Example 10

Analysis of raft with straight and curved borders

Contents Page

1 Description of the problem

An example of a raft with straight and curved borders is selected to illustrate some of the essential features of *ELPLA*.

1.1 Loads and dimensions

The raft is subjected to 12 concentrated loads, each is 500 [kN] as shown in Figure 10-1 and Table 10-1.

Table 10-1 Point loads *P*

Load No. I [-]	Load value P [kN]	X-coord. x [m]	Y-coord. y [m]
1	500	5.42	9.5
2	500	3.25	4.96
3	500	7	0.5
4	500	8.02	9.5
5	500	11.12	9.5
6	500	11.14	0.5
7	500	15	2
8	500	15	8
9	500	0.75	0.5
10	500	16.25	4.98
11	500	3.84	0.5
12	500	8	5

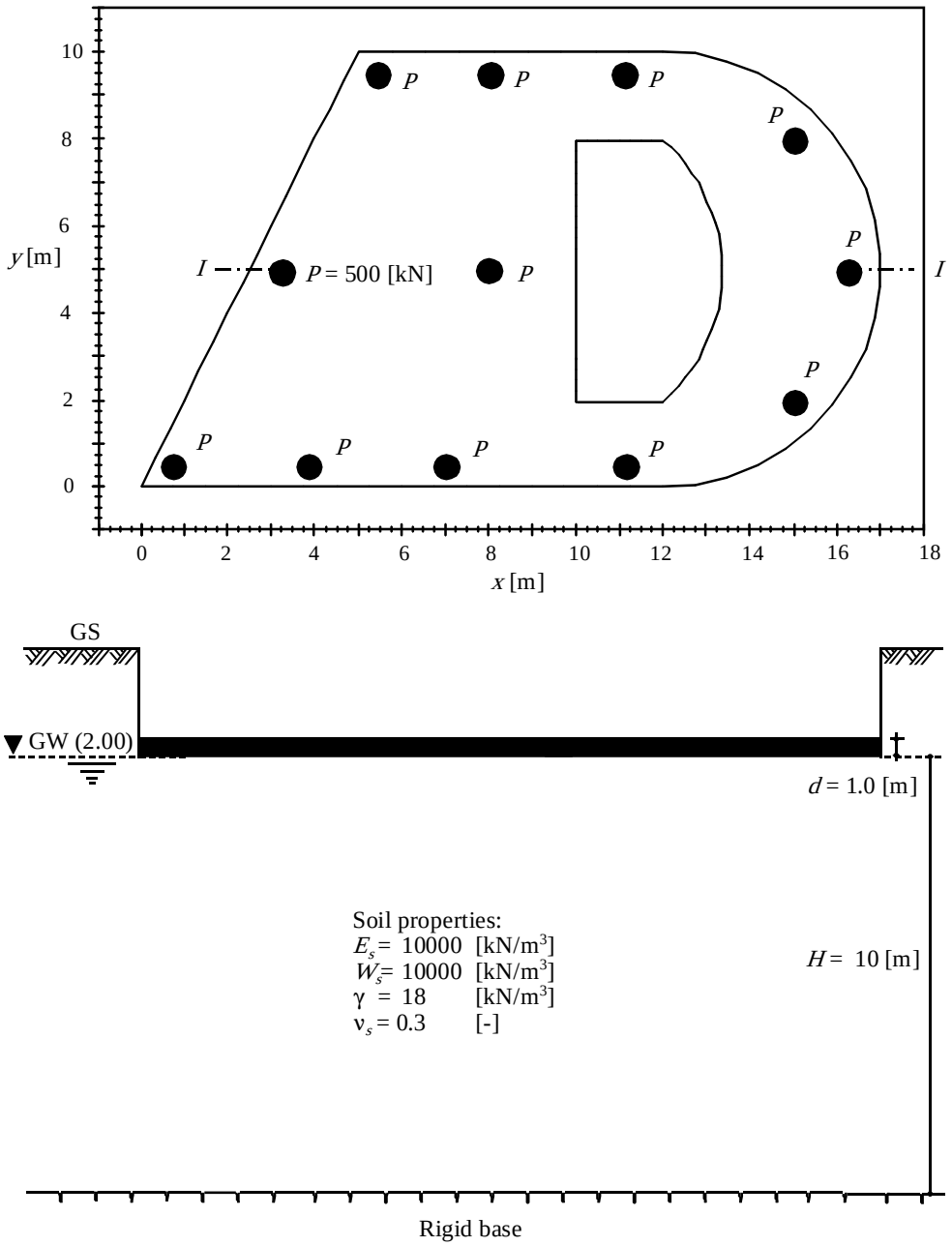


Figure 10-1 Plan and section elevation with raft geometry [m] and acting loads [kN]

1.2 Foundation material and thickness

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

Raft thickness	d	= 1.00	[m]
Young's modulus	E_b	= 2 * 10^7	[kN/m^2]
Unit weight	γ_b	= 25	[kN/m^3]
Poisson's ratio	ν_b	= 0.25	[-]

1.3 Soil properties

The subsoil under the raft consists of a layer of silt, 12 [m] thick, the water table level is 2 [m] below the ground surface. Foundation level is also 2 [m] below the ground surface as shown in Figure 10-1. The soil is supposed to have the following parameters:

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

1.4 Mathematical model

In this example, the Modulus of compressibility method (method 6) 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 a raft foundation. The example will be processed step by step to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation method" wizard appears, Figure 10-2. This wizard will guide you through the steps required to create the project. As shown in Figure 10-2, the first form of the wizard is the "Analysis type" form. In this form, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a foundation problem, select "Analysis of slab foundation" then click "Next" button to go to the next form.

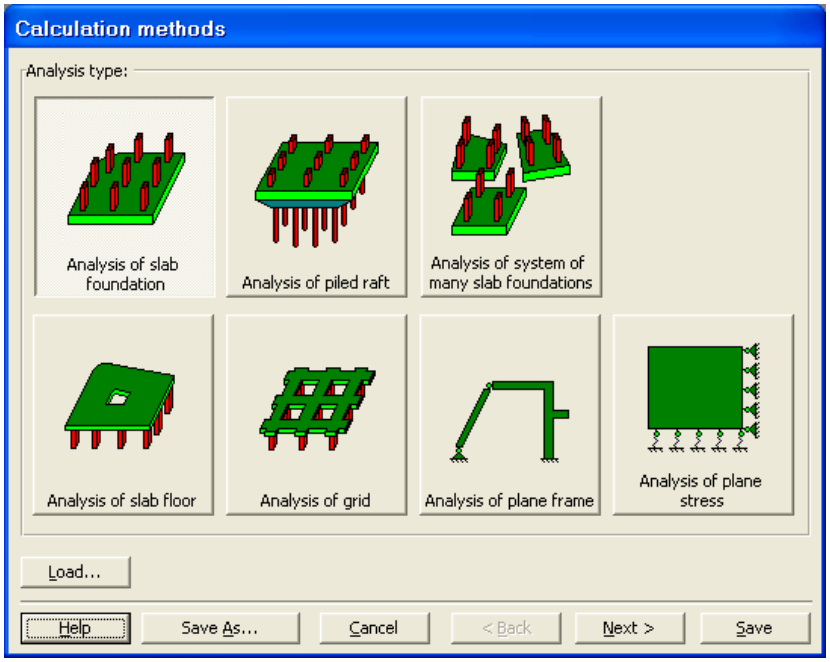


Figure 10-2 "Calculation method" wizard with "Analysis type" form

After clicking "Next" button, the "Calculation methods" form appears, Figure 10-3.

To define the calculation method

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

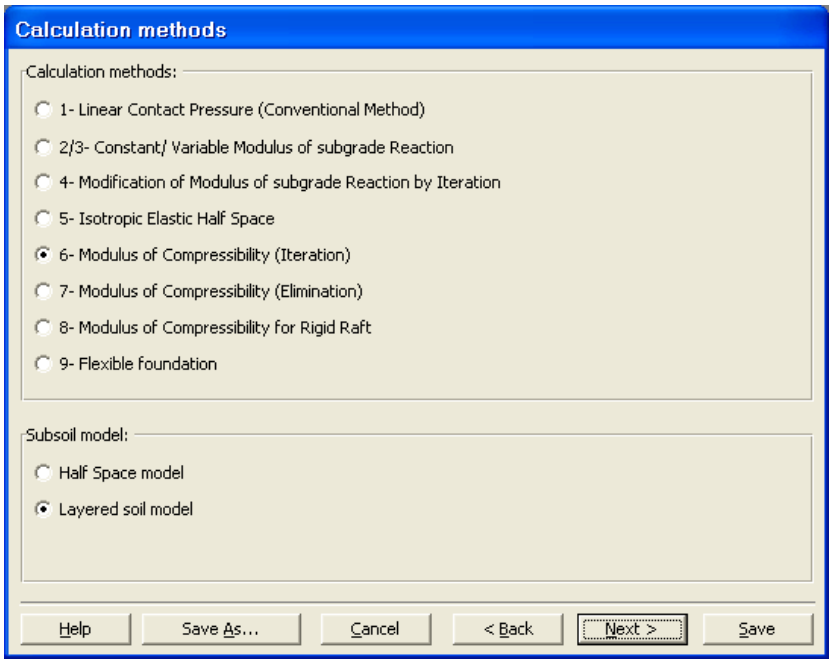


Figure 10-3 "Calculation methods" form

The next form is the "System symmetry" (Figure 10-4). In this form choose "Unsymmetrical system" then click "Next" button.

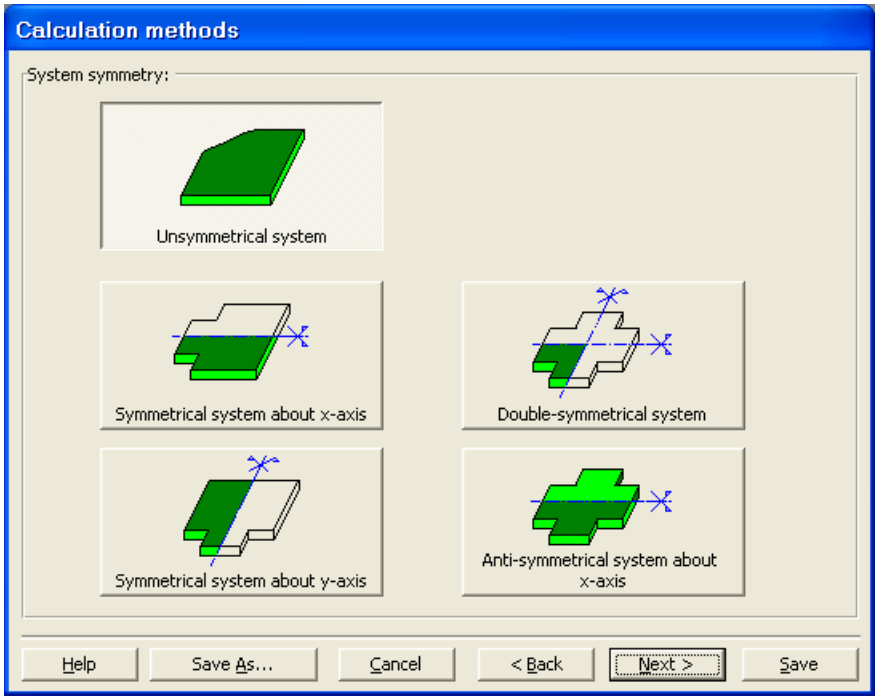


Figure 10-4 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 10-5. 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.

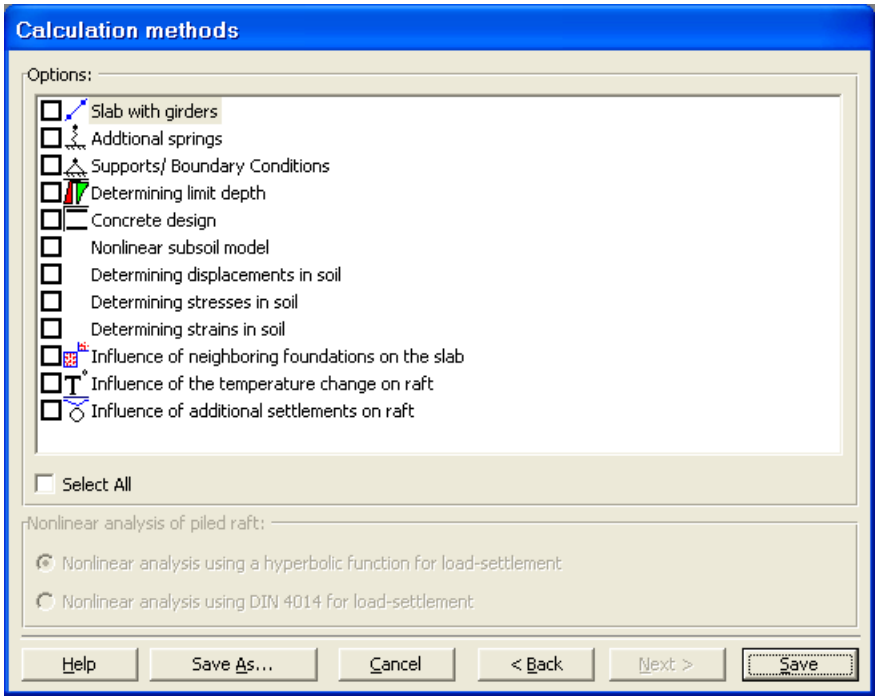


Figure 10-5 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 10-6.

