

Example 1

**Analysis of an irregular shaped
raft on irregular subsoil**

Contents	Page
1 Description of the problem	3
1.1 Loads.....	3
1.2 Raft material and thickness	3
1.3 Soil properties	4
1.4 Method of analysis	4
2 Creating the project	7
2.1 Calculation method	7
2.2 Project identification	12
2.3 FE-Net data	12
2.4 Soil Properties.....	20
2.5 Foundation properties.....	29
2.6 Boring fields.....	32
2.7 Loads.....	33
3 Carrying out the calculations	36
4 Viewing data and results	40
4.1 Viewing data and result graphics	40
4.2 Plot a diagram of the results at a specified section	42
4.3 Print the drawing	44
4.4 Listing data and results in tables	45
5 Index.....	49

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/m²], 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.

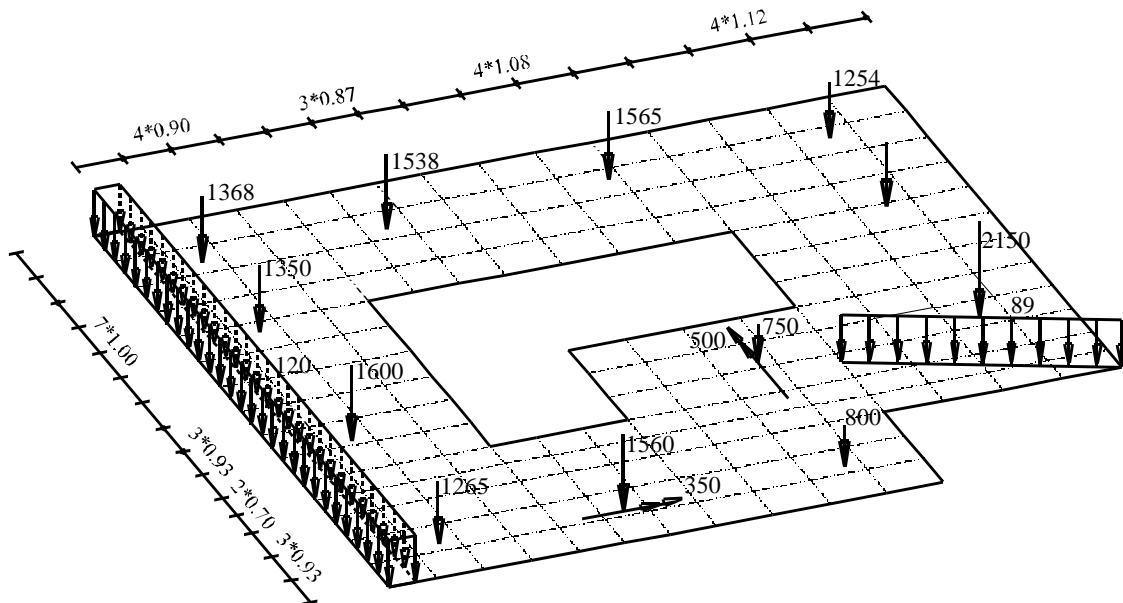


Figure 1.1 Raft dimensions [m] and loads

1.2 Raft material and thickness

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

Young's modulus	E_b	$= 2 \times 10^7$	[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 $\gamma_b = 0.0$ [kN/m³] 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 $E_{s1} = 9500$ [kN/m²], $E_{s2} = 22000$ [kN/m²] and $E_{s3} = 120000$ [kN/m²] while those for reloading are $W_{s1} = 26000$ [kN/m²], $W_{s2} = 52000$ [kN/m²] and $W_{s3} = 220000$ [kN/m²]. *Poisson's* ratio is 0.3 [-] for all soil layers. Unit weight of the soil above the ground water is $\gamma_s = 19$ [kN/m³], while that under the ground water is $\gamma'_s = 9$ [kN/m³]. 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*.

Example 1

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 M_x

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

Table 1.3 Moment M_y

Moment No. [-]	Moment value M_y [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 P1 [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/m ²]	Load start x1 [m]	Load start y1 [m]	Load end x2 [m]	Load end y2 [m]
1	120	0	0	0.5	13.97

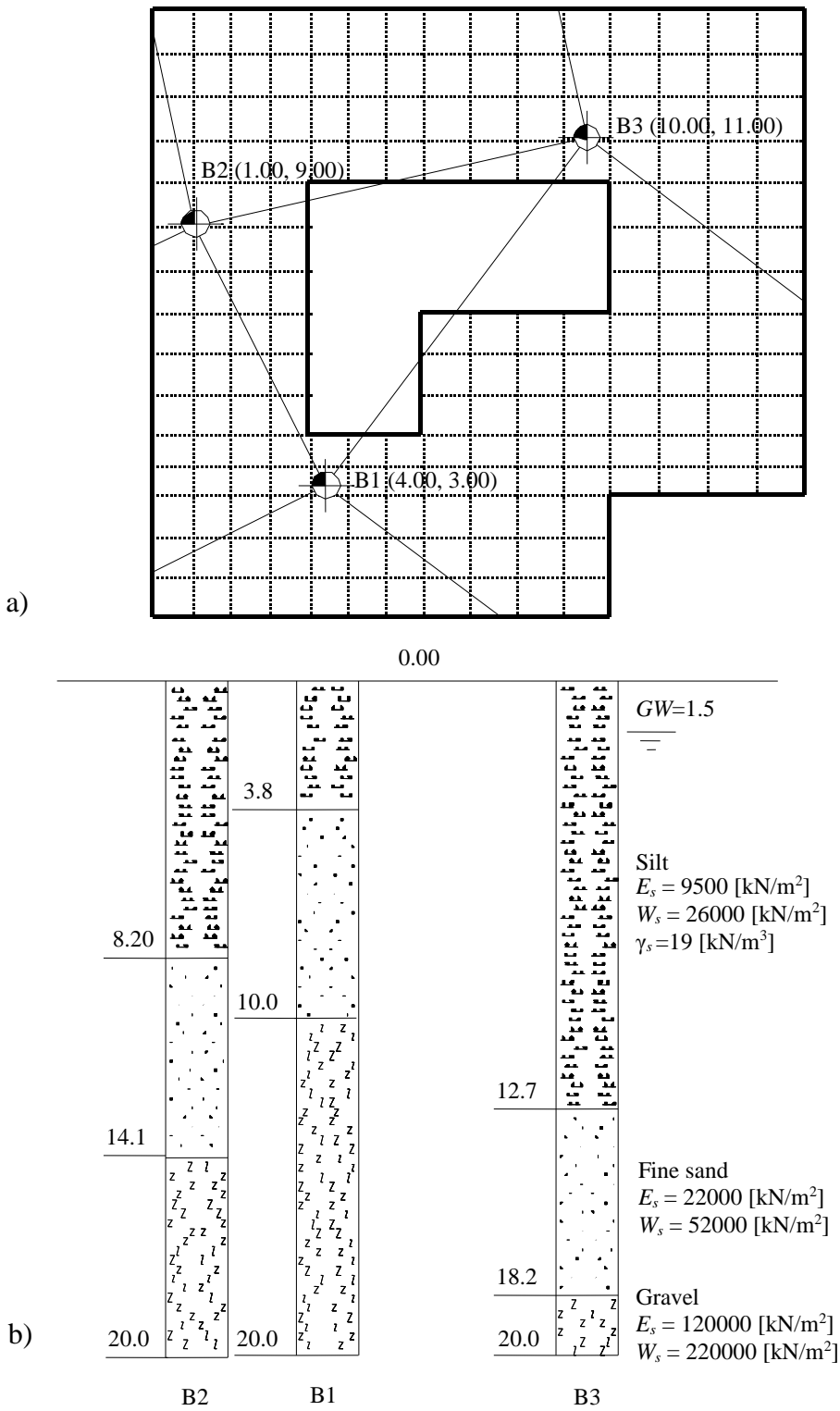


Figure 1.2 a) Boring locations and interpolation zones
b) Boring logs B1 to B3

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 gradually 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. 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. 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

Calculation Method

Calculation Method:

- ☐ 1- Linear Contact Pressure (Conventional Method)
- ☐ 2/3- Constant/ Variable Modulus of Subgrade Reaction
- ☐ 4- Modification of Modulus of Subgrade Reaction by Iteration
- ☐ 5- Isotropic Elastic Half Space
- ☒ 6- Modulus of Compressibility (Iteration)
- ☐ 7- Modulus of Compressibility (Elimination)
- ☐ 8- Modulus of Compressibility for Rigid Raft
- ☐ 9- Flexible Foundation

Subsoil model:

- ☐ Half Space model
- ☒ Layered soil model

Help Load... Save As... Cancel < Back Next > Save

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.

