *ELPLA-Data* window. The following embedded program in Figure 3-29 appears.
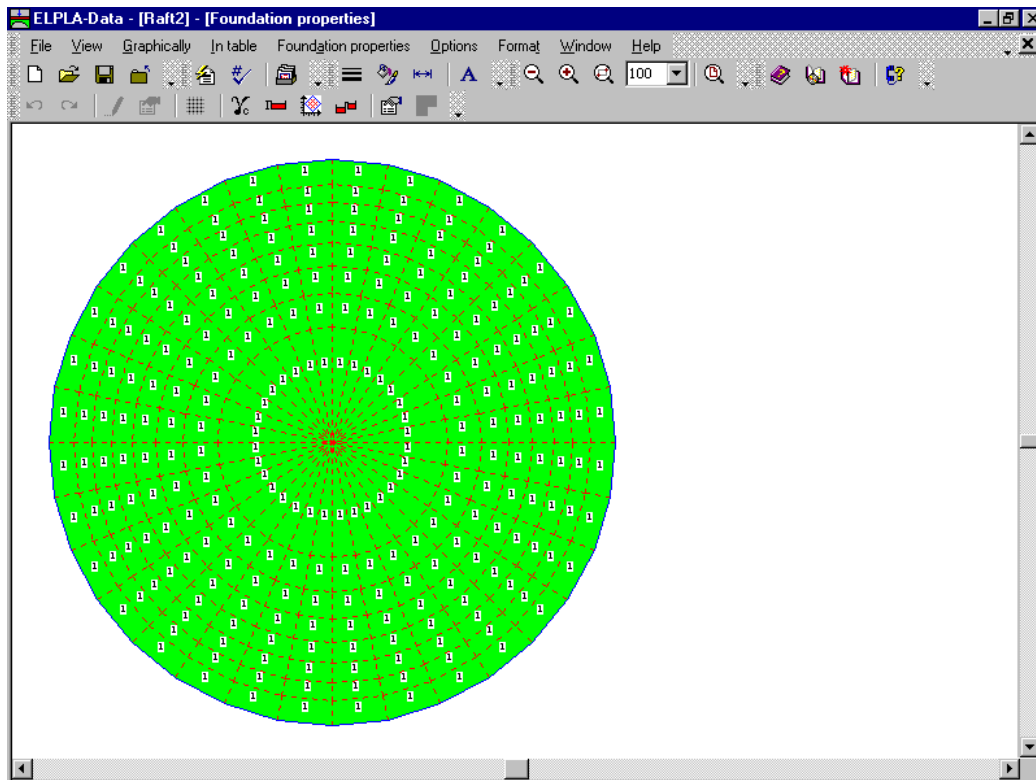
Figure 3-29   "Foundation properties" embedded program

In this "Foundation properties" embedded program

- Choose "Origin coordinates" command from "Foundation properties" menu in the window of Figure 3-29. The following dialog box in Figure 3-30 appears
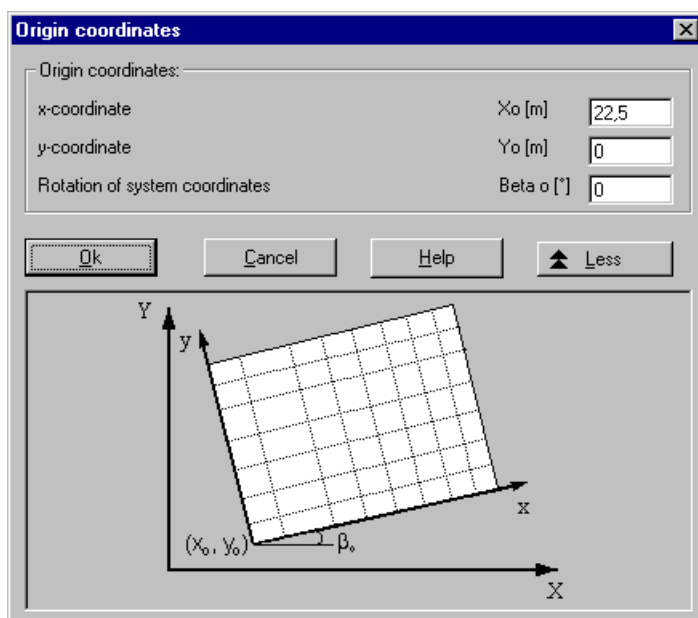- Type 22.5 in the "*x*-coordinate" edit box
- Click "OK" button

Figure 3-30   "Origin coordinates" dialog box

After entering the foundation properties do the following two steps:

- Choose "Save foundation properties" command from "File" menu in Figure 3-29 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 3-29 to close the "Foundation properties" embedded program and return to *ELPLA-Data*

Creating the projects of the two rafts 1 and 2 is now complete.