In this dialog box

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

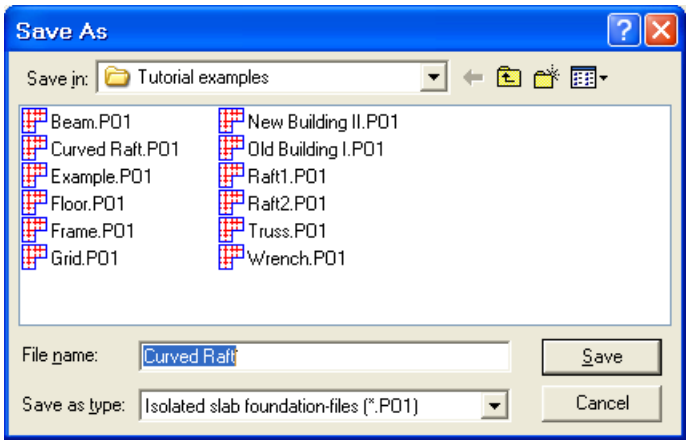


Figure 10-6 "Save as" dialog box

ELPLA will activate the "Data" menu. In addition, the file name of the current project [Curved Raft] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 10-7. 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 10-7. *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.

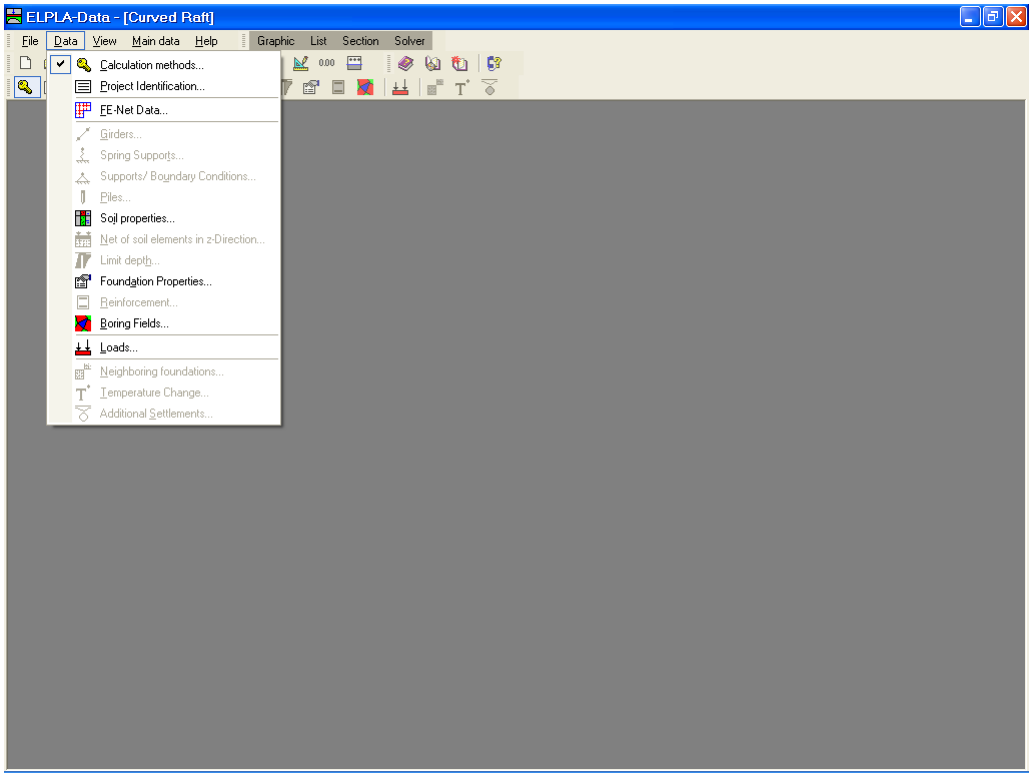


Figure 10-7 *ELPLA-Data* after defining the calculation method

2.2 Project identification

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 10-8 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a raft with straight and curved borders"
- Type the date of the project in the "Date" edit box
- Type "Tutorial Manual" in the "Project" edit box
- Click "Save" button

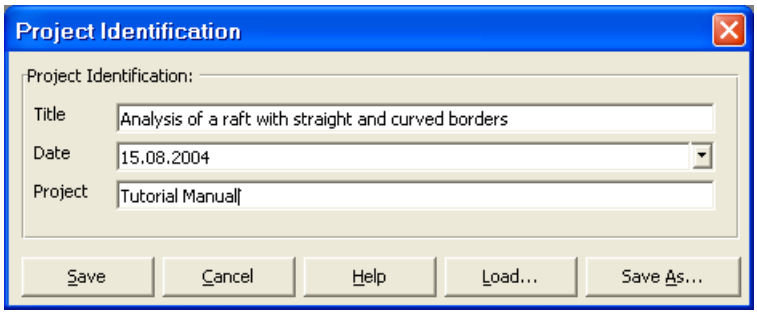


Figure 10-8 "Project identification" dialog box

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 10-9. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 10-9 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.

To generate the FE-Net

- In the "Slab type" options, choose the irregular slab option
- Click "Next" button to go to the next form

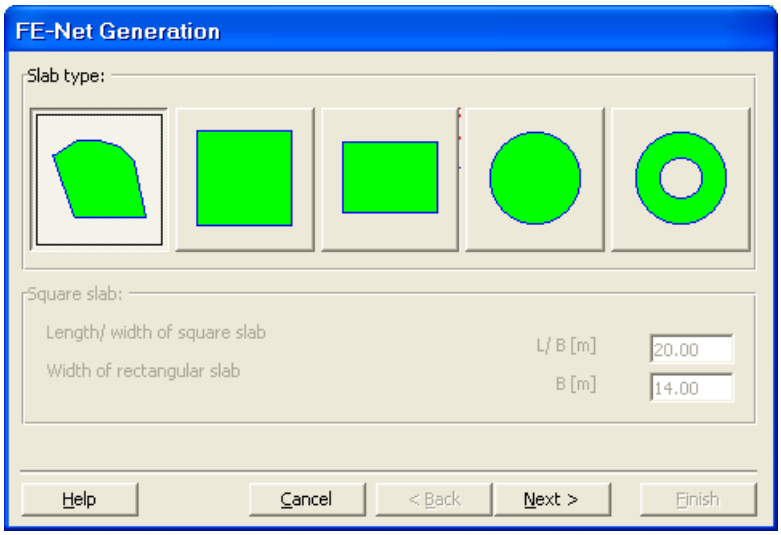


Figure 10-9 "Slab type" form

After clicking "Next" in "Slab type" form, the following "Generation type" form appears, Figure 10-10. *ELPLA* can deal with various type of generations with rectangular/ triangular elements. Choose the triangular element which is the last option as the generation type. Then click "Next" button to go to the next form.

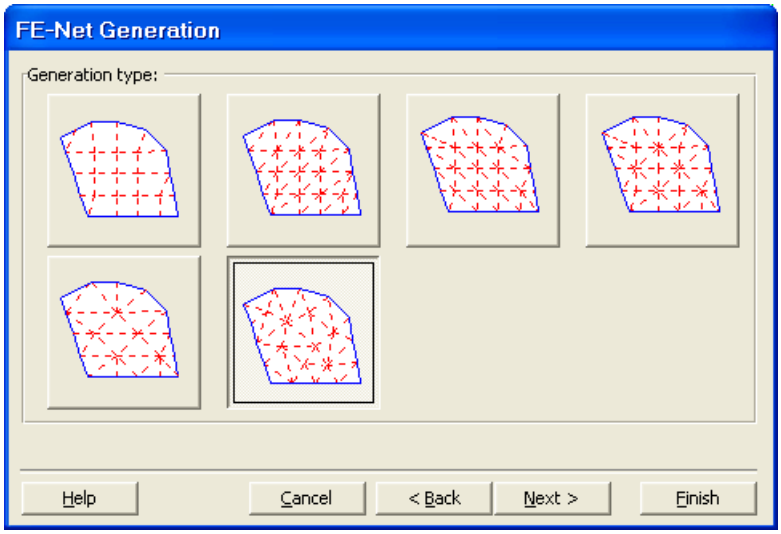


Figure 10-10 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "FE-Net Generation" dialog box in Figure 10-11 appears with the default generation parameters. In this dialog box, click "Finish" button.

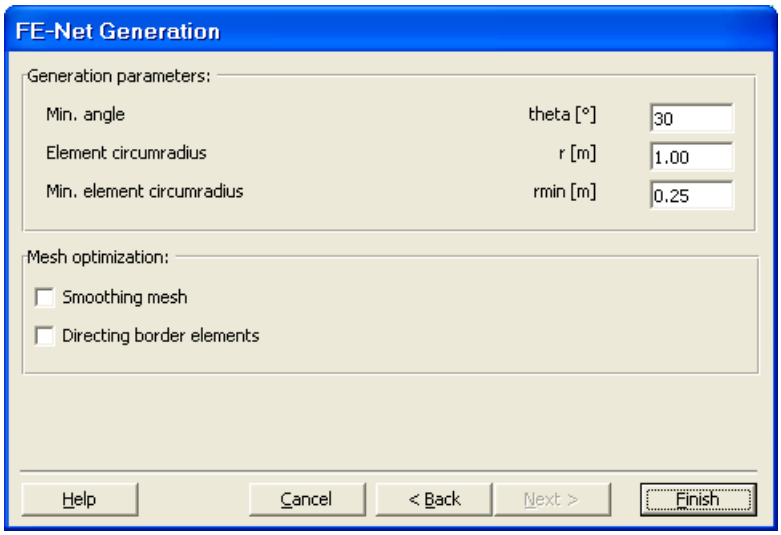


Figure 10-11 "Fe-Net generation" dialog box

ELPLA will generate an imaginary net as shown in the following embedded program, Figure 10-12.

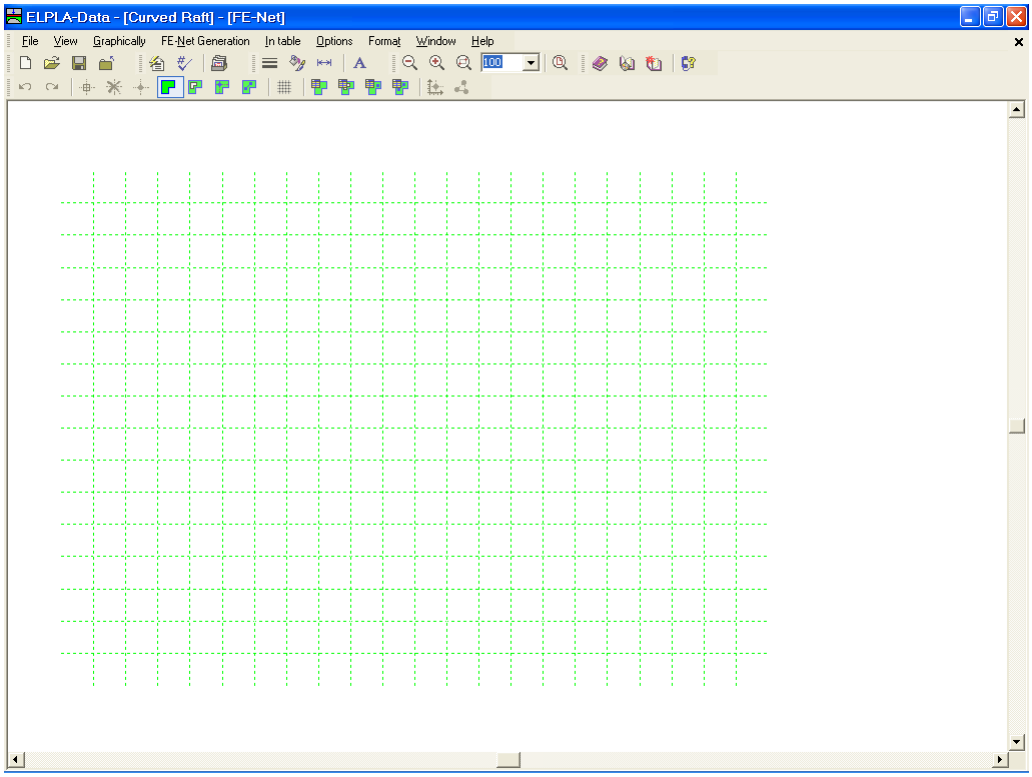


Figure 10-12 Imaginary net of a rectangular area on the screen

You can either draw the raft directly on this imaginary net or input the raft corner points in tabulated form. To input the FE-Net corner points, choose "Slab corners" command from "In Table" menu. The dialog box in Figure 10-13 appears.