Example 1

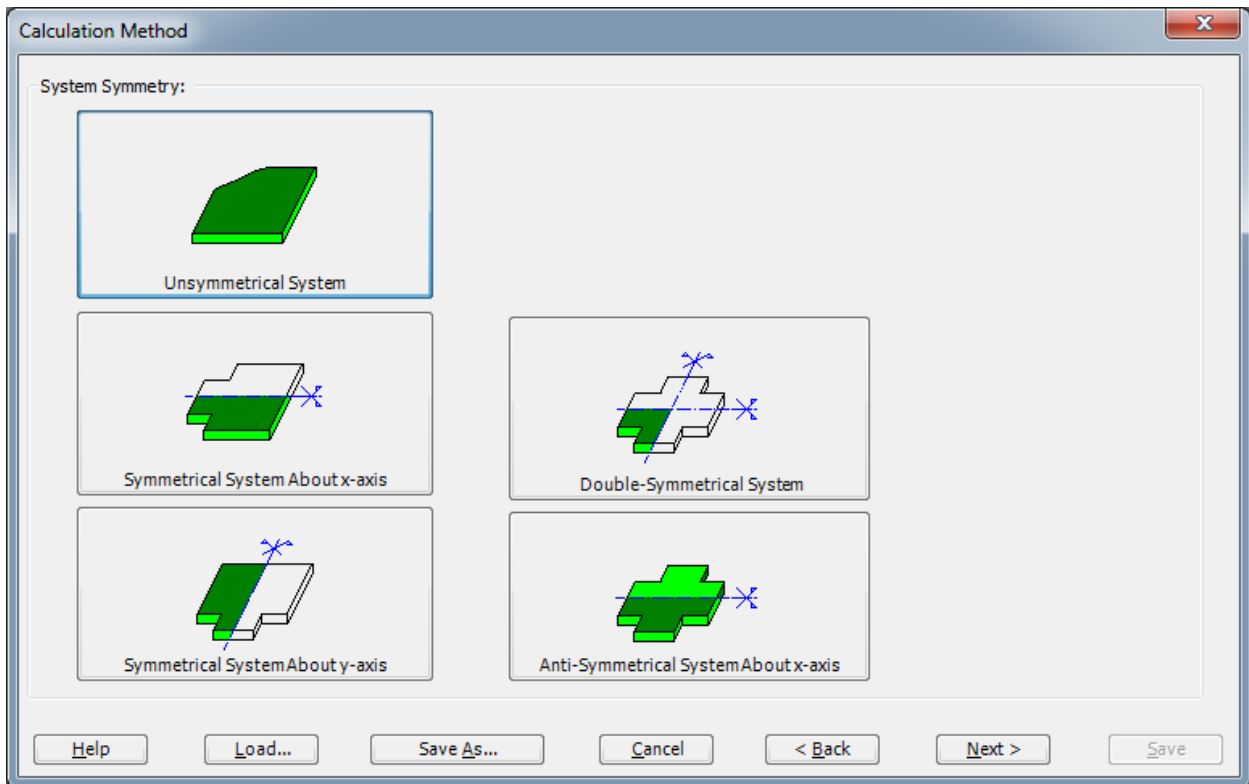


Figure 1.5 "System Symmetry" Form

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.

Calculation Method

Options:

- ☐ Slab With Girders
- ☐ Additional Springs
- ☐ Supports/ Boundary Conditions
- ☐ Determining Limit Depth
- ☐ Concrete Design
- ☐ Nonlinear Subsoil Model
- ☐ Determining Displacements in Soil
- ☐ Determining Stresses in Soil
- ☐ Determining Strains in Soil
- ☐ Influence of Neighboring Foundations on Raft
- ☐ Influence of Temperature Change on the Raft
- ☐ Influence of Additional Settlements on the Raft

☐ Select All

Nonlinear analysis of piled raft:

- ☒ Nonlinear analysis using a hyperbolic function for load-settlement
- ☐ Nonlinear analysis using German standard DIN 4014 for load-settlement
- ☐ Nonlinear analysis using German recommendations EA-Piles for load-settlement
- ☐ Nonlinear analysis using a given load-settlement curve

Help Load... Save As... Cancel < Back Next > Save

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 "Irregular raft". *ELPLA* will use automatically this file name in all reading and writing processes.

Example 1

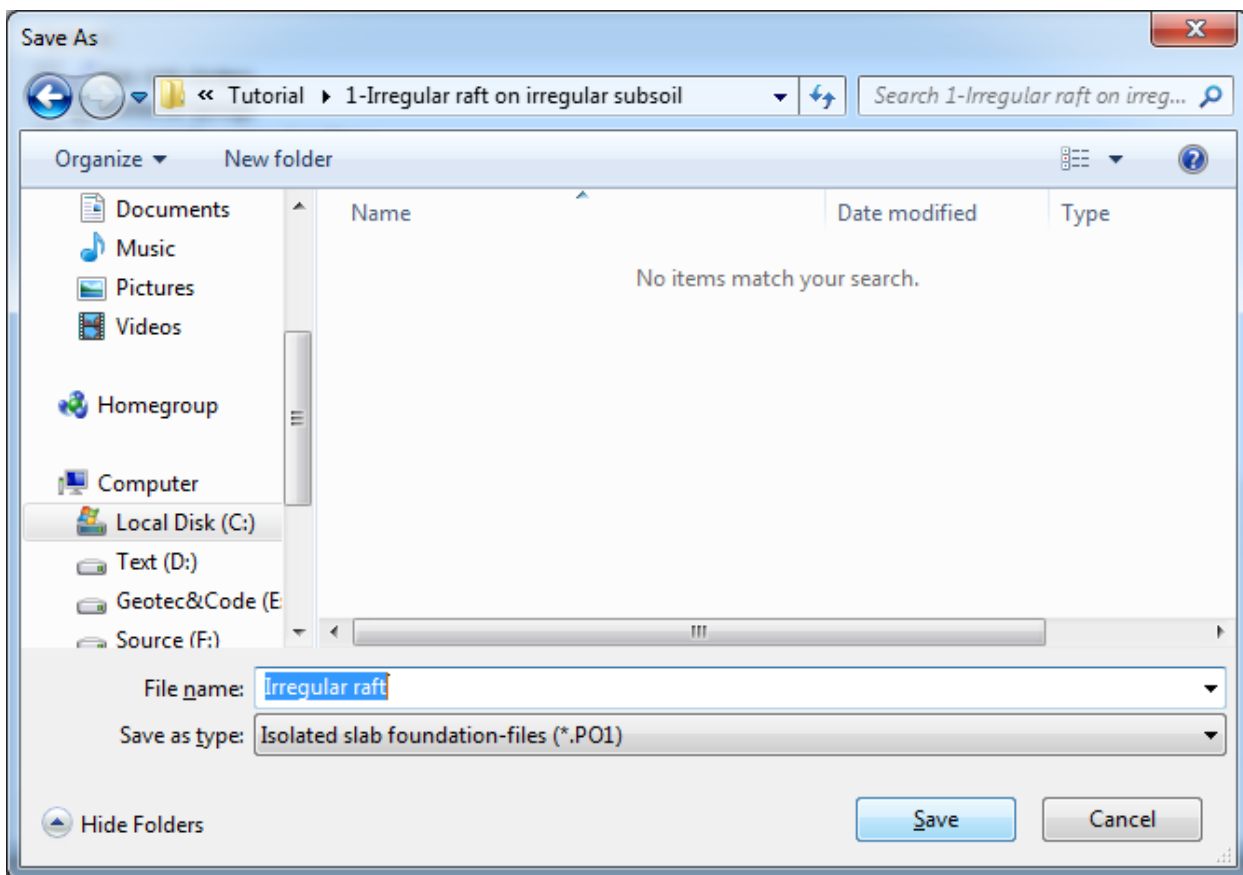


Figure 1.7 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Irregular raft] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

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 the "Data" Tab. The dialog box in Figure 1.8 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 " Irregular raft " in the "Project" edit box
- Click "Save" button