4 Creating the project of the system of rafts 1 and 2

Data of system of many slabs are defined through the "Data" menu in *ELPLA-Data*, which in this case contains the following two commands:

- "Filenames of slabs foundations" command
- "Project identification" command

4.1 Filenames of slab foundations

Choose "New project" command from the "File" menu of *ELPLA-Data* window. The following "Calculation method" wizard in Figure 3-31 appears.
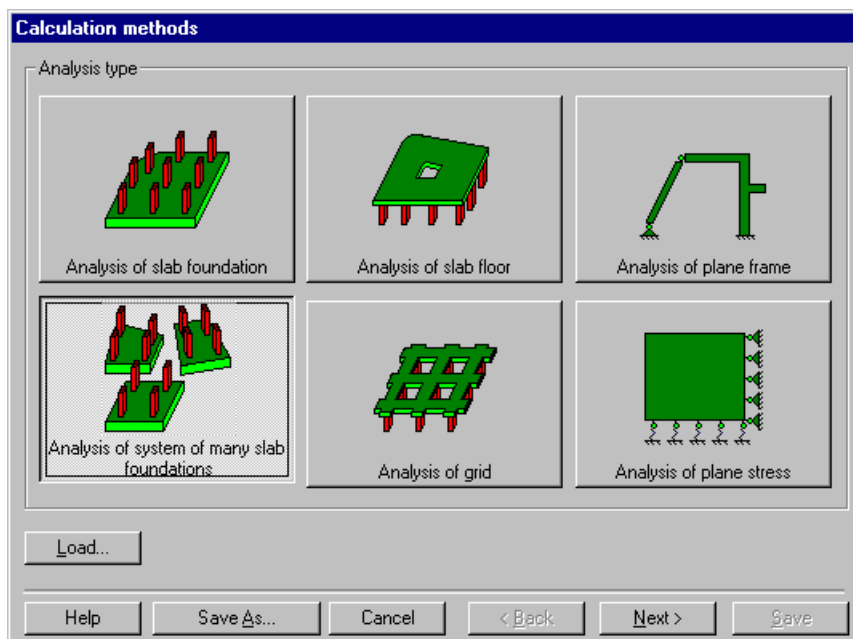

Figure 3-31   "Calculation method" wizard

In this wizard

- Select "Analysis of system of many slab foundations", as the analysis type is a system of two rafts
- Click "Next" button

After clicking "Next" button "Filenames of slab foundations" list box in Figure 3-32 appears.
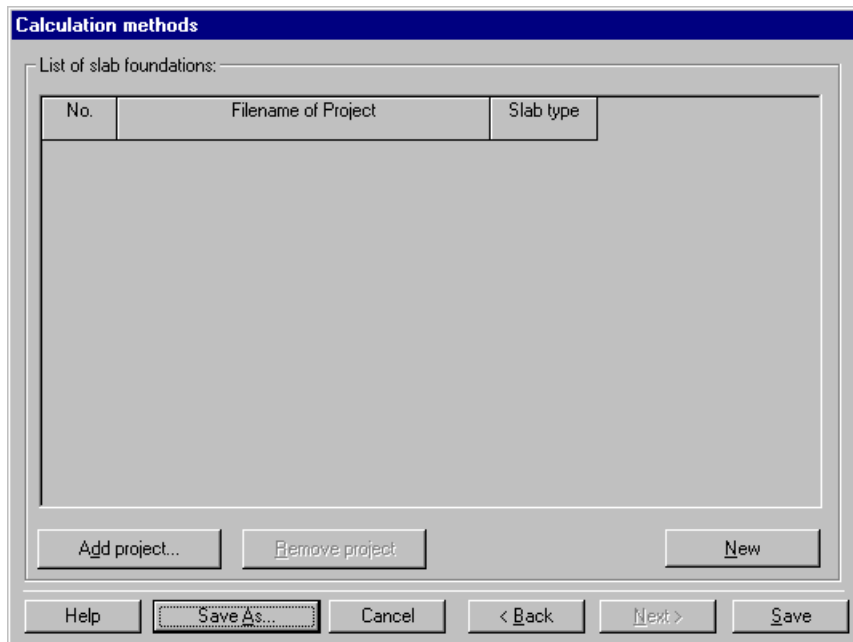

Figure 3-32   "Filenames of slab foundations" list box

To enter the file names of the rafts, which are required to be analyzed as a system of foundations, click "Add project". After clicking "Add project" the "Open" dialog box in Figure 3-33 appears.

In this dialog box

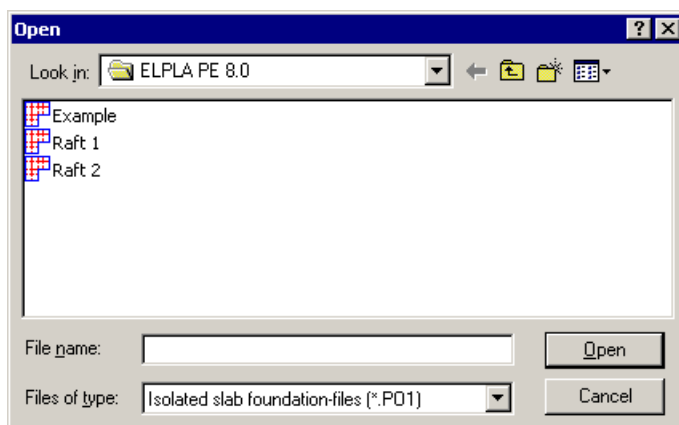- Type the file name of raft 1 in the file name edit box
- Click "Open"