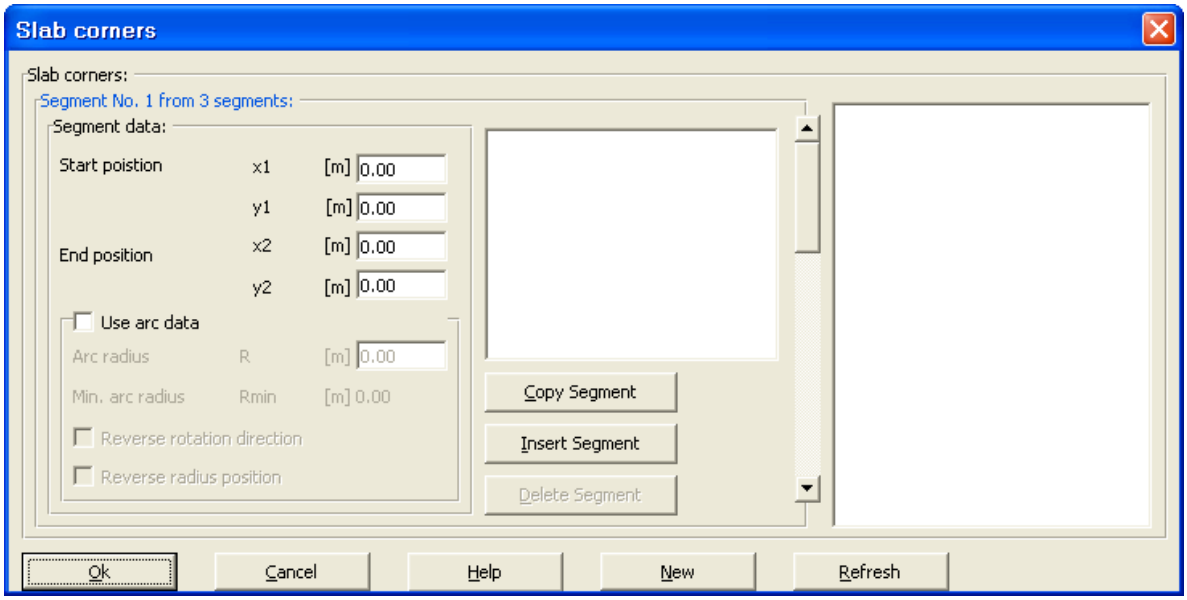


Figure 10-13 "Slab corners" dialog box

To define the first segment of the raft that connects the first two points on the raft of Figure 10-1

- Do not change the default coordinates of the first point, which the program considers as (0.00, 0.00)
- Type 12 in the "*x*2" edit box to define the *x*-coordinate of the second point
- Type zero in the "*y*2" edit box to define the *y*-coordinate of the second point

To define the second segment of the raft

- Use the Scroll bar to define segment No. 2
- Type 12 in the "*x*2 edit box to define the *x*-coordinate of the third point
- Type 10 in the "*y*2" edit box to define the *y*-coordinate of the third point
- Select the "Use arc data" option to convert the line segment to arc segment
- Select the "Reverse rotation direction" option to reverse the direction of rotation
- Select the "Reverse radius position" option to reverse the radius position
- Type 5 in the "Arc radius" edit box to define the radius of the arc segment

In the "Slab corners" dialog box, *ELPLA* assumed that there are at least three segments with three corner points. Since the raft drawing contains four segments, you can use the command "Segment insert" to insert the last segment of the raft. Use the corner points and arc information provided in Table 10-2 to complete the definition of the raft corner points. Repeat the steps used for defining the first two segments to define any segment.

Table 10-2 Raft corner points

Segment No. [-]	Start position		End position		Arc Radius
	x_1 [m]	y_1 [m]	x_2 [m]	y_2 [m]	[m]
1	0	0	12	0	
2	12	0	12	10	5
3	12	10	5	10	
4	5	10	0	0	

After completing the definition of the raft corner points, the slab corner dialog box should be like this in Figure 10-14 where a small sketch of the raft appears in the dialog box picture. Click the "OK" button to see the drawing of the raft outlines as shown in Figure 10-15.

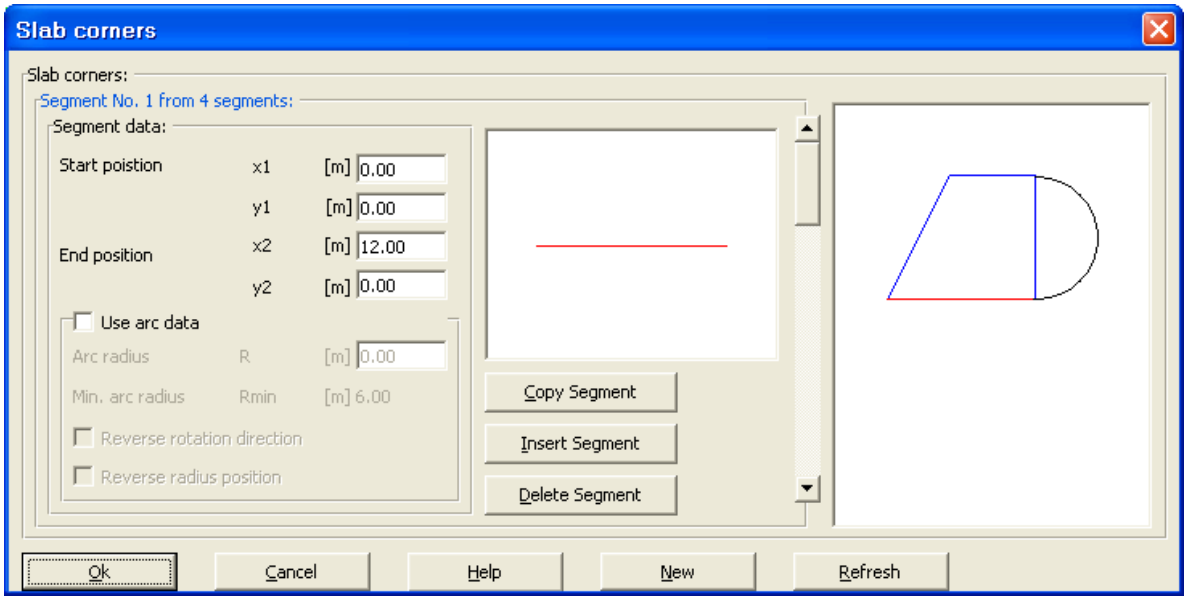


Figure 10-14 "Slab corners" dialog box after defining the raft corner points

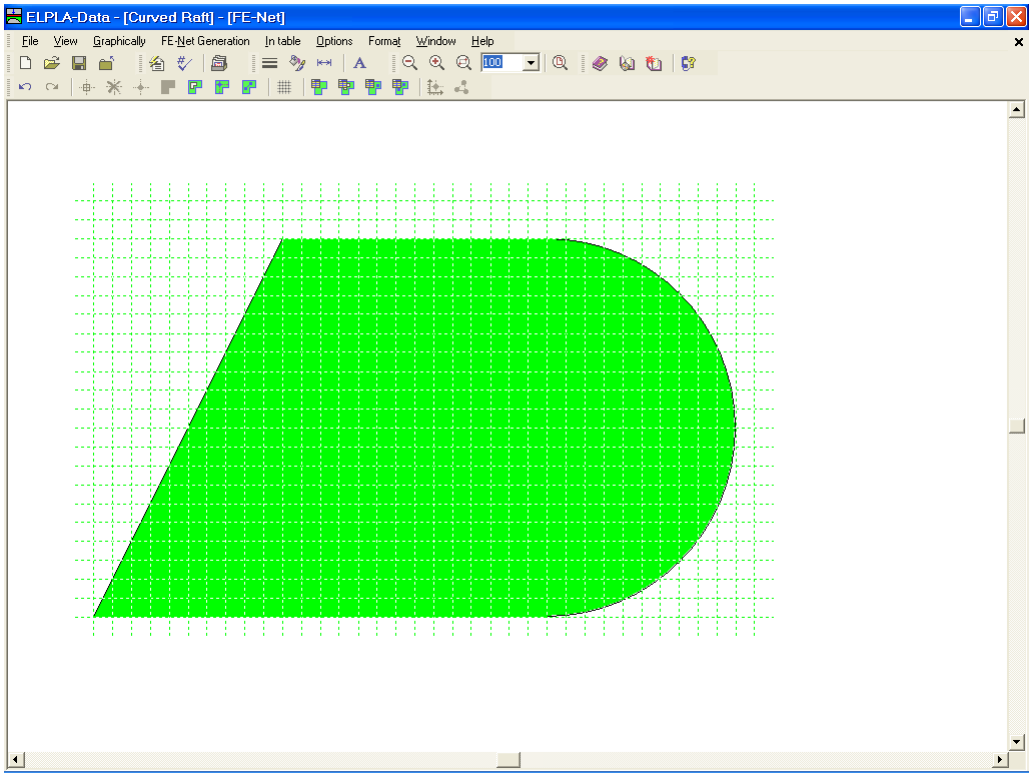


Figure 10-15 Raft outlines

Raft openings

To draw the openings in the raft, choose the command "Opening corners" from "In Table" menu, the dialog box in Figure 10-16 appears.

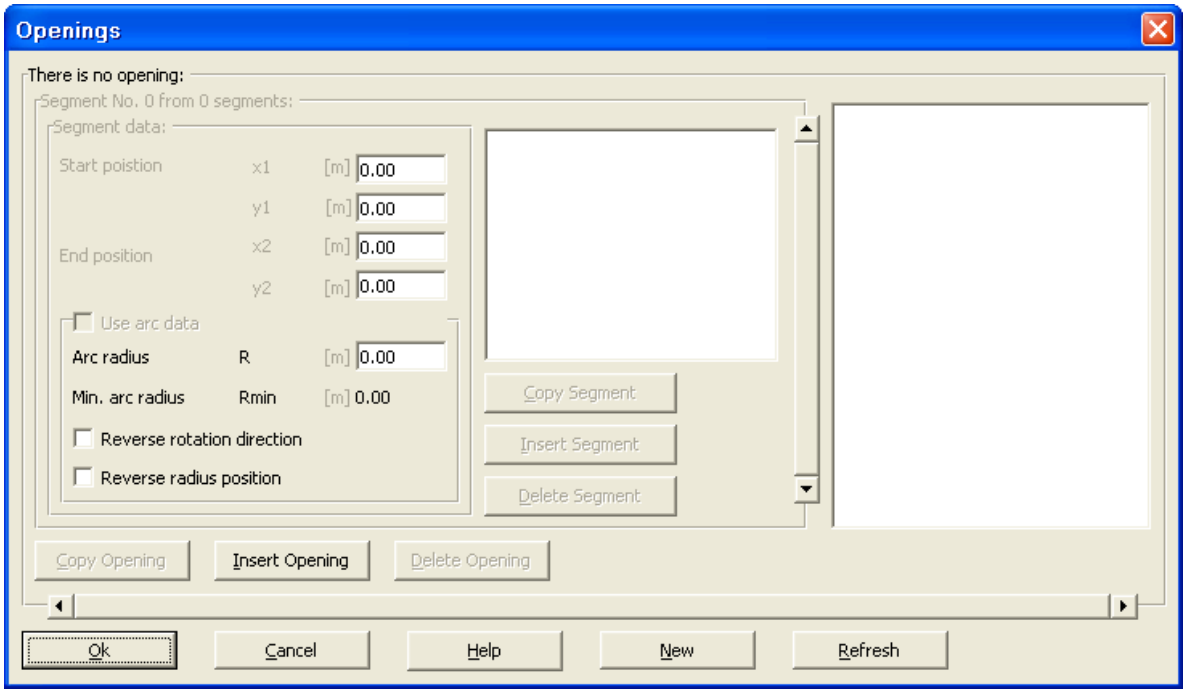


Figure 10-16 "Opening corners" dialog box

In this dialog box, press on "Insert opening" button, then write the coordinates of the opening exactly as in slab corners.

To define the first segment of the raft opening

- Type 10 in the "*x*1" edit box to define the *x*-coordinate of the first point
- Type 2 in the "*y*1" edit box to define the *y*-coordinate of the first point
- Type 12 in the "*x*2" edit box to define the *x*-coordinate of the second point
- Type 2 in the "*y*2" edit box to define the *y*-coordinate of the second point

To define the second segment of the raft opening

- Use the Scroll bar to define segment No. 2
- Type 12 in the "*x*2" edit box to define the *x*-coordinate of the third point
- Type 8 in the "*y*2" edit box to define the *y*-coordinate of the third point
- Select the "Use arc data" option to convert the line segment to arc segment
- Select the "Reverse rotation direction" option to reverse the direction of rotation
- Select the "Reverse radius position" option to reverse the radius position
- Type 4 in the "Arc radius" edit box to define the radius of the arc segment

In the "Opening corners" dialog box, *ELPLA* assumed that there are at least three segments with three corner points. Since the raft opening drawing contains four segments, you can use the command "Segment insert" to insert the last segment of the raft opening. Use the corner points and arc information provided in Table 10-3 to complete the definition of the raft opening corner points. Repeat the steps used for defining the first two segments to define any segment.

Table 10-3 Raft opening corner points

Segment No. [-]	Start position		End position		Arc Radius
	x_1 [m]	y_1 [m]	x_2 [m]	y_2 [m]	[m]
1	10	2	12	2	
2	12	2	12	8	4
3	12	8	10	8	
4	10	8	10	2	

After completing the definition of the raft opening corner points, the opening corners dialog box should be like this in Figure 10-17 where a small sketch of the raft opening appears in the dialog box picture. Click the "OK" button to see the drawing of the opening outlines as shown in Figure 10-18.