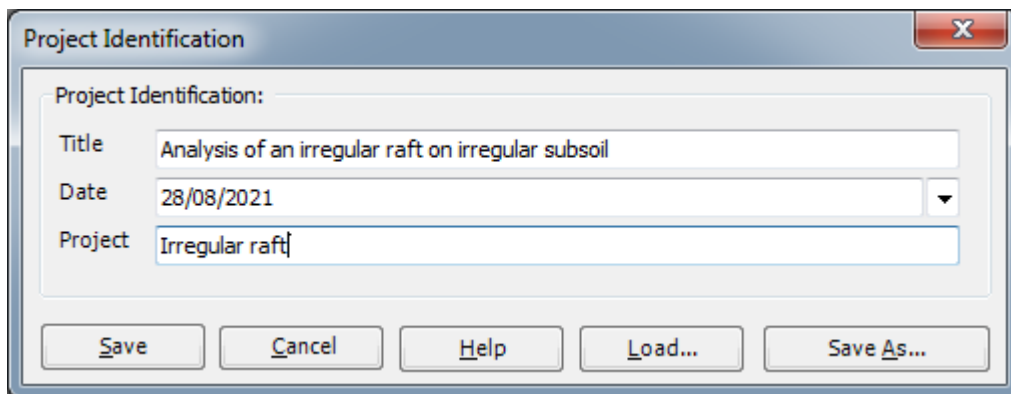


Figure 1.8 "Project Identification" dialog box

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" Tab. "FE-Net Generation" wizard appears as shown in Figure 1.9. This wizard will guide you through the steps required to generate a FE-Net. As shown in Figure 1.9, 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.

Example 1

The image shows a software window titled "FE-Net Generation". Inside, there is a section labeled "Slab Type:" with five icons representing different slab geometries: an irregular polygon, a square, a rectangle (which is highlighted with a blue border), a circle, and an annulus. Below this is a section labeled "Rectangular Slab:" containing two input fields. The first is labeled "Length of Rectangular Slab" with a value of "15.01" and a unit "[m]". The second is labeled "Width of Rectangular Slab" with a value of "13.98" and a unit "[m]". At the bottom of the window are five buttons: "Help", "Cancel", "< Back", "Next >", and "Finish".

Figure 1.9 "FE-Net Generation" wizard with "Slab Type" Form

To generate the FE-Net

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

After clicking "Next" in "FE-Net Generation" wizard, the following "Generation Type" Form appears, Figure 1.10. *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.

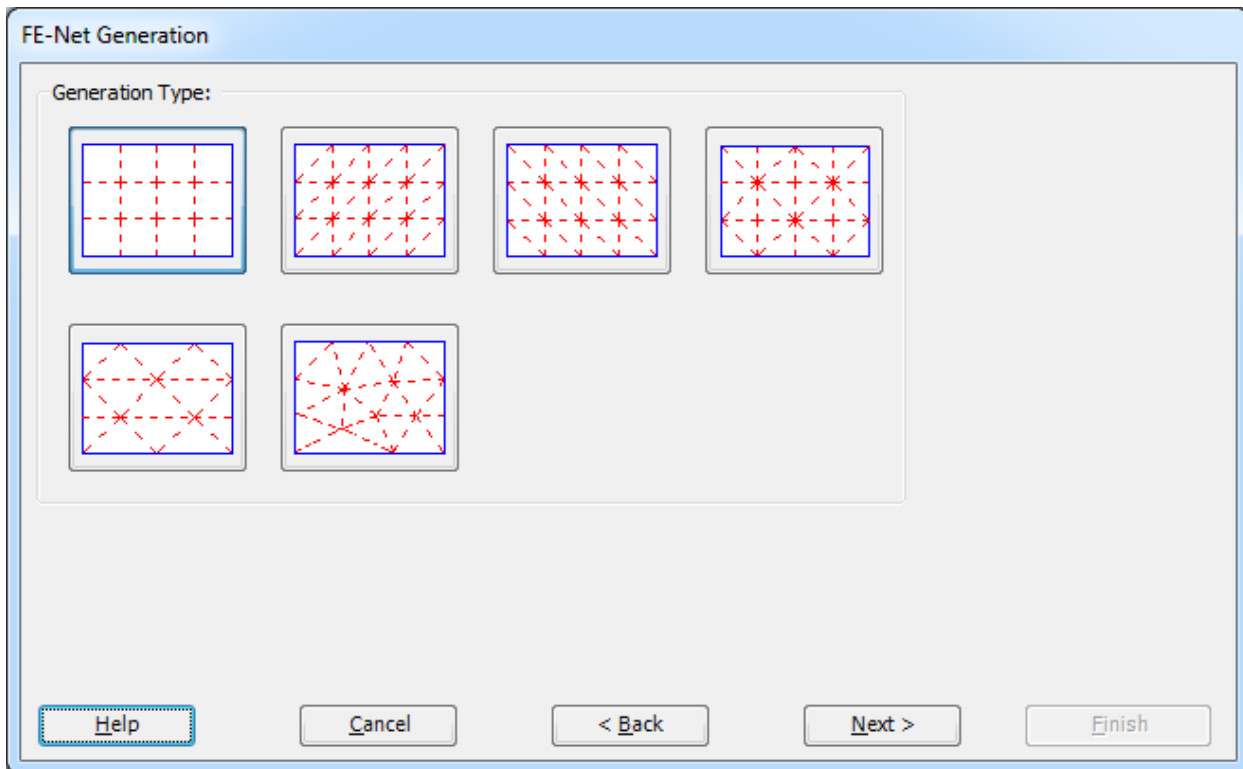


Figure 1.10 "Generation Type" Form

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 1.11. To edit the variable grid spacing in x -direction, do the following steps in "Grid in x -direction" frame:

- Uncheck the "Constant grid interval" check box. "Grid Intervals" button will be activated
- Click the button "Grid Intervals". "Grid intervals in x -direction" list box appears, Figure 1.12. 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 intervals in y -direction, repeat the previous steps. After that, click "Finish" button in "FE-Net Generation" wizard to see the FE-Net.

Example 1

FE-Net Generation

Grid Definition:

Grids in x-direction:

☐ Constant grid interval

No. of grid intervals

10

Grid Intervals

1.00

Grids in y-direction:

☐ Constant grid interval

No. of grid intervals

10

Grid Intervals

1.00

Help

Cancel

< Back

Next >

Finish

Figure 1.11 "Grid Definition" Form

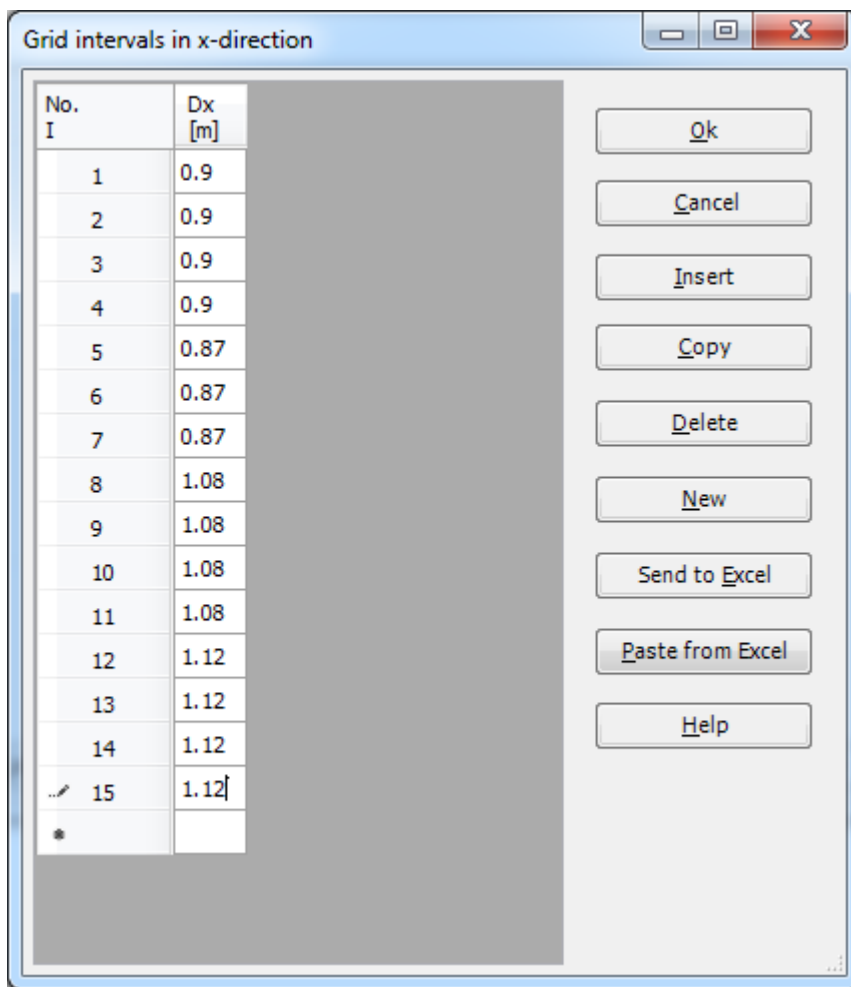


Figure 1.12 "Grid intervals in x -direction" list box

ELPLA will generate an imaginary net for a rectangular area of 15.01[m] length and 13.98 [m] width with different elements for each side. The following Window in Figure 1.13 appears with the generated imaginary net.