Figure 3-33   "Open" dialog box

Repeat the previous steps and enter the file name of raft 2. After the user has completed the definition of file names of the projects, the list box should look like the following Figure 3-34.
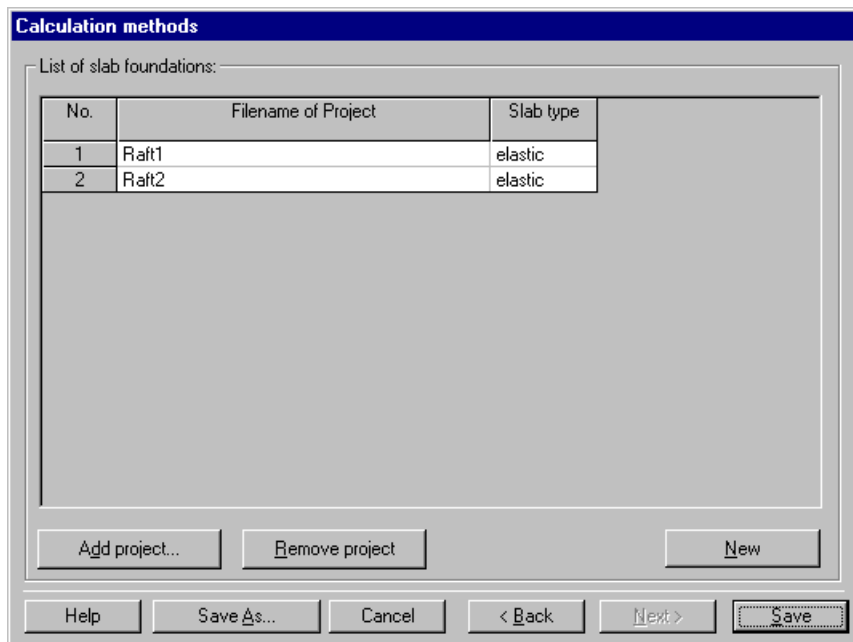
Figure 3-34   "Filenames of slab foundations" list box after entering file names of the projects

Click "Save" button in the "List of filenames of slab foundations" list box in Figure 3-34. After clicking "Save" button the "Save as" dialog box in Figure 3-35 appears.

In this dialog box

- Type a file name for the project of a system of rafts in the file name edit box. For example type "Raft 1 + 2"
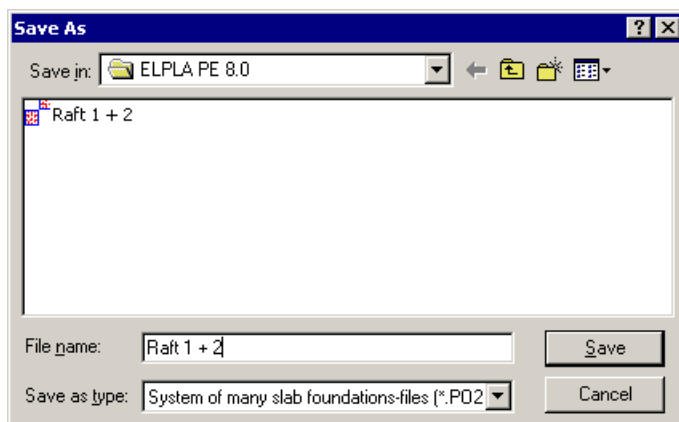- Click "Save" button

Figure 3-35   "Save as" dialog box

4.2   Project identification

The project of a system of rafts is considered as an independent project. Therefore, like the single rafts, a new identification of the project of the system of rafts must be entered.

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data* window. The dialog box in Figure 3-36 appears.

In this dialog box

- Click "Load" button and open the project identification data of raft 1
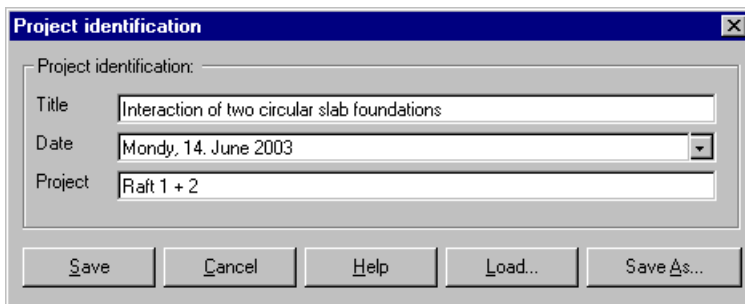- Modify "Raft 1" to "Raft 1 + 2" in the "Project" edit box
- Click "Save" button

Figure 3-36   "Project identification" dialog box

Creating the project is now complete. The next step is analyzing the problem.

5   Carrying out the calculations
5.1   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* window. Then *ELPLA-Solver* window appears, Figure 3-37.