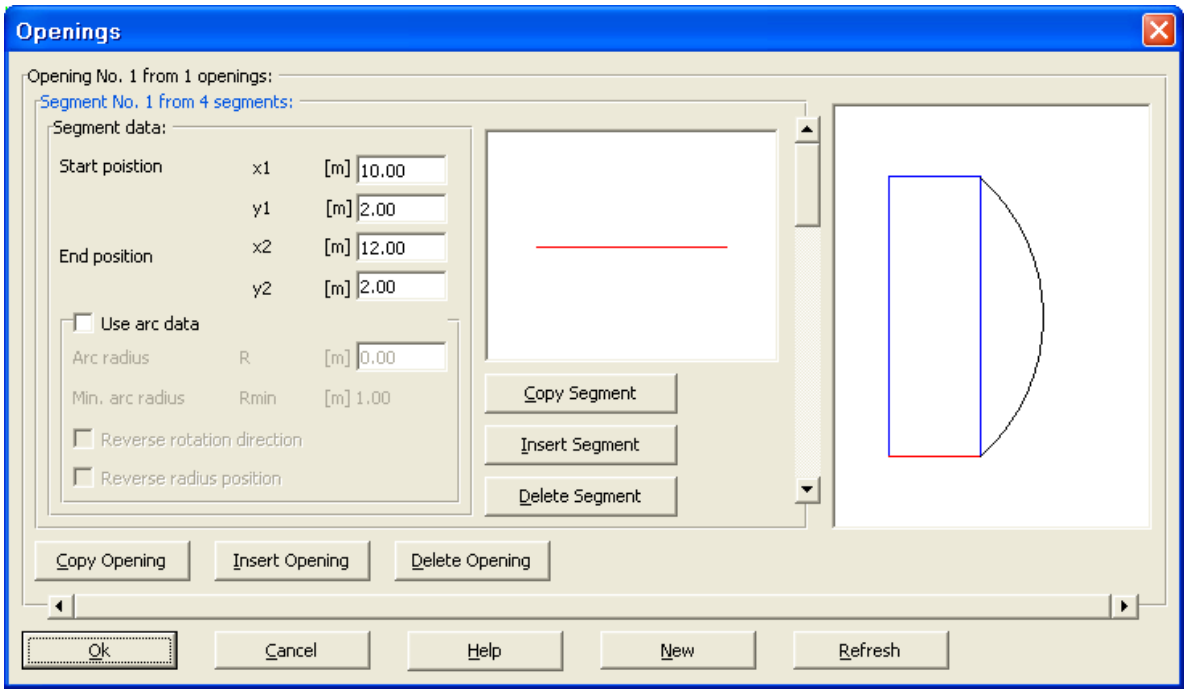


Figure 10-17 "Opening corners" dialog box after defining the opening corner points

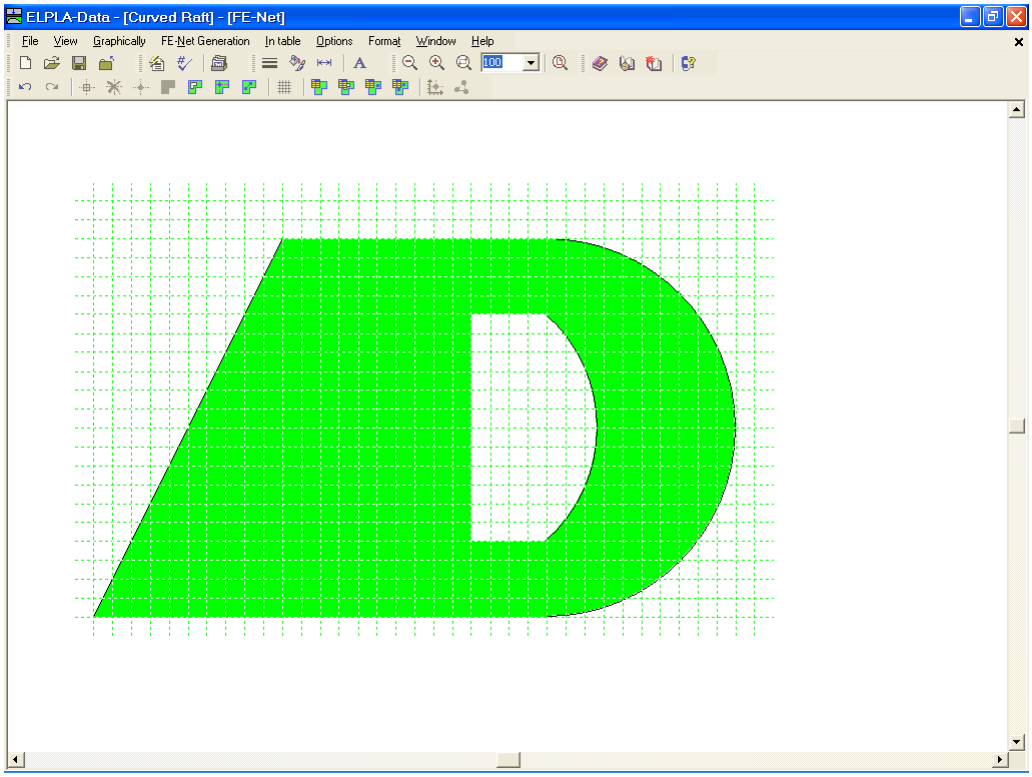


Figure 10-18 Raft opening outlines

To complete the FE-Net generation of the problem, choose "Generating FE-Net" command from "FE-Generation" menu. The following "Generation of FE-Net" dialog box appears.

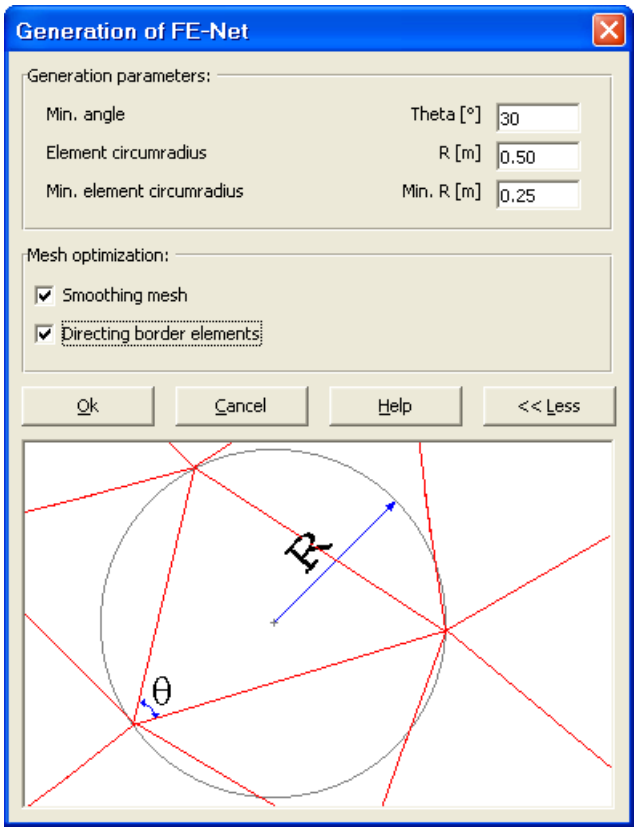


Figure 10-19 "Generation of FE-Net" dialog box

To enter the generation data

- Type 0.5 in the "Element circumradius" edit box to define the radius of circles containing elements
- Type 0.25 in the "Min. element circumradius" edit box to define the minimum radius of circles containing elements
- Check the "Smoothing mesh" option to optimize the dimension of FE-Net by making all elements having nearly the same area as possible as
- Click "OK" button

After clicking the "OK" button, FE-Net generation progress menu in Figure 10-20 appears in which various phases of the generation are progressively reported. After finishing the generation of the FE-Net and creating the raft FE-Net as shown in Figure 10-21, do the following two steps:

- Choose "Save FE-Net" command from "File" menu in Figure 10-21 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 10-21 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*.

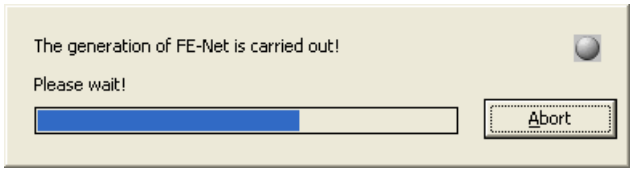


Figure 10-20 "FE-Net generation" progress menu

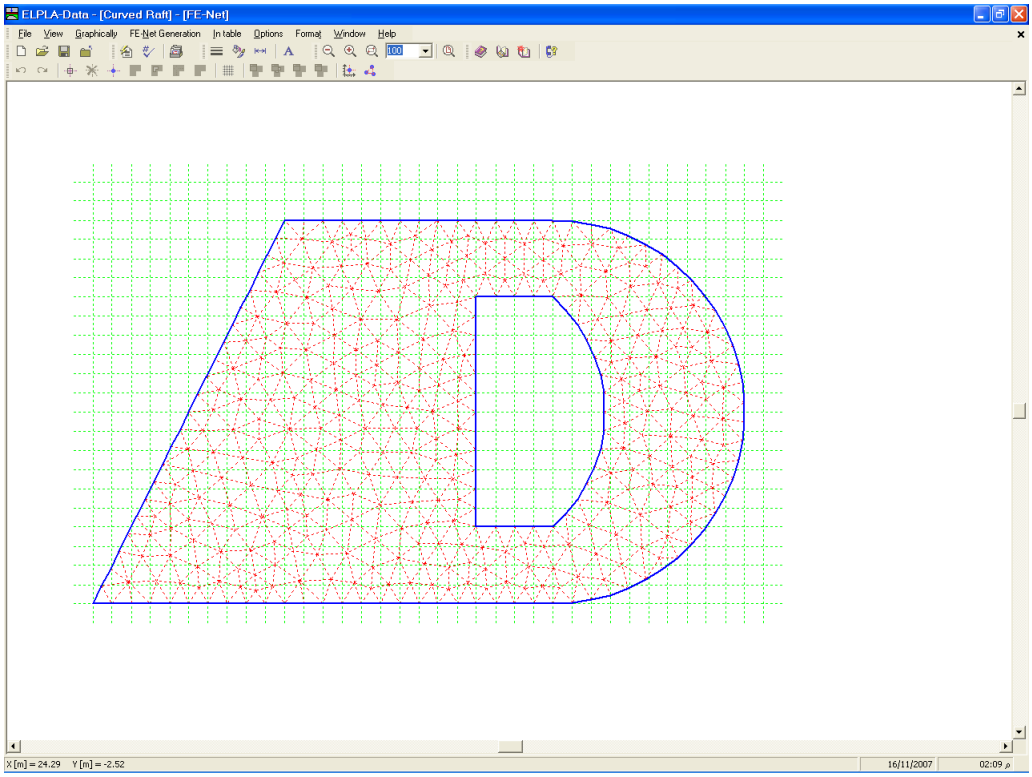


Figure 10-21 Final FE-Net of the raft

2.4 Soil properties

To define the soil properties, choose "Soil properties" command from "Data" menu of *ELPLA-Data*. The following sub program in Figure 10-22 appears with a default-boring log.

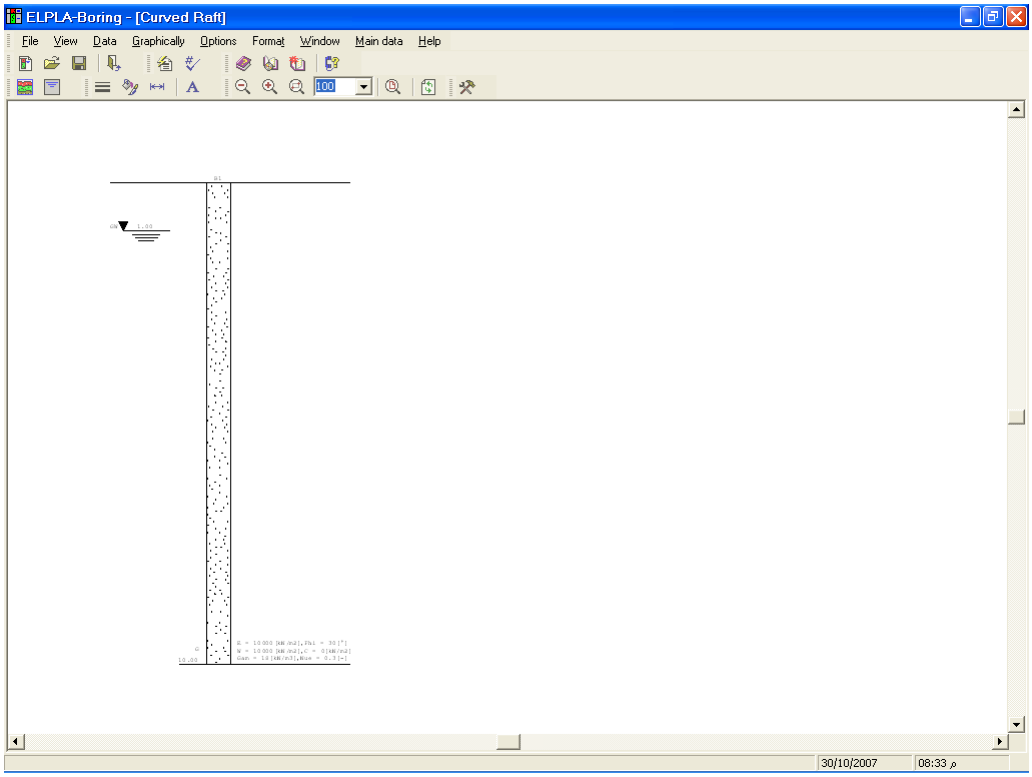


Figure 10-22 *ELPLA-Boring* sub program with a default-boring log

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

- Choose "Soil data" command from "Data" menu in the window of Figure 10-22. The following dialog box in Figure 10-23 with default-boring log data appears