Example 1

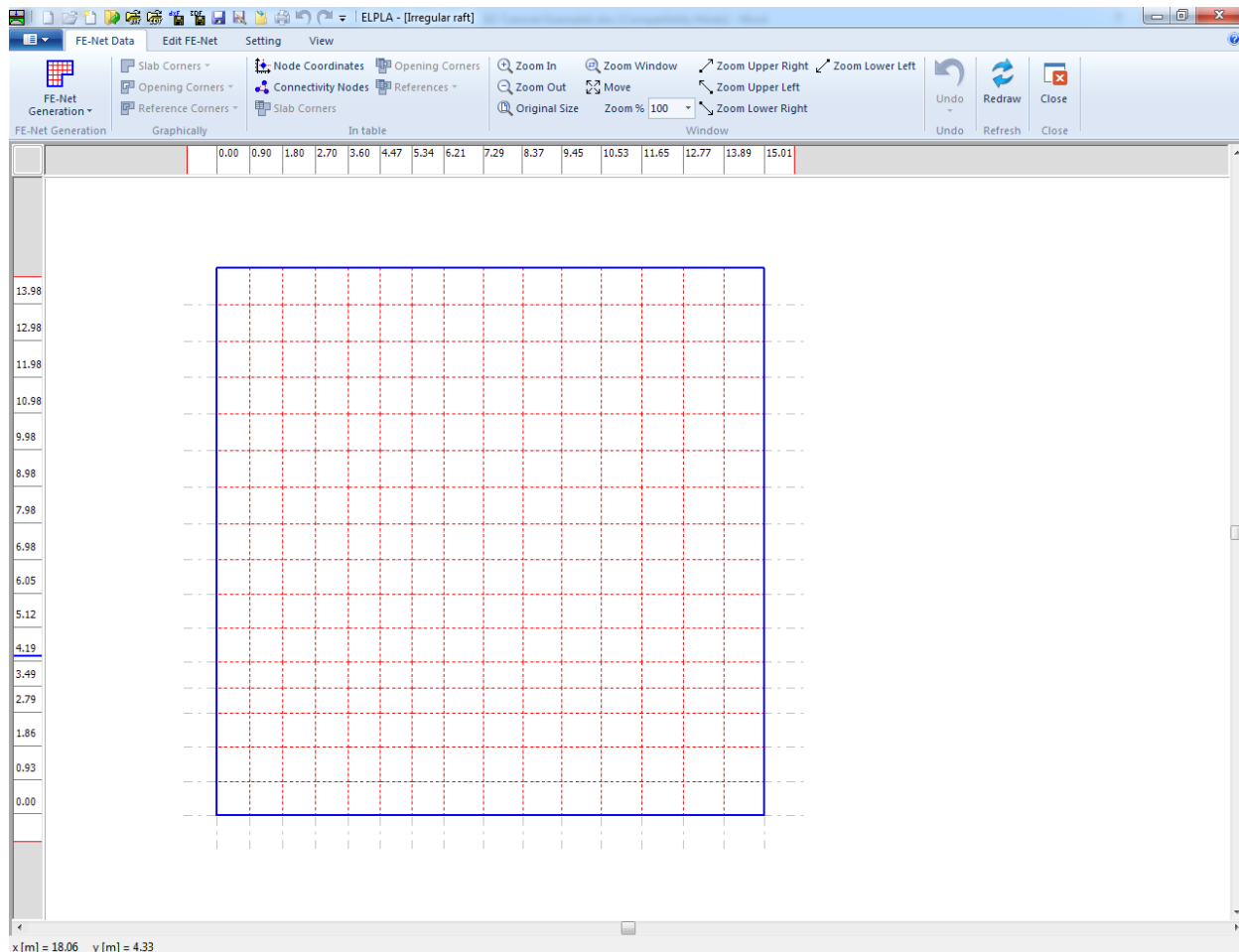


Figure 1.13 Generated FE-Net

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 "FE-Net Data" Tab (Double click anywhere will also activate the select nodes mode). When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair. If any node is selected, the command "Delete" in "Edit FE-Net" menu 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 "Delete" command from "Edit FE-Net" menu. The action of this command is indicated in Figure 1.15. To leave the graphic mode, press "Esc" key.