Figure 3-37   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 the current example the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Determining flexibility coefficients of the soil for system of slabs
- 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.

5.2   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 3-38 appears
- In "Iteration parameters" option box select the option of the iteration condition
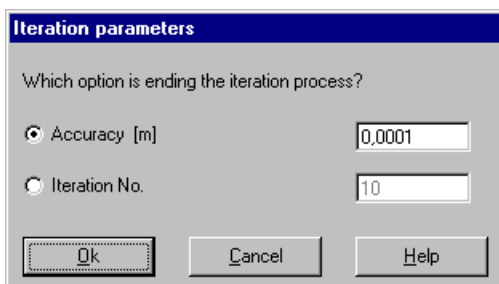- Click "OK" button

Figure 3-38   "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 3-39 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.
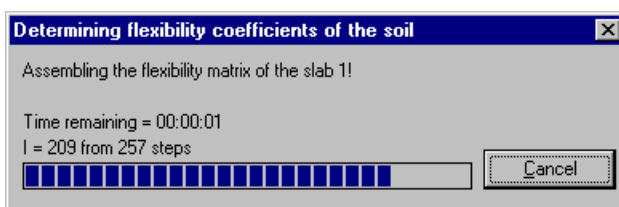

Figure 3-39   Analysis progress menu

Iteration process

Information about the convergence progress of the computations is displayed in the "Iteration process" list box in Figure 3-40 during the iteration process.
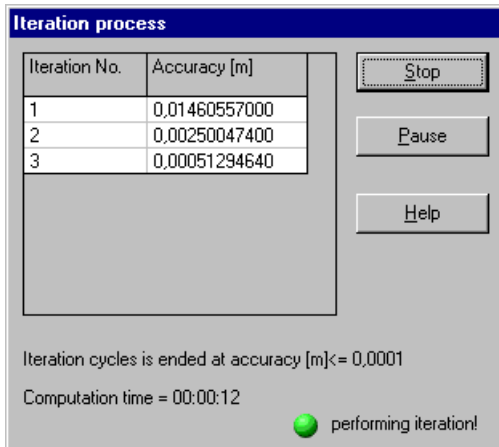

Figure 3-40   "Iteration process" list box

Check of convergence

An accuracy number controls the convergence progress of the solution. The solution is considered convergent if the accuracy number of the step $i + 1$ is less than that of the previous step i. The maximum difference between the soil settlement and the slab deflection in [m] is considered as an accuracy number. A message box appears during the iteration process when the convergence becomes no longer convergent for example (Figure 3-41),

where   the accuracy number of the iteration No. 5 = 0.00021766310 [m]

while   the accuracy number of the iteration No. 4 = 0.00020922720 [m]
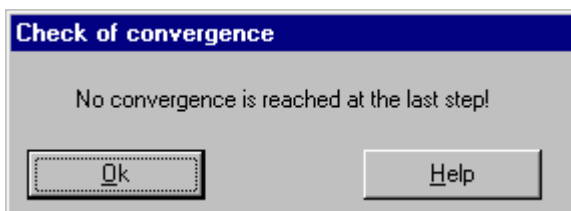

Figure 3-41   "Check of convergence" message box

In this case the user can either stop the iteration process and save the results of iteration No. 5 or continue the iteration process until reaching the convergence.

Do the next steps to continue the iteration process:

- Click "OK" button in the "Check of convergence" message box in Figure 3-41.
  The "Iteration process" list box in Figure 3-42 appears. Note that the "Pause" button changes to "Continue" button
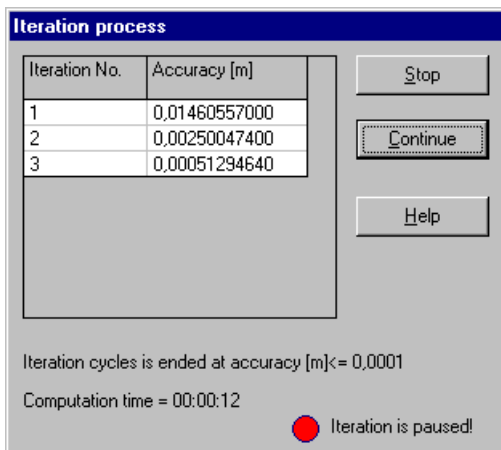- Click "Continue" button

Figure 3-42   "Iteration process" list box

Check of the solution

Once the analysis is complete, a check menu of the solution appears, Figure 3-43. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.
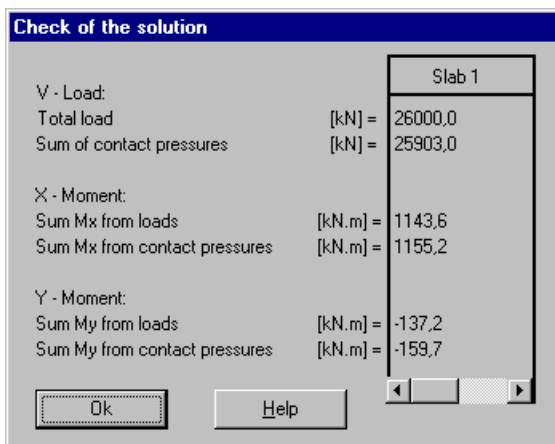

Figure 3-43   Menu "Check of the solution"

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

6   Viewing data and results

ELPLA can view the data and results for each raft separately or for the system of rafts together. Individual data or results can be viewed in a manner similar to that in the previous examples. Here the user will learn to view the results of the system of rafts together through the following two samples.