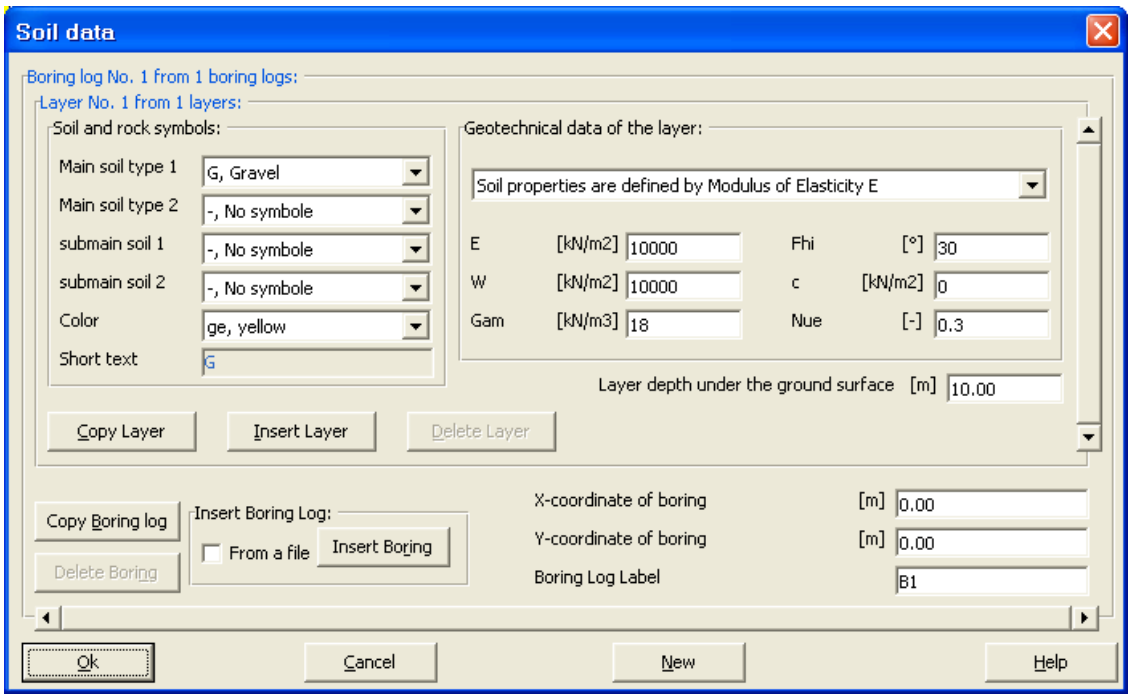


Figure 10-23 "Soil data" dialog box with default-boring log data

In the "Geotechnical data of the layer" dialog group box in Figure 10-23, define the geotechnical data of the first soil layer of the boring log as follows:

E_s	= 10000	[kN/m^2]
W_s	= 10000	[kN/m^2]
γ_s	= 18	[kN/m^3]
ν_s	= 0.3	[-]

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.

φ_s	= 30	[°]
c	= 0	[kN/m^2]

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

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

To define the soil type and color for the first layer, select "U, Silt" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box, Figure 10-23. The color of the silt according to the German Standard DIN 4023 will be automatically created. The user can change this color. Also, a short text "U" will be automatically created for the silt.

To enter the second layer

- Click "Layer copy" button in 0. A layer that has the same properties of the first layer will be copied
- Use the vertical scrollbar to move to the second soil layer. Layer No. will be typed automatically at the upper-left corner of the main dialog box of soil layers as a head title
- Change the value of the unit weight of the soil for the second soil layer from 18 [kN/m^3] to 9 [kN/m^3]
- Change the value of the layer depth under the ground surface from 2 [m] to [12 m]

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, the boring coordinates and labels which describe the boring will be entered.

To enter the boring coordinates and label

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

To enter the main soil data for all layers, choose "Main soil data" command from "Data" menu in Figure 10-22. The following dialog box in Figure 10-24 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 10-24, enter the settlement reduction factor α [-] and the groundwater depth under the ground surface G_w [m]. Then click "OK" button.

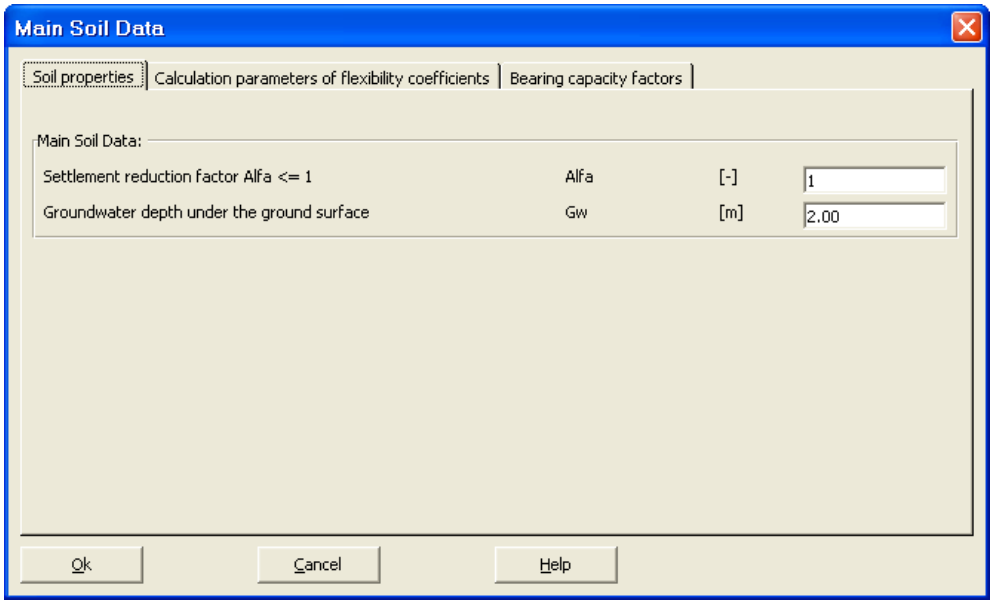


Figure 10-24 "Main soil data" dialog box

Now all data and parameters for the boring log have been entered. After finishing the creation of the boring log, click "OK" button in "Soil data" dialog box in Figure 10-23 to see the defined boring log on the screen where the user can control or modify the input data and parameters, Figure 10-25.

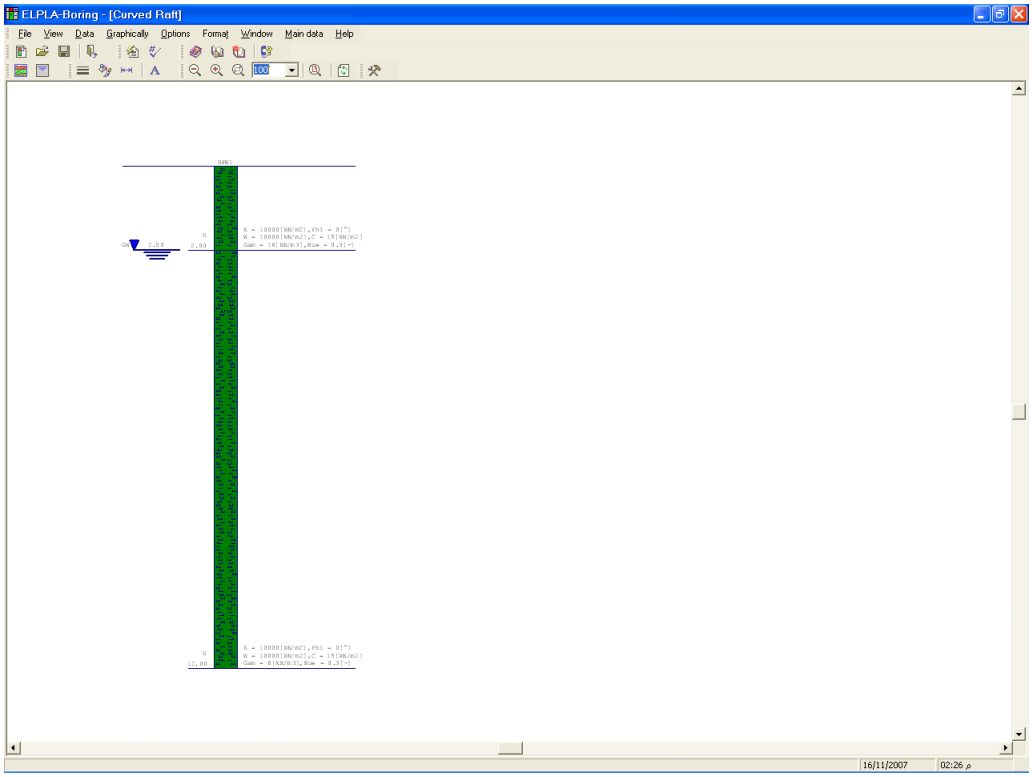


Figure 10-25 Boring log on the screen

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

- Choose "Save boring log" command from "File" menu in Figure 10-25 to save the data of the boring log
- Choose "Close boring logs" command from "File" menu in Figure 10-25 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*.

2.5 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 10-26 appears with default foundation properties. The data of foundation properties for the current example, which are required to define, are foundation material and foundation thickness. Any other data corresponding to foundation properties in the program menus are not required. Therefore, the user can take these data from the default foundation properties.

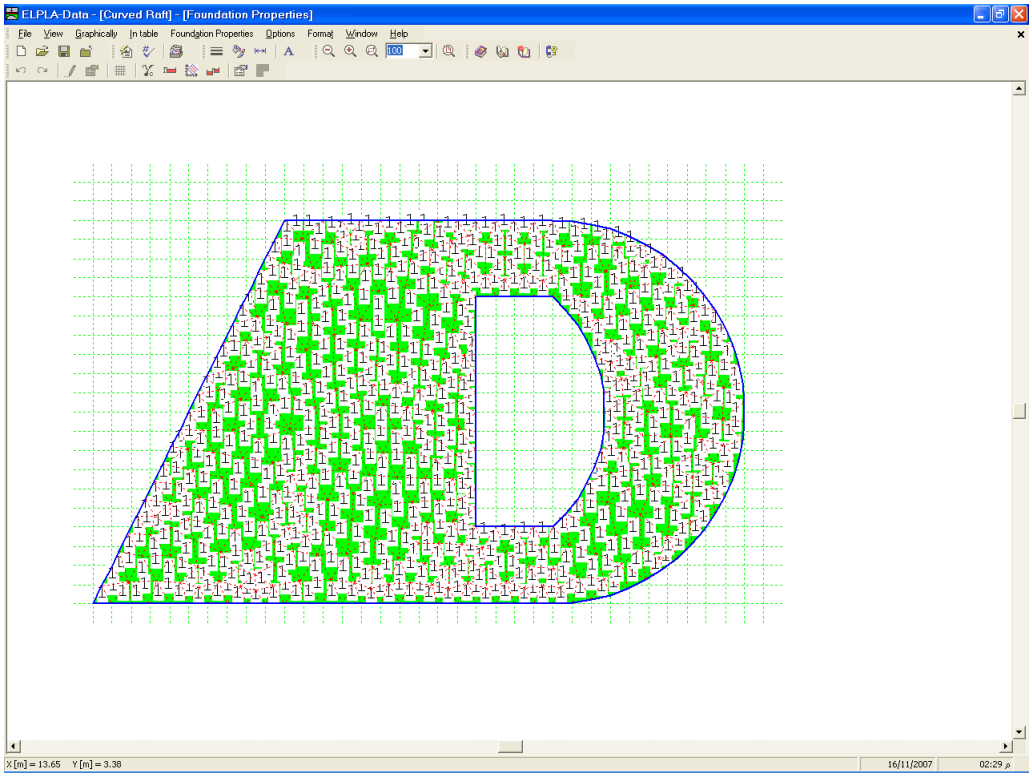


Figure 10-26 "Foundation properties" embedded program

To enter the foundation material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 10-26. The following list box in Figure 10-27 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 10-27, enter E-Modulus of the foundation, *Poisson's* ratio of the foundation and foundation thickness
- Click "OK" button

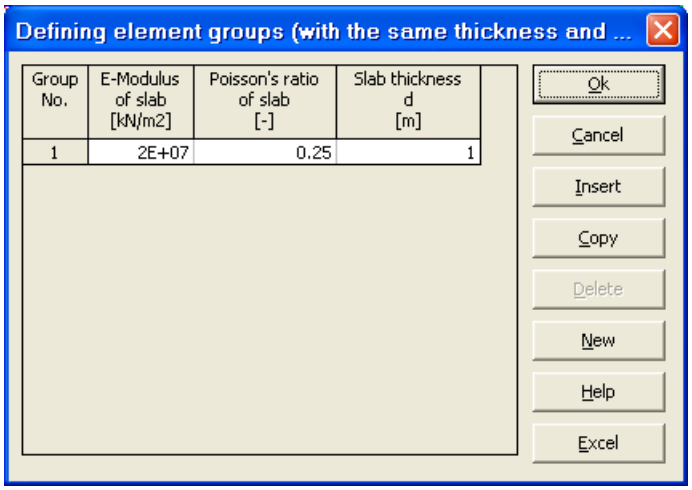


Figure 10-27 "Defining element groups" list box

To enter the unit weight of the foundation

- Choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 10-26. The following dialog box in Figure 10-28 with a default unit weight of 25 [kN/m^3] appears. Let the default value as written in the edit box "Unit weight of the foundation"
- Click "OK" button

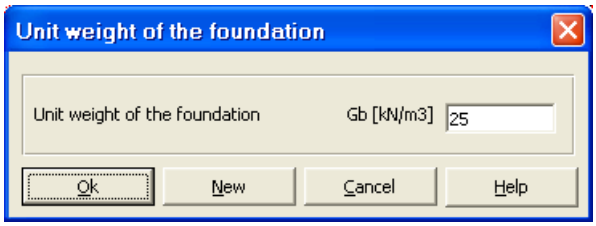


Figure 10-28 "Unit weight of the foundation" dialog box

To enter the foundation level

- Choose "Foundation depth" command from "Foundation properties" menu in the window of Figure 10-26. The following dialog box in Figure 10-29 appears
- In this dialog box type 2 in the "Foundation depth under ground surface (a)/ (b)" edit box
- Click "OK" button

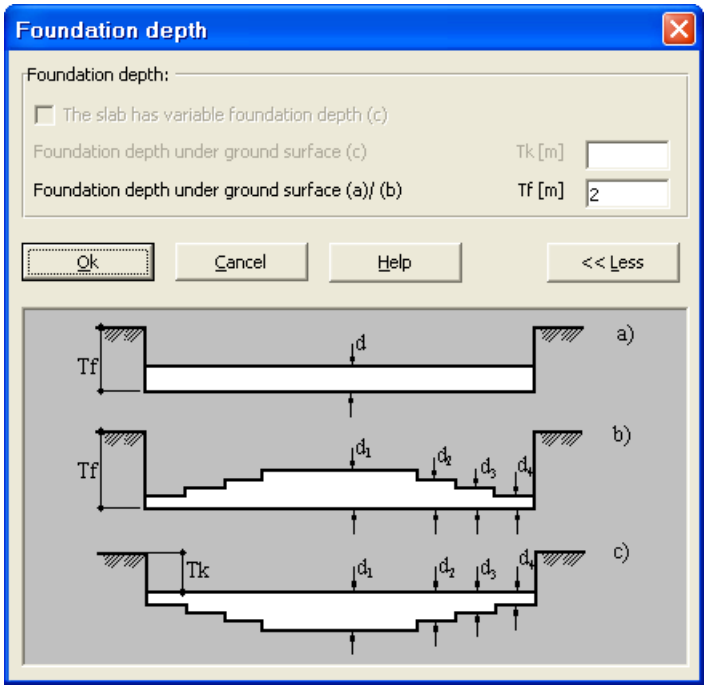


Figure 10-29 "Foundation depth" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 10-26 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 10-26 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*.