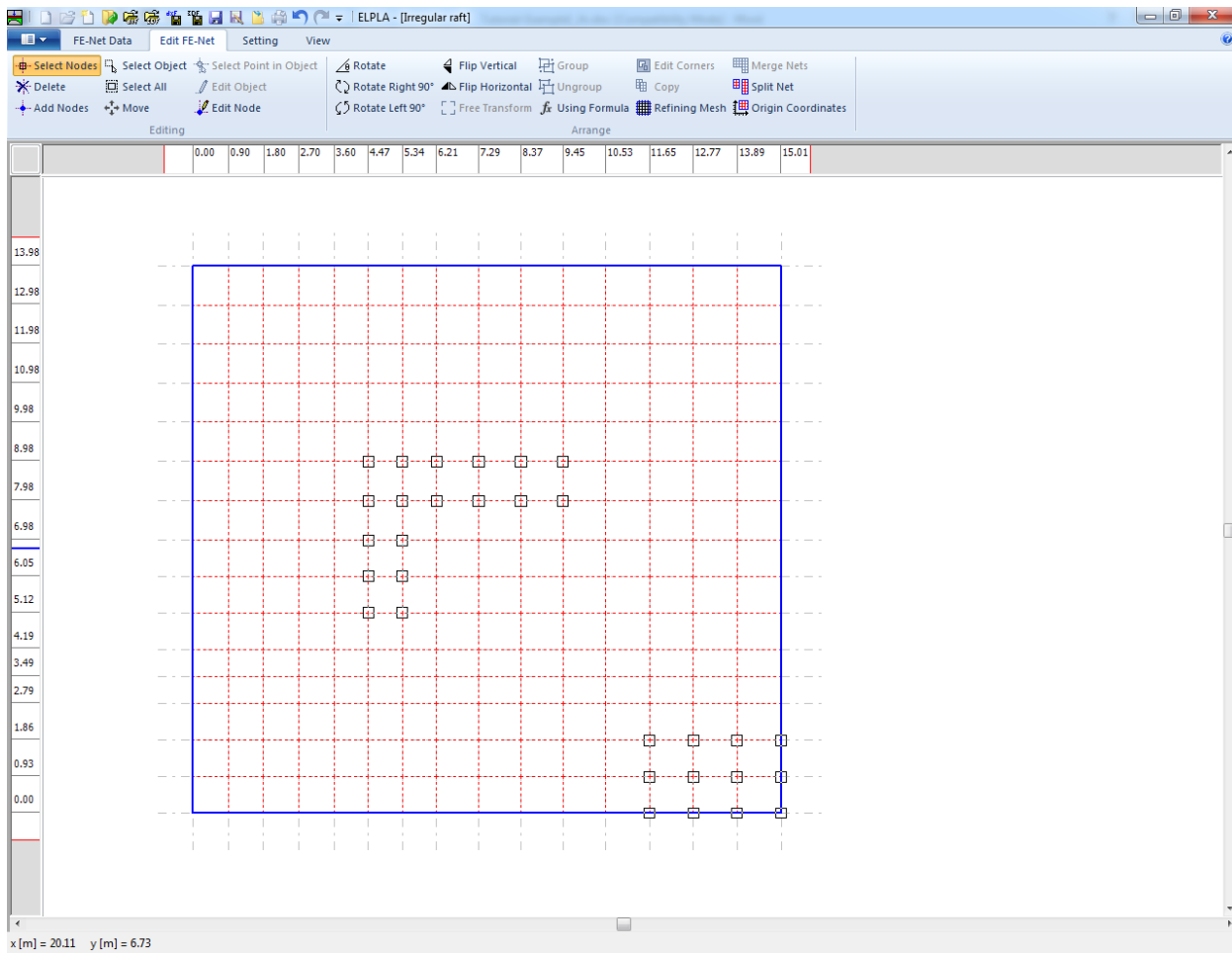


Figure 1.14 Generated FE-Net after selecting the unnecessary nodes

Example 1

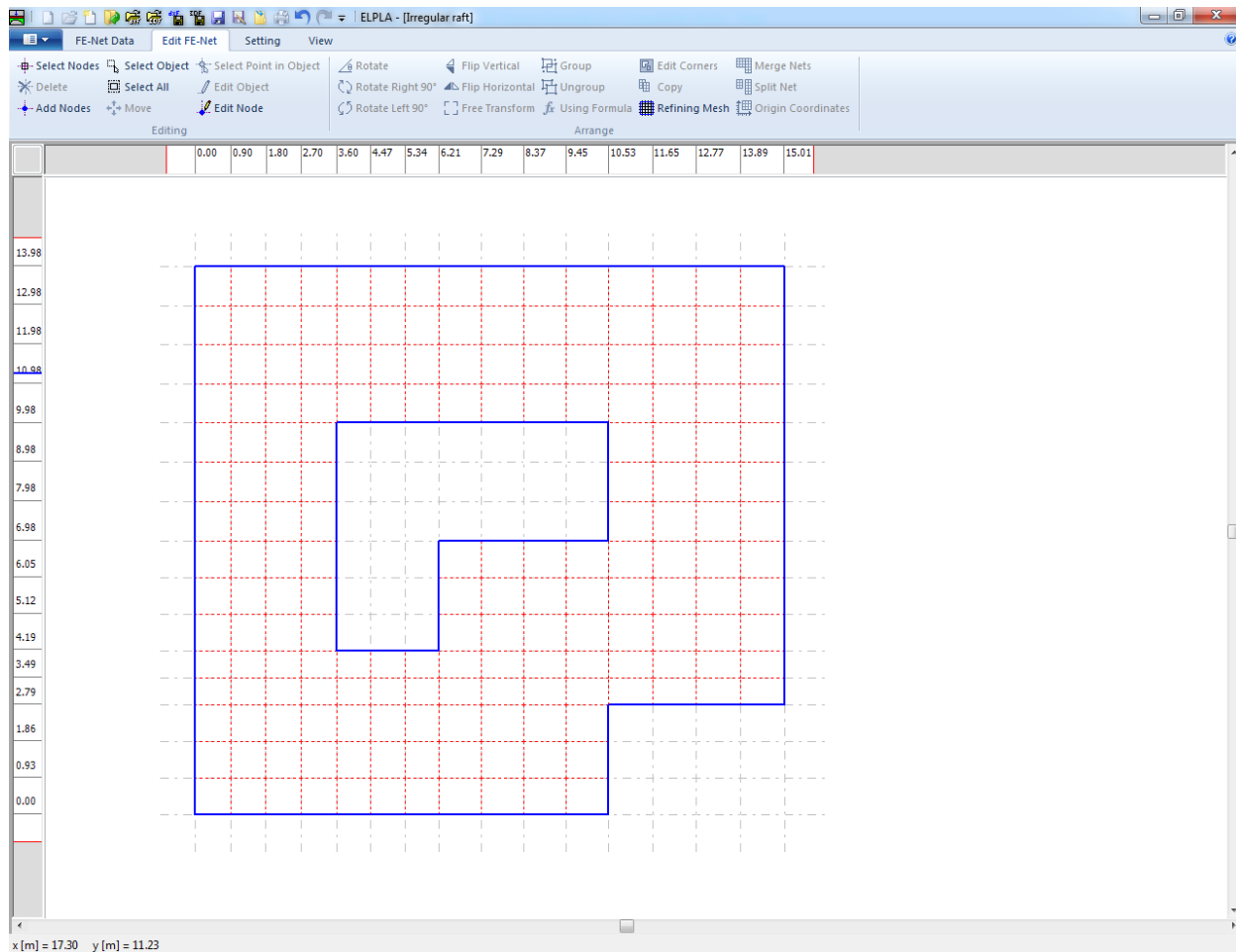


Figure 1.15 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save" command from "File" menu in Figure 1.15 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 1.15 to close the "FE-Net" Window and return to ELPLA main window.

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 have 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" Tab. The following "Soil Properties" Window in Figure 1.16 appears with a default-boring log.

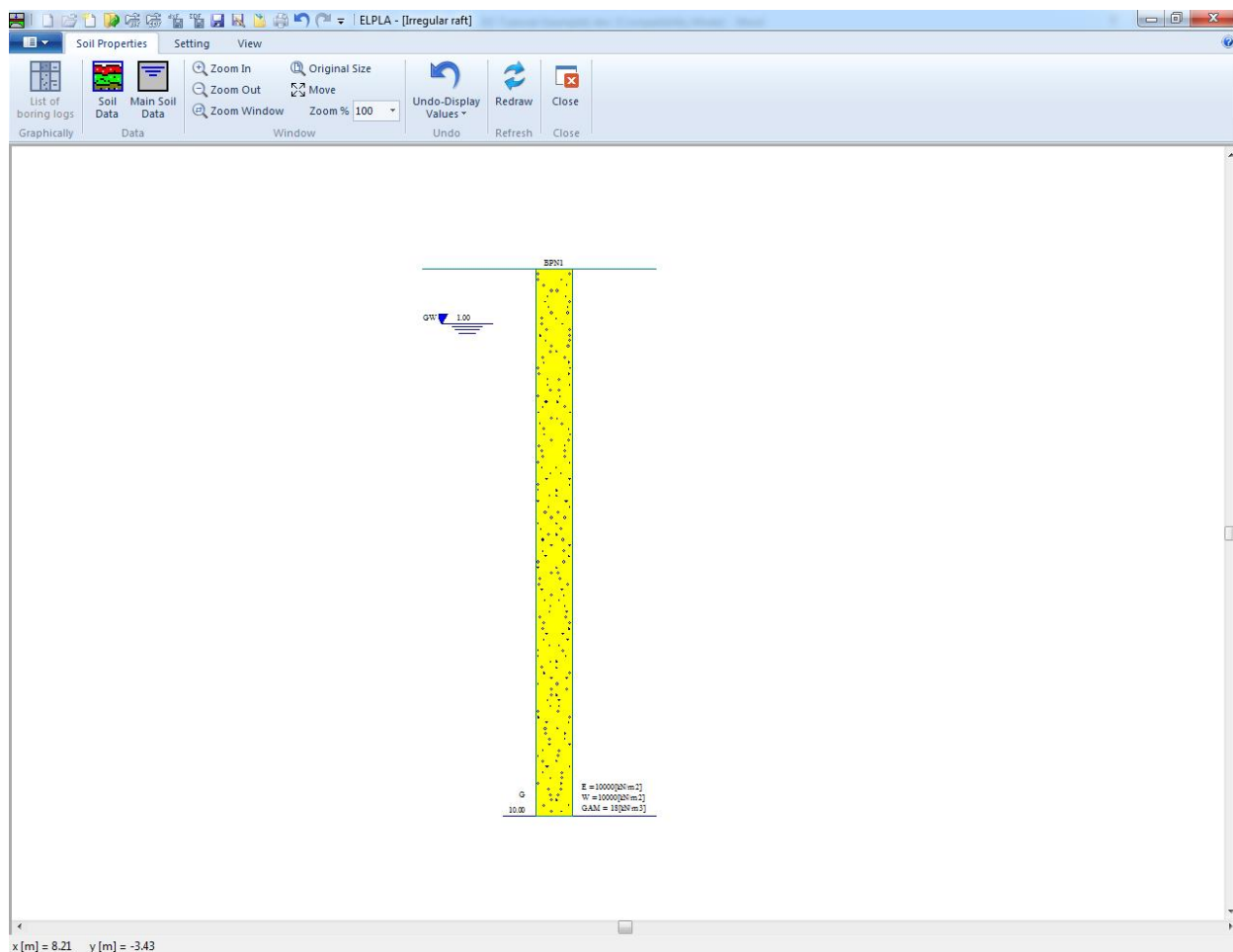


Figure 1.16 "Soil Properties" window with a default-boring log

Example 1

In Figure 1.16, soil properties are defined through the "Soil Properties" Tab, 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