6.1   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 3-44. *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.
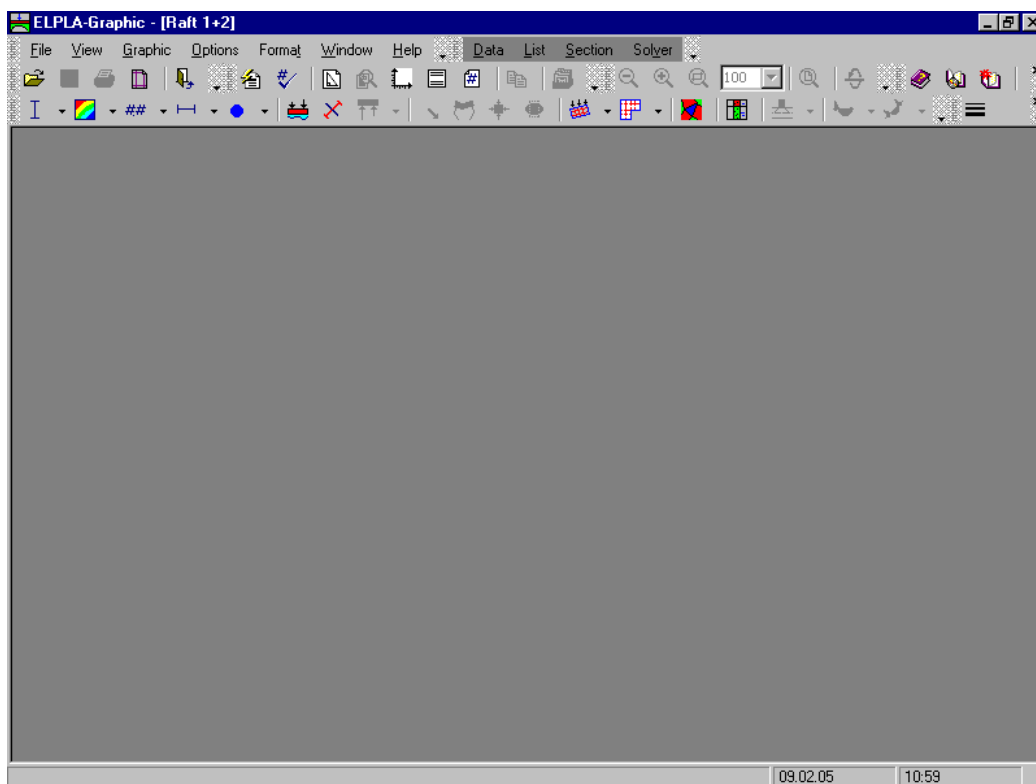
Figure 3-44   Opening screen of the sub program *ELPLA-Graphic*

To view the results as contour lines

-   Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 3-45 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 two rafts together as shown in Figure 3-46.
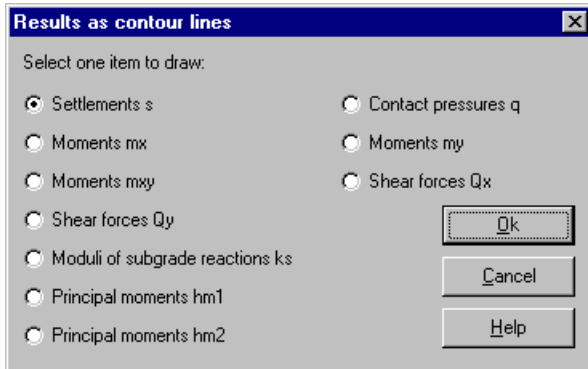