2.6 Loads

To define the loads, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 10-30 appears.

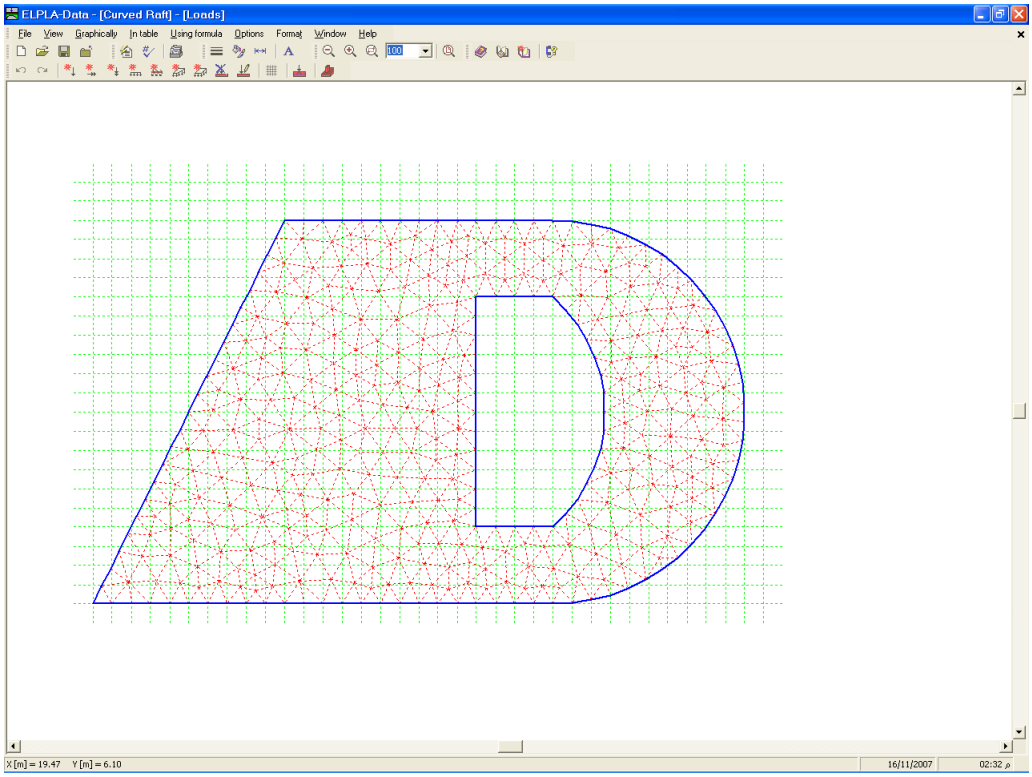
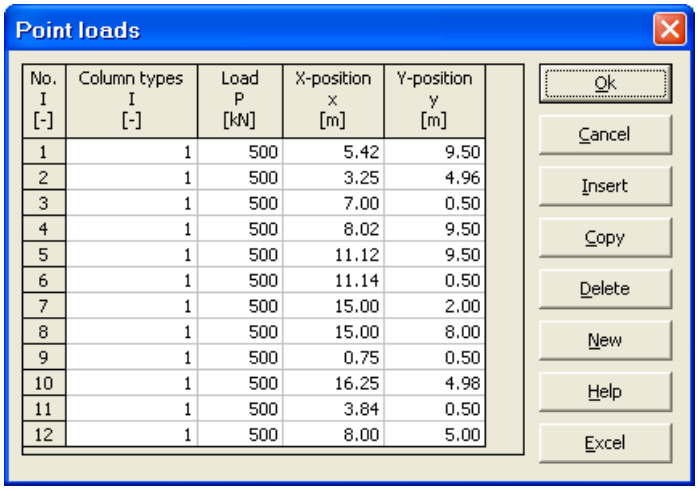


Figure 10-30 "Loads" embedded program

To enter loads

- Choose "Point loads" command from "In Table" menu in the window of Figure 10-30. The following list box in Figure 10-31 appears
- Enter the external point loads *P* [kN] and their corresponding coordinates (*x*, *y*) in the list box of Figure 10-31. 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 foundation (local coordinates)
- Click "OK" button



No. I [-]	Column types I [-]	Load P [kN]	X-position x [m]	Y-position y [m]
1	1	500	5.42	9.50
2	1	500	3.25	4.96
3	1	500	7.00	0.50
4	1	500	8.02	9.50
5	1	500	11.12	9.50
6	1	500	11.14	0.50
7	1	500	15.00	2.00
8	1	500	15.00	8.00
9	1	500	0.75	0.50
10	1	500	16.25	4.98
11	1	500	3.84	0.50
12	1	500	8.00	5.00

Figure 10-31 "Point loads *P*" list box

After you have completed the definition of all load data, the screen should look like the following Figure 10-32.

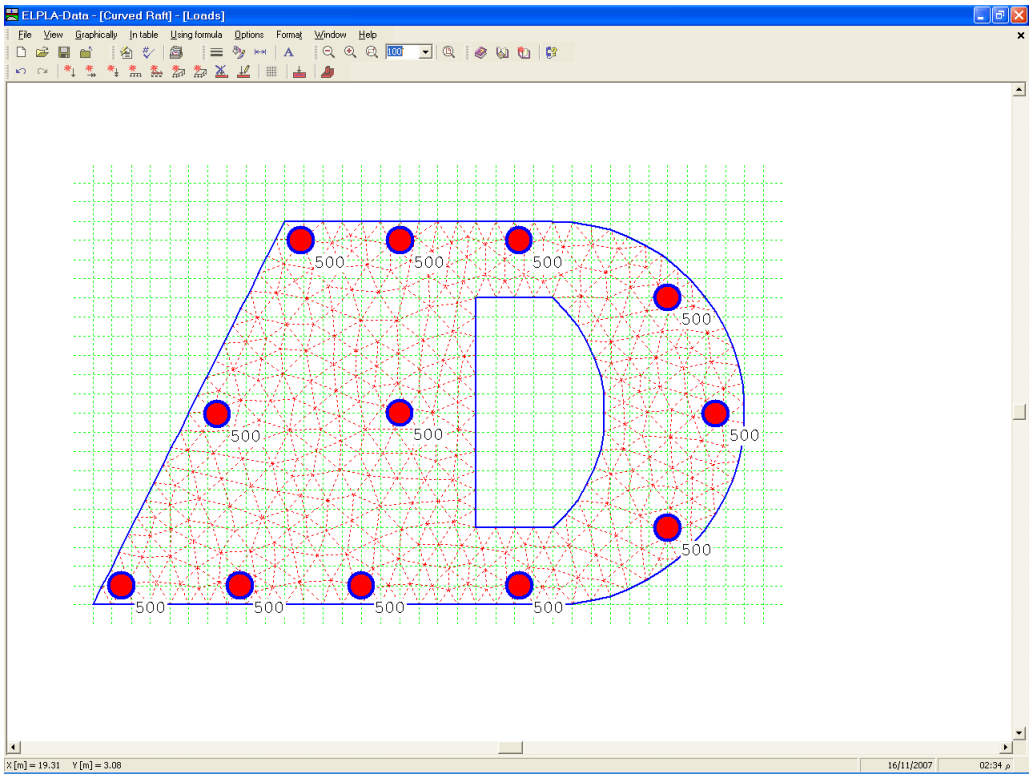


Figure 10-32 Loads on the screen

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

- Choose "Save loads" command from "File" menu in Figure 10-32 to save the load data
- Choose "Close loads" command from "File" menu in Figure 10-32 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 raft is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

3 Carrying out the calculations

3.1 Starting *ELPLA-Solver*

To analyze the problem leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window appears, Figure 10-33.

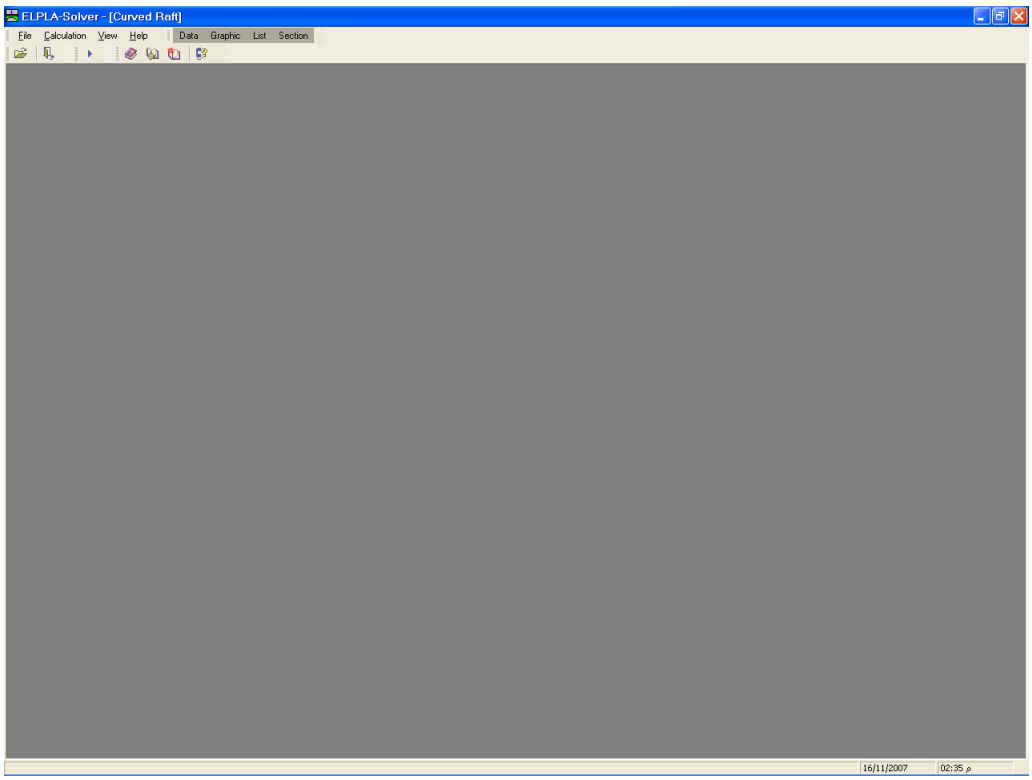


Figure 10-33 Opening screen of the sub program *ELPLA-Solver*

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
- Iteration process
- 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 "Iteration parameters" option box will appear (Figure 10-34)
- Choose the suitable iteration option
- Press the "OK" button

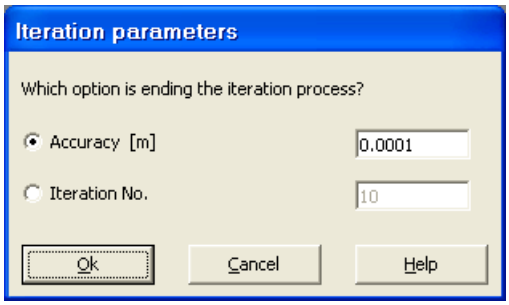


Figure 10-34 "Iteration parameters" option box

Analysis progress

Analysis progress menu in Figure 10-35 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.

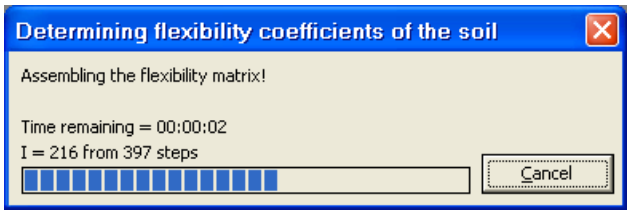


Figure 10-35 Analysis progress menu

Iteration process

Information about the convergence progress of the computations is displayed in the "Iteration process" list box in Figure 10-36 during the iteration process.