Soil Data

Boring log No. 1 from 1 boring logs:
Layer No. 1 from 1 layers:

Soil and rock symbols:

Main soil type 1: G, Gravel
Main soil type 2: -, No symbol
submain soil 1: -, No symbol
submain soil 2: -, No symbol
Color: ge, yellow
Short text: G

Geotechnical data of the layer:
Soil properties are defined by Modulus of Elasticity E

E [kN/m²]: 10000 FHI [°]: 30
W [kN/m²]: 10000 c [kN/m²]: 0
GAM [kN/m³]: 18 Nue [-]: 0.3

Layer depth under the ground surface [m]: 10.00

Copy Layer Insert Layer Delete Layer

Copy Boring log Insert Boring Log: X-coordinate of boring [m]: 0.00
Delete Boring log From a file Insert Boring Y-coordinate of boring [m]: 0.00
Boring Log Label: BPN1

Ok Cancel New Help

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:

$$\begin{aligned} E_s &= 9\,500 & [\text{kN/m}^2] \\ W_s &= 26\,000 & [\text{kN/m}^2] \\ \text{Gam} &= 19 & [\text{kN/m}^3] \end{aligned}$$

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

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. In addition, a short text "U" will be automatically created for the silt.

To enter the second layer of the first boring log

- Click "Copy Layer" 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/m³] to 9 [kN/m³]
- 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 "Insert Layer" 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:

E_s	= 22 000	[kN/m ²]	Φ	= 30	[°]
W_s	= 52 000	[kN/m ²]	c	= 5	[kN/m ²]
γ_{sat}	= 9	[kN/m ³]			

Example 1

- 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:

E_s	= 120 000	[kN/m ²]	Φ	= 30	[°]
W_s	= 220 000	[kN/m ²]	c	= 5	[kN/m ²]
γ_m	= 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/m²] due to the removed soil, which is equal to $\gamma_s \times d_f$. This means that the unit weight of the soil under the foundation depth d_f is not required. However, the unit weight of the soil under the foundation depth for all soil layers is entered by the value 9 [kN/m³].

After editing the geotechnical data 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 (Figure 1.18). 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.

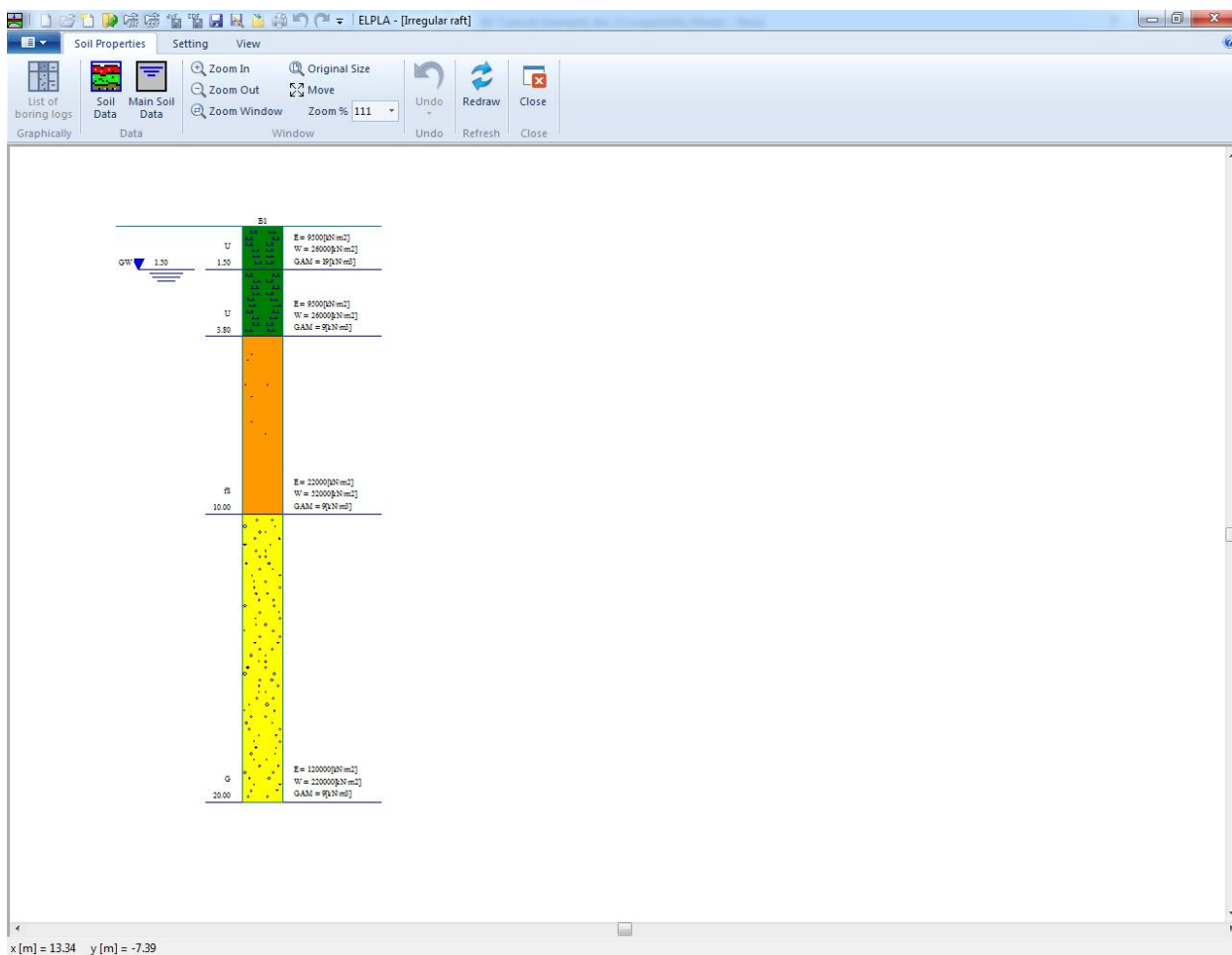


Figure 1.18 First boring log

To create the other two boring logs, click twice "Copy Boring log" button in Figure 1.17. Two boring logs with the same data and parameters of the first boring log will be copied.

Example 1

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 (2 nd layer)	8.2	[m]
Layer depth under the ground surface (3 rd layer)	14.1	[m]
Layer depth under the ground surface (4 th 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 (2 nd layer)	12.7	[m]
Layer depth under the ground surface (3 rd layer)	18.2	[m]
Layer depth under the ground surface (4 th 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 where the user can control or modify the input data and parameters. As a default plot parameter, *ELPLA* displays only the first boring log (Figure 1.18).

To display all boring logs or specified boring logs, choose "List of boring logs" command from "Soil Properties" Tab in Figure 1.16. 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 "Add" 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 to control or modify the boring data graphically, Figure 1.20