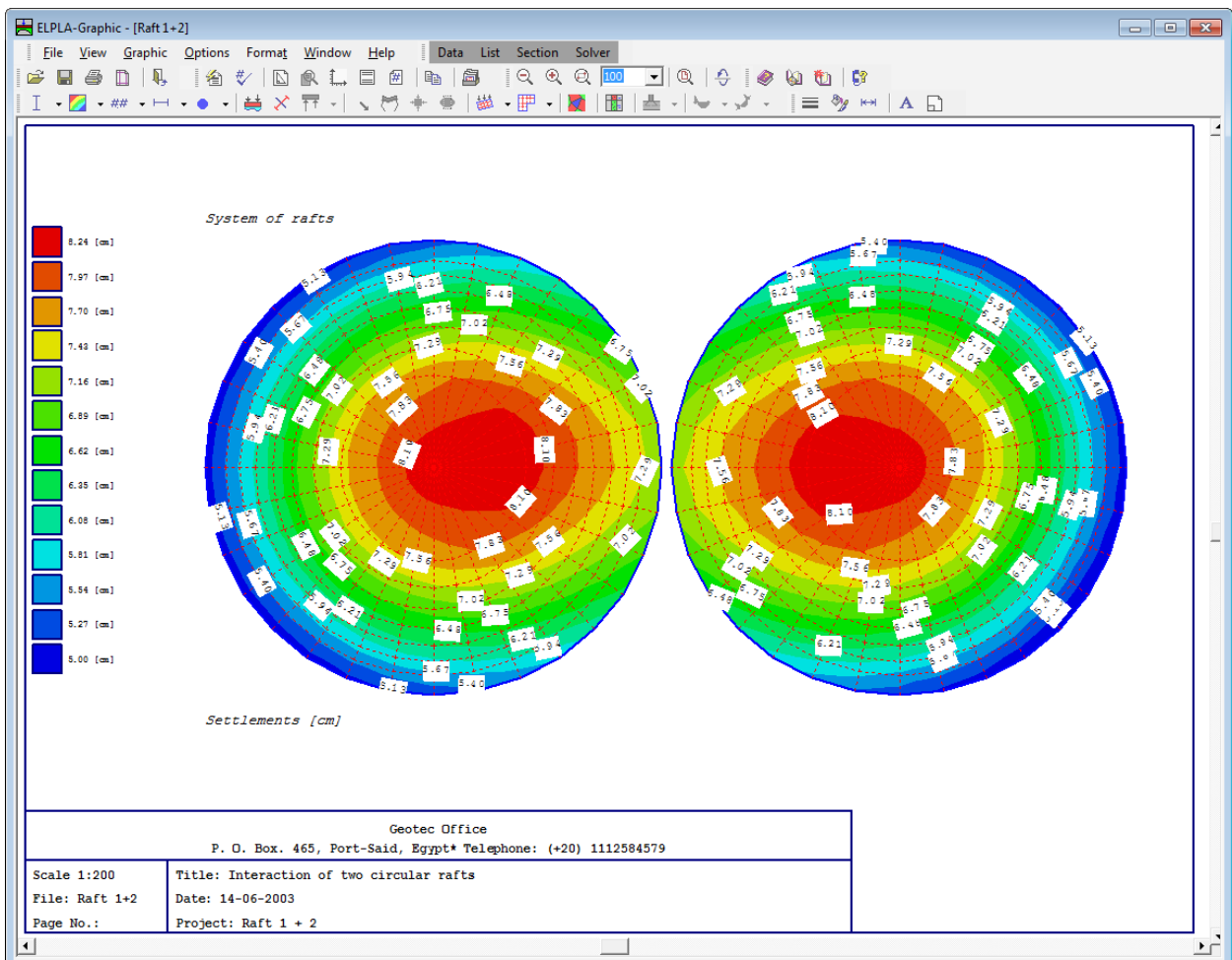Figure 3-45   "Results as contour lines" option box

Figure 3-46   Settlements as contour lines

6.2   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 window appears without file name in the window title bar, Figure 3-47. The disappearance of the file name in the window title bar indicates that *ELPLA-Section* cannot deal with the results of a system of rafts directly.
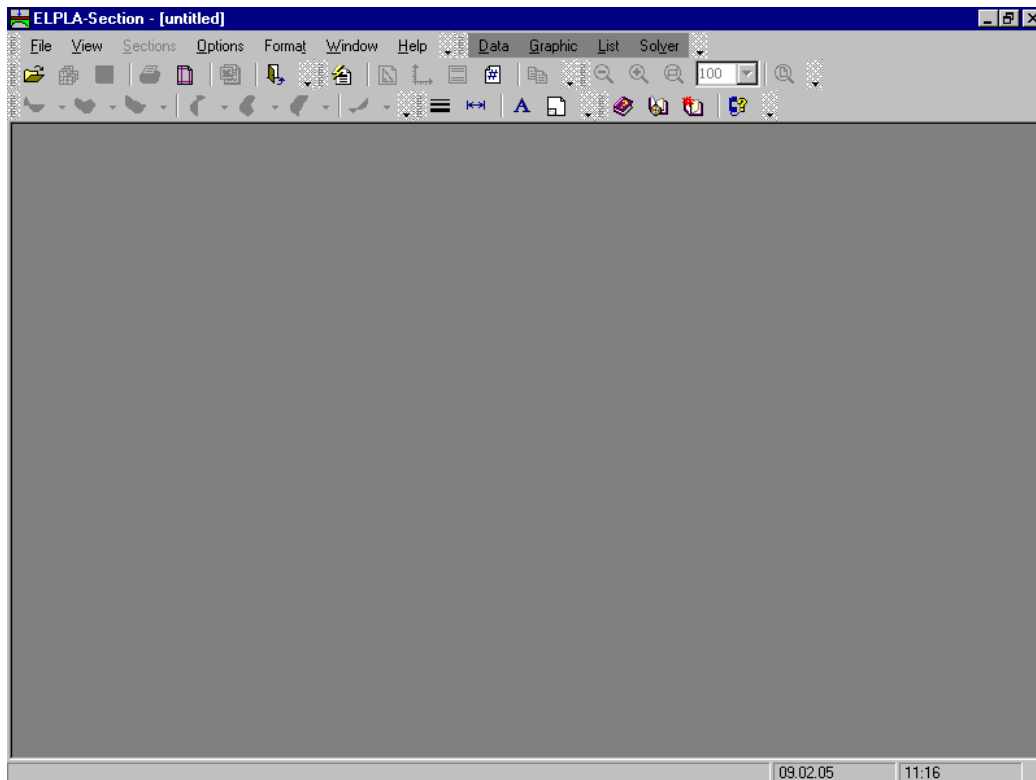
Figure 3-47   Opening screen of the sub program *ELPLA-Section*