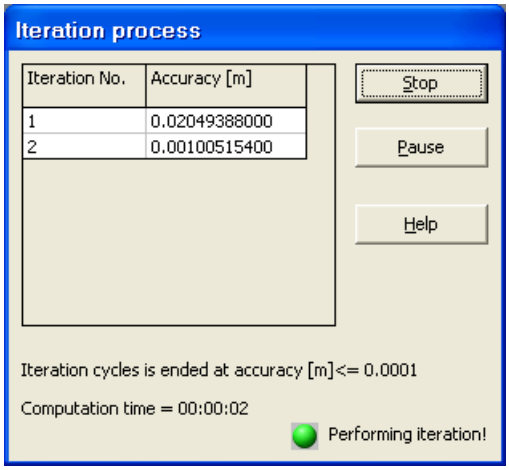


Figure 10-36 "Iteration process" list box

Check of the solution

Once the analysis is complete, a check menu of the solution appears, Figure 10-37. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

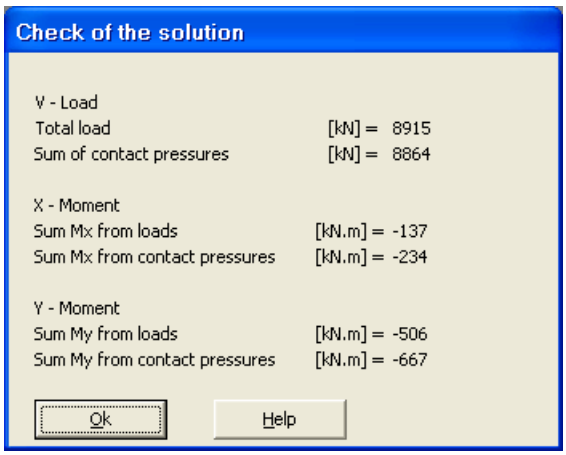


Figure 10-37 Menu "Check of the solution"

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 10-38. *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.

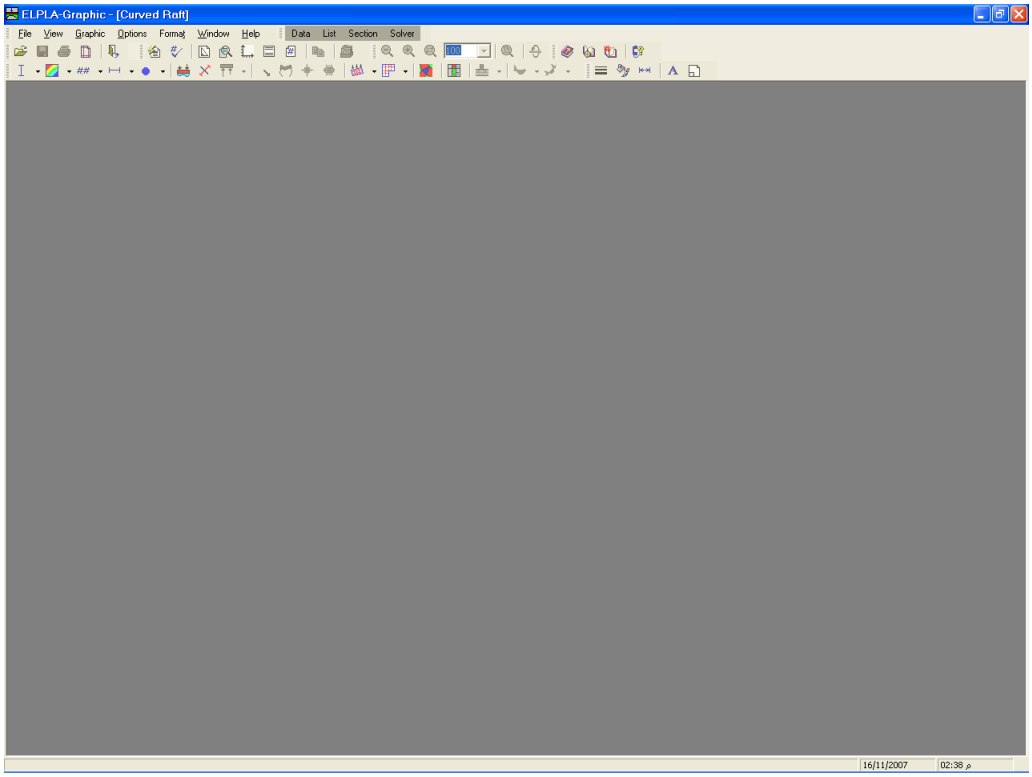


Figure 10-38 Opening screen of the sub program *ELPLA-Graphic*

To view the results for the raft as contour lines

- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 10-39 appears
- In "Results as contour lines" option box, select "Settlements *s*" as a sample for the results to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for the raft as shown in Figure 10-40.

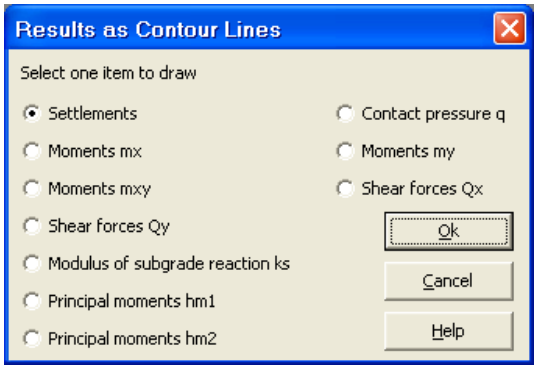


Figure 10-39 "Results as contour lines" option box

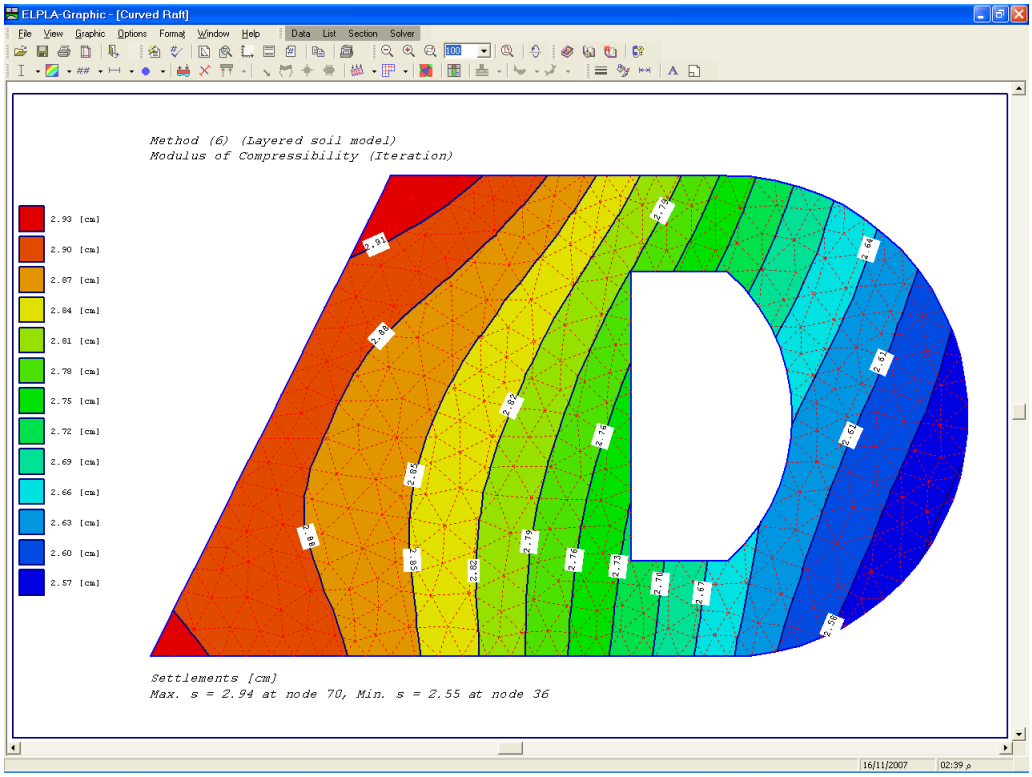


Figure 10-40 Settlements as contour lines for the raft

4.2 Plot a diagram of the results

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

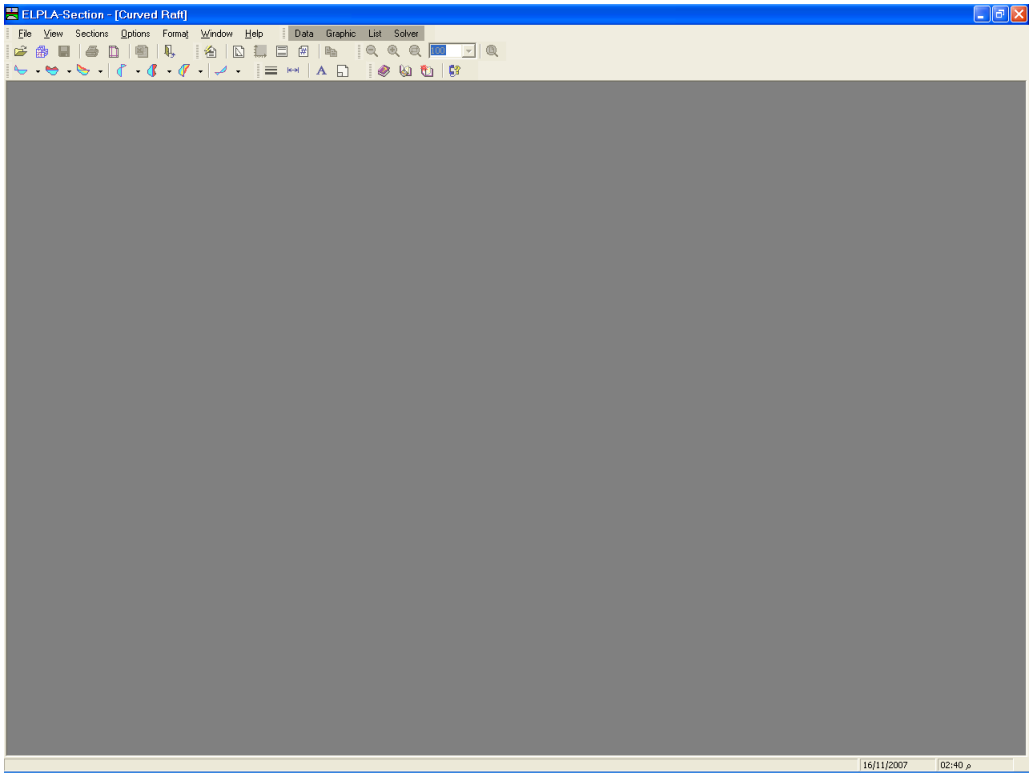


Figure 10-41 Opening screen of the sub program *ELPLA-Section*

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

To plot a section in *x*-direction

- Choose "Section in *x*-direction" command from "Sections" menu. The following option box in Figure 10-42 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

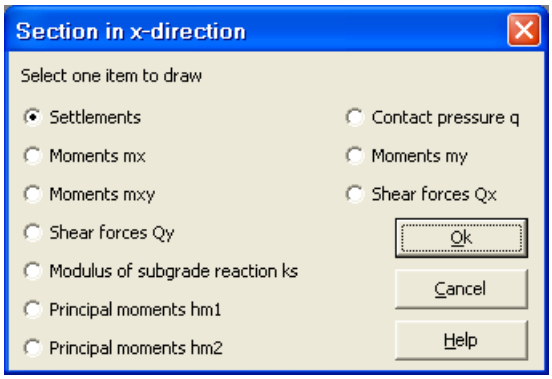


Figure 10-42 "Section in *x*-direction" option box

The following option box in Figure 10-43 appears to specify the section in *x*-direction that is required to be plotted.

In this dialog box

- Type 5.00 in the "Section at *y*-coordinate" edit box to plot a diagram at the middle of the raft
- Click "OK" button

The settlements are now plotted in a diagram at the middle of the raft as shown in Figure 10-44.