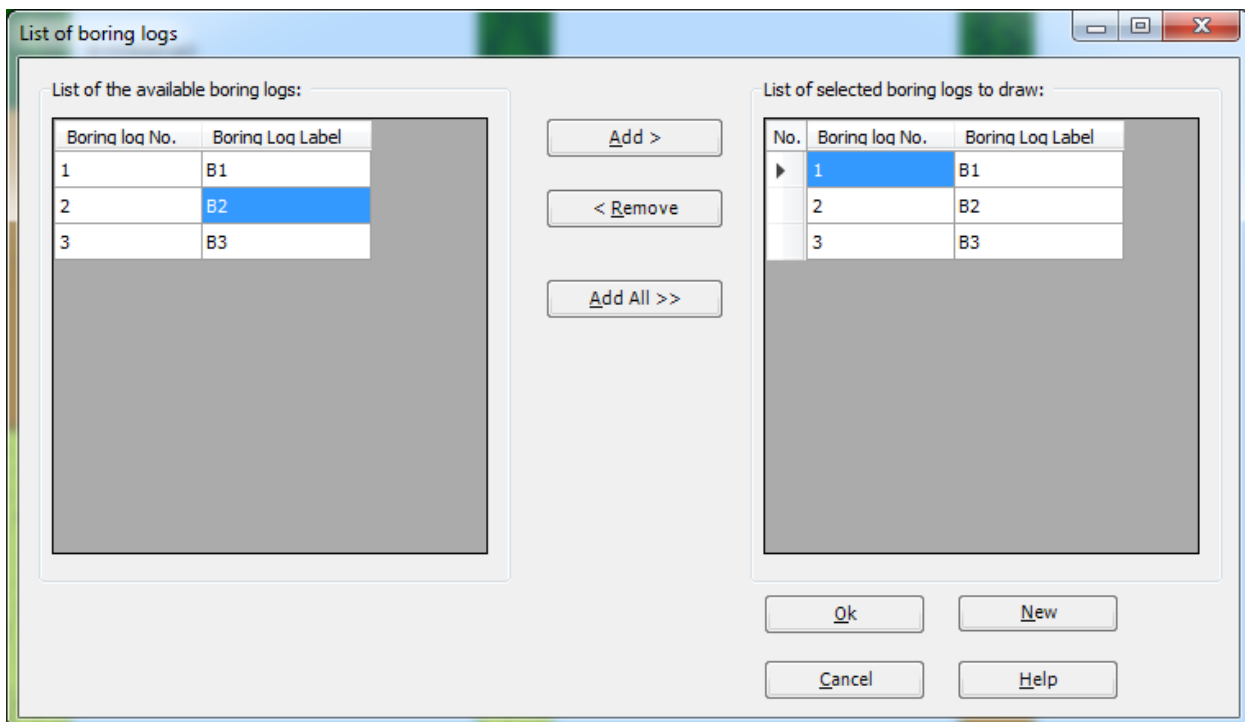


Figure 1.19 "List of boring logs" list box

Example 1

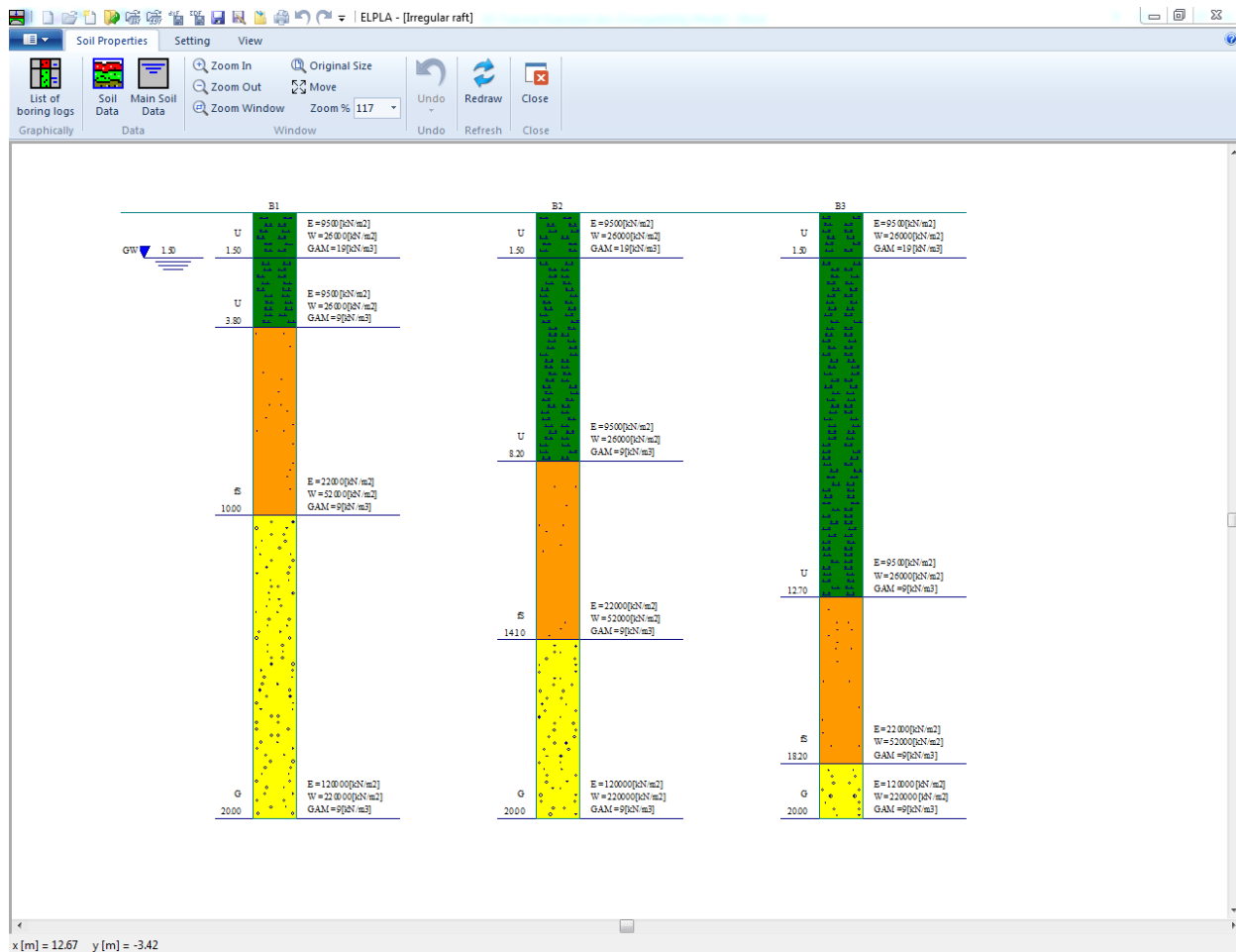
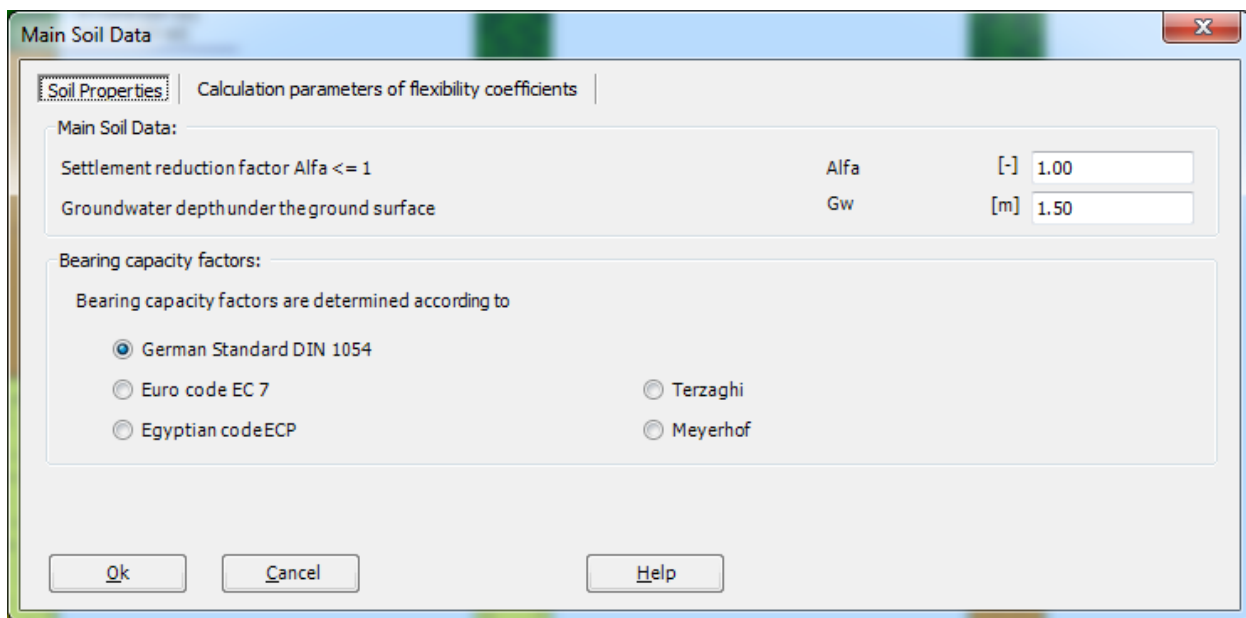


Figure 1.20 Boring logs

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.