To consider the results for the system of rafts together

- Choose "Open" command from "File" menu of *ELPLA-Section*. Then open the project "Raft 1"
- Choose "Combination from many projects" command from "File" menu of *ELPLA-Section*. The following list box in Figure 3-48 appears. *ELPLA-Section* automatically considers the project "Raft 1" in the list of projects to be combined
- Click "Add project" button in "Combination from many projects" list box. Then open the project "Raft 2"
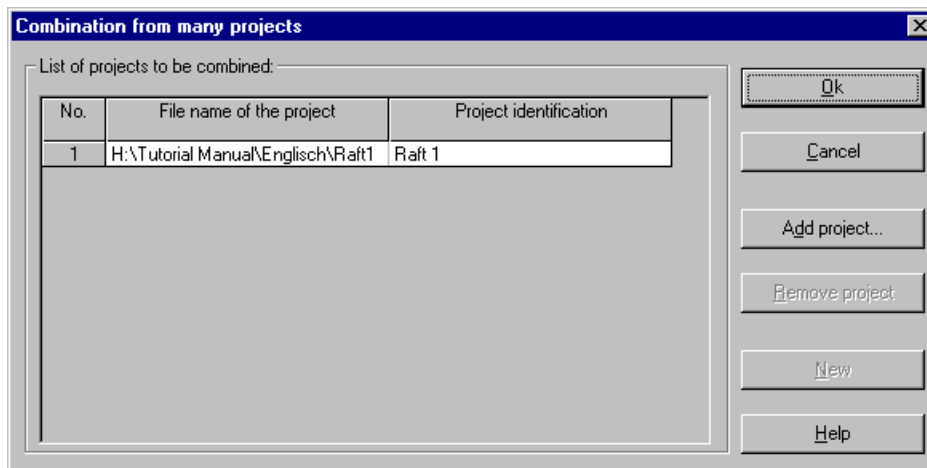- Click "OK" button in the "Combination from many projects" list box

Figure 3-48   "Combination from many projects" list box

To plot a section in *x*-direction

- Choose "Section in *x*-direction" command from "Sections" menu of *ELPLA-Section.*
  The following option box in Figure 3-49 appears
- In "Section in *x*-direction" option box select "Settlements *s*" as a sample for the results to
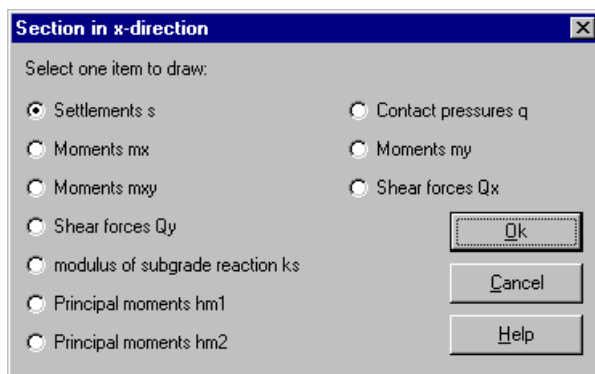 be plotted in a diagram
- Click "OK" button

Figure 3-49   "Section in *x*-direction" option box

The following option box in Figure 3-50 appears, to specify the section in *x*-direction that is required to be plotted.

In this dialog box

- Type 11 in the "Section at *y*-coordinate" edit box, to plot a diagram at the middle of the
  two rafts
- Click "OK" button