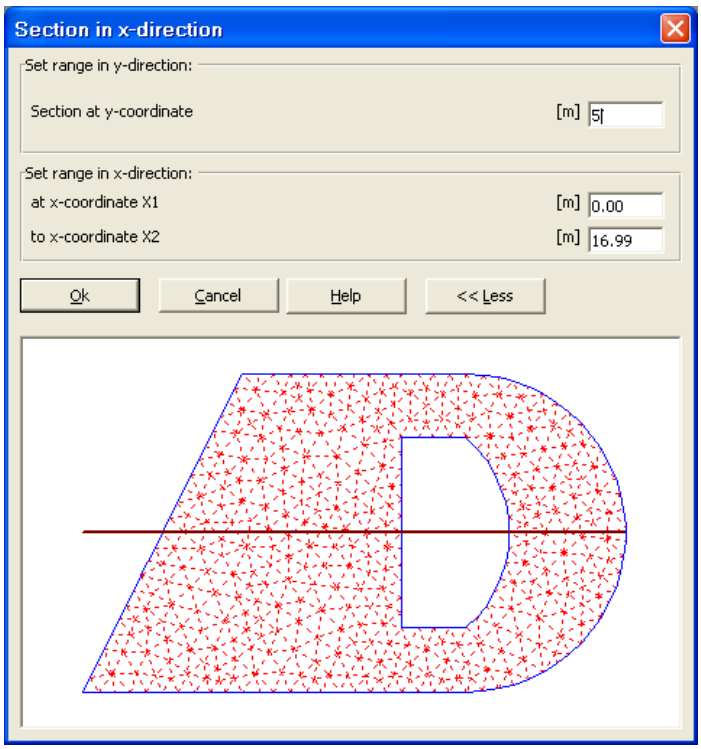


Figure 10-43 "Section in *x*-direction" dialog box

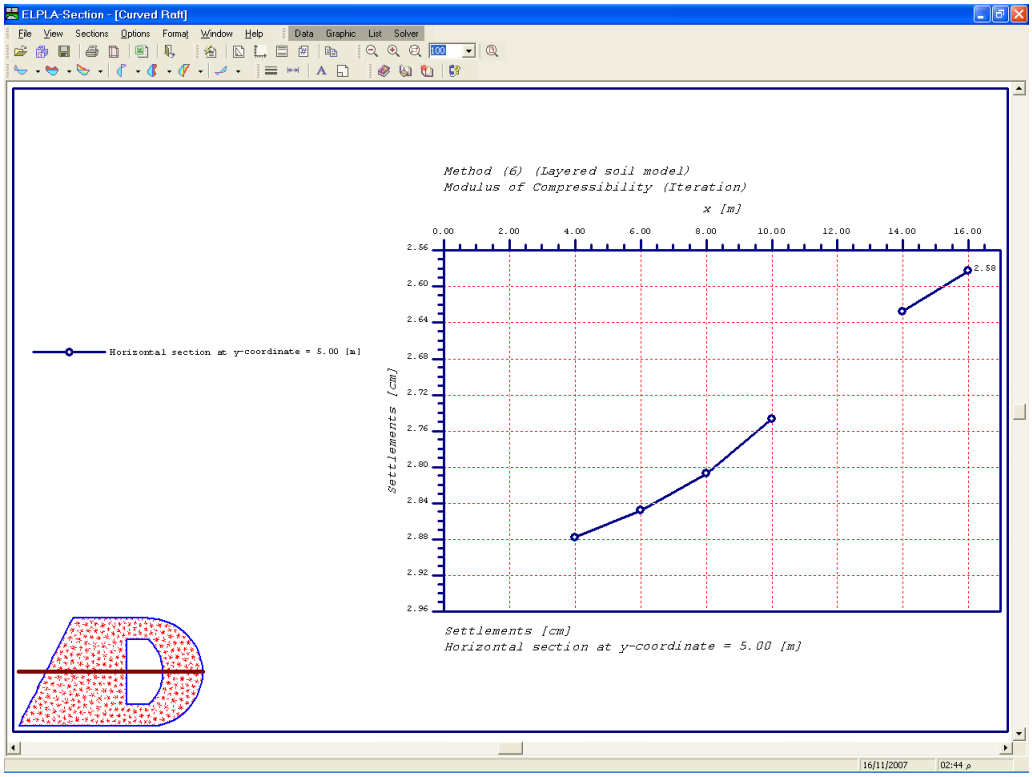


Figure 10-44 Diagram of settlements in *x*-direction at the middle of the raft

4.3 Listing data and result in tables

To list tables of data and results, switch to *ELPLA-List*. This is done by clicking on "List" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Section* window. *ELPLA-List* window appears, Figure 10-45.

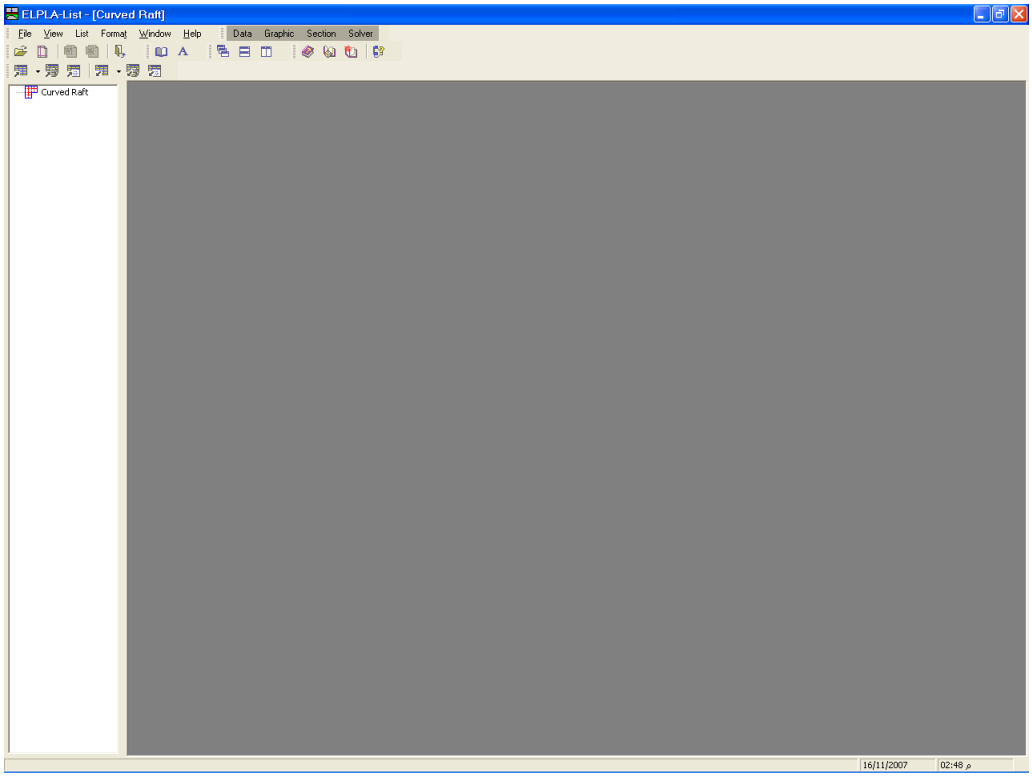


Figure 10-45 Opening screen of the sub program *ELPLA-List*

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 10-46 appears
- Select "Settlements" as an example for the results to be listed in a table
- Click "OK" button. The settlement results are now listed on the screen (Figure 10-47)
- Choose "Send to Excel" from "File" menu if you wish to export the table to the MS Excel application, Figure 10-48

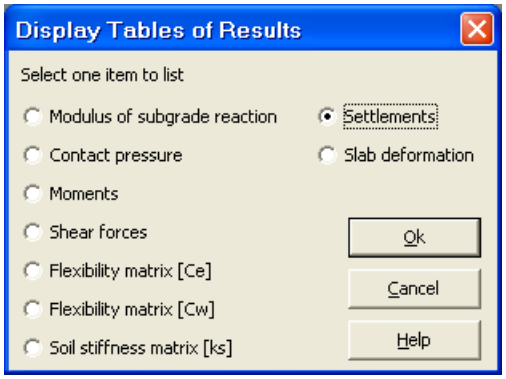


Figure 10-46 "Display tables of results" option box

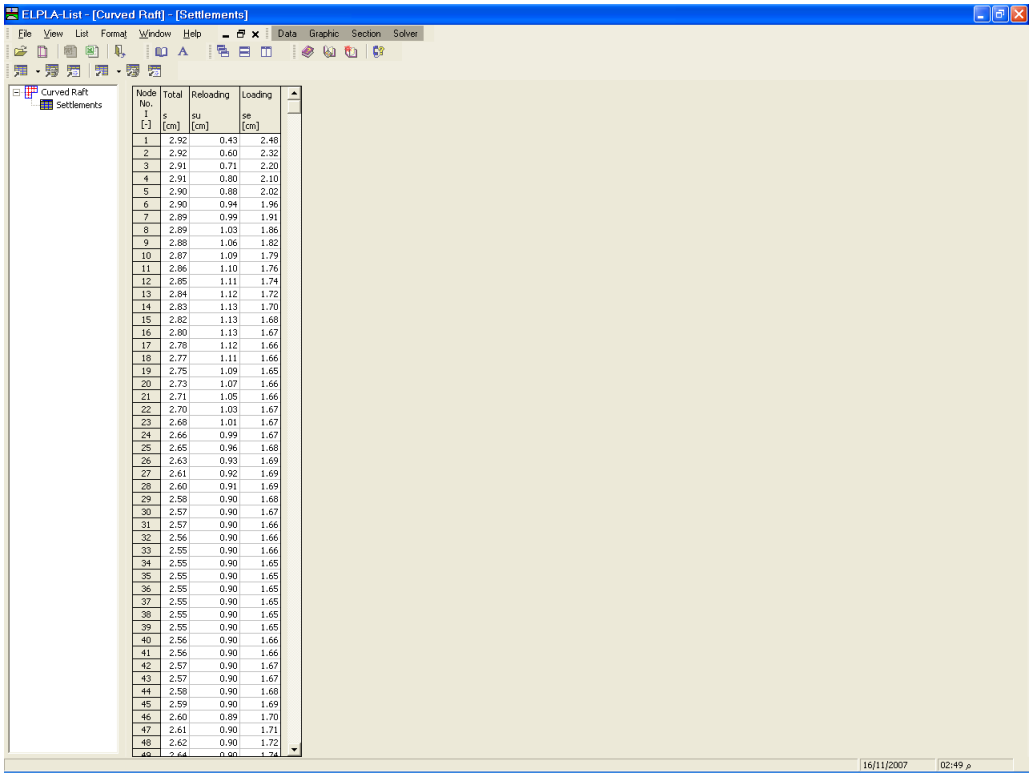
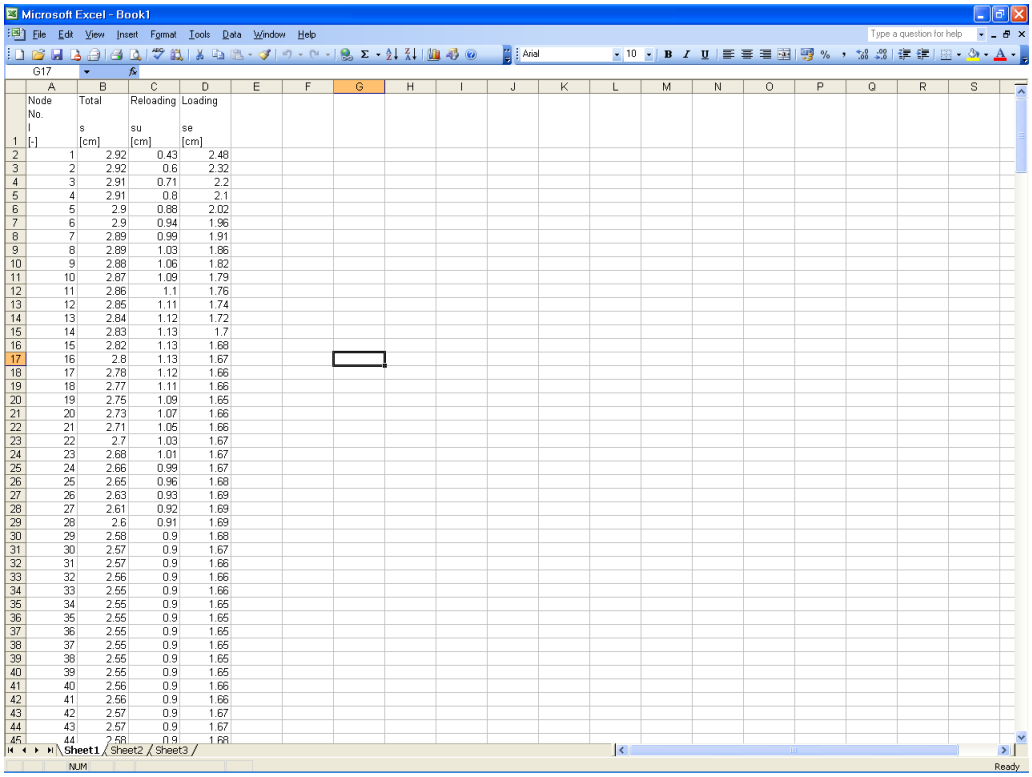


Figure 10-47 List of settlement results



	A	B	C	D
1	Node No. I [-]	Total s [cm]	Reloading su [cm]	Loading se [cm]
2	1	2.92	0.43	2.48
3	2	2.92	0.6	2.32
4	3	2.91	0.71	2.2
5	4	2.91	0.8	2.1
6	5	2.9	0.88	2.02
7	6	2.9	0.94	1.96
8	7	2.89	0.99	1.91
9	8	2.89	1.03	1.86
10	9	2.88	1.06	1.82
11	10	2.87	1.09	1.79
12	11	2.86	1.1	1.76
13	12	2.85	1.11	1.74
14	13	2.84	1.12	1.72
15	14	2.83	1.13	1.7
16	15	2.82	1.13	1.68
17	16	2.8	1.13	1.67
18	17	2.78	1.12	1.66
19	18	2.77	1.11	1.66
20	19	2.75	1.09	1.65
21	20	2.73	1.07	1.66
22	21	2.71	1.05	1.66
23	22	2.7	1.03	1.67
24	23	2.68	1.01	1.67
25	24	2.66	0.99	1.67
26	25	2.65	0.96	1.68
27	26	2.63	0.93	1.69
28	27	2.61	0.92	1.69
29	28	2.6	0.91	1.69
30	29	2.58	0.9	1.68
31	30	2.57	0.9	1.67
32	31	2.57	0.9	1.66
33	32	2.56	0.9	1.66
34	33	2.55	0.9	1.66
35	34	2.55	0.9	1.65
36	35	2.55	0.9	1.65
37	36	2.55	0.9	1.65
38	37	2.55	0.9	1.65
39	38	2.55	0.9	1.65
40	39	2.55	0.9	1.65
41	40	2.56	0.9	1.66
42	41	2.56	0.9	1.66
43	42	2.57	0.9	1.67
44	43	2.57	0.9	1.67
45	44	2.58	0.9	1.68

Figure 10-48 Exported results in MS Excel

5 Index

S

T

U

Y

Z