The settlements are now plotted in a diagram at the middle of the two rafts together as shown in Figure 3-51.
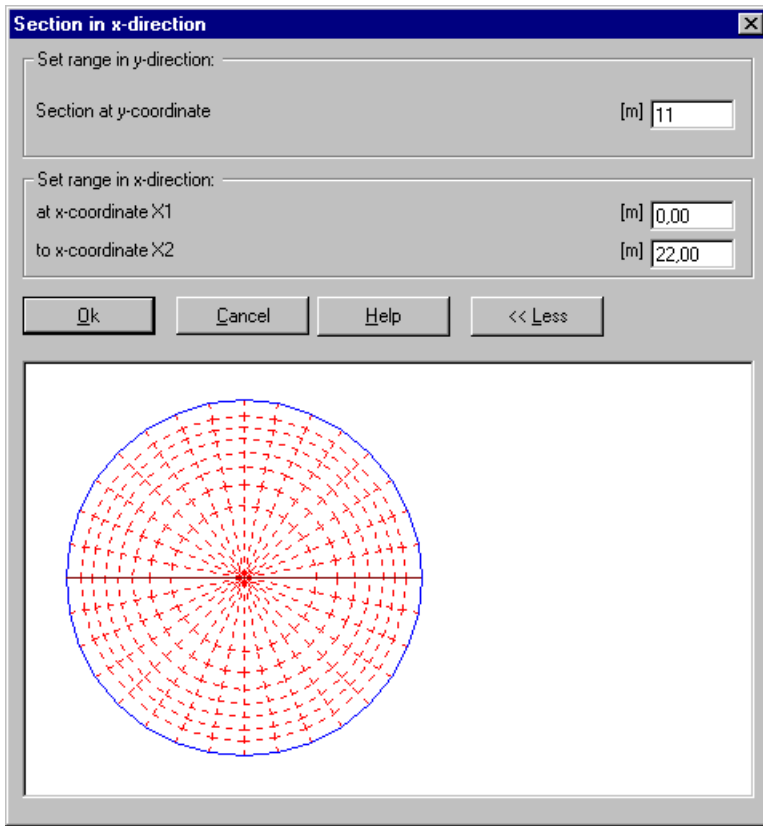
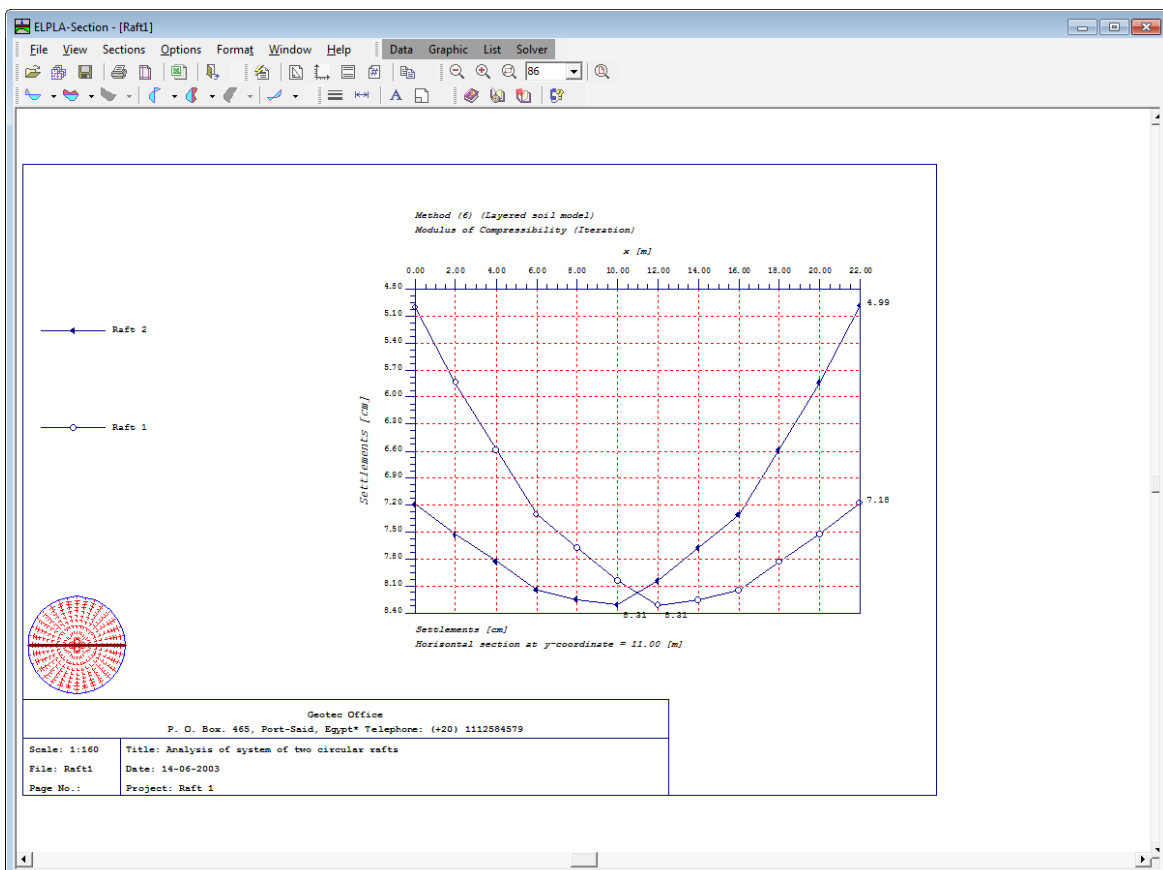Figure 3-50   "Section in *x*-direction" dialog box

Figure 3-51   Diagram of settlements in *x*-direction at nodal row No. 14 for the two rafts

7   Index