The image shows a software dialog box titled "Main Soil Data". It has two tabs: "Soil Properties" (which is active) and "Calculation parameters of flexibility coefficients". Under "Main Soil Data:", there are two input fields: "Settlement reduction factor Alfa <= 1" with a value of 1.00 and unit [-], and "Groundwater depth under the ground surface" with a value of 1.50 and unit [m]. Below this, under "Bearing capacity factors:", there is a text label "Bearing capacity factors are determined according to" followed by five radio button options: "German Standard DIN 1054" (selected), "Euro code EC 7", "Egyptian code ECP", "Terzaghi", and "Meyerhof". At the bottom are three buttons: "Ok", "Cancel", and "Help".

Figure 1.21 "Main Soil Data" dialog box

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

- Choose "Save" command from "File" menu in Figure 1.20 to save the data of boring logs
- Choose "Close" command from "File" menu in Figure 1.20 to close "Soil properties Window" and return to ELPLA main window.

Example 1

2.5 Foundation properties

To define the foundation properties, choose "Foundation Properties" command from "Data" Tab. The following Window 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.

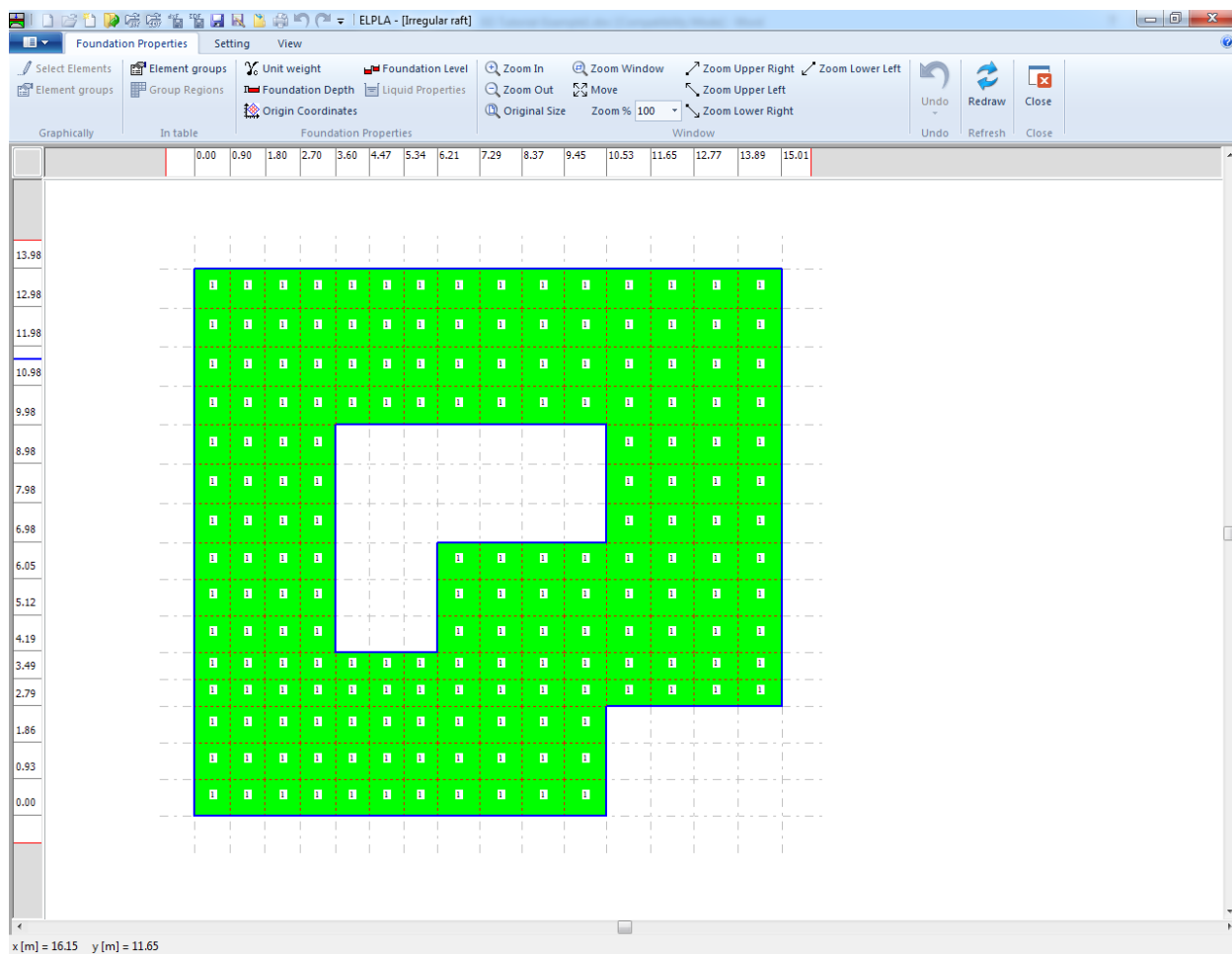


Figure 1.22 "Foundation Properties" Window

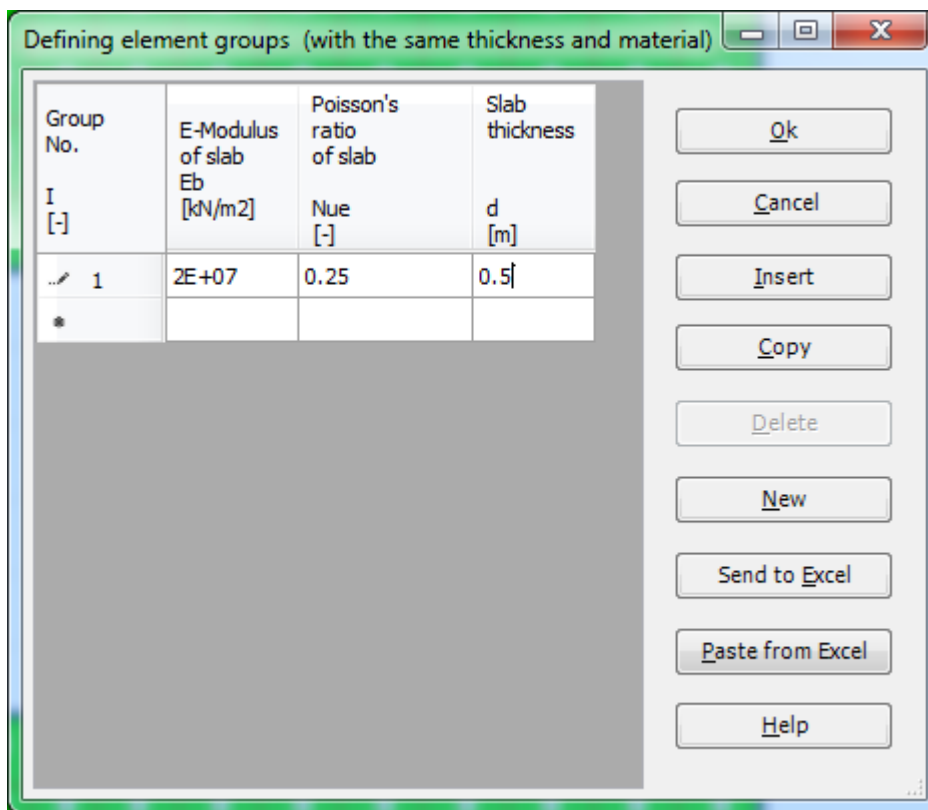


Figure 1.23 "Defining element groups" list box

To enter the unit weight of the raft, choose "Unit Weight" 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/m³] appears. To neglect the self-weight of the raft in the analysis, type 0 in the "Unit Weight" edit box. Click "OK" button.

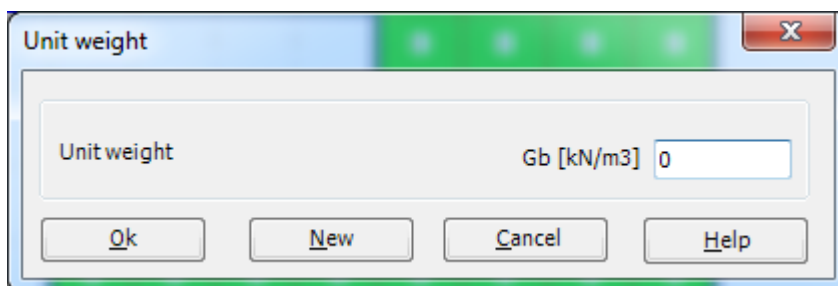


Figure 1.24 "Unit weight of the foundation" dialog box

Example 1

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(a)/ (b)" edit box. Then click "OK" button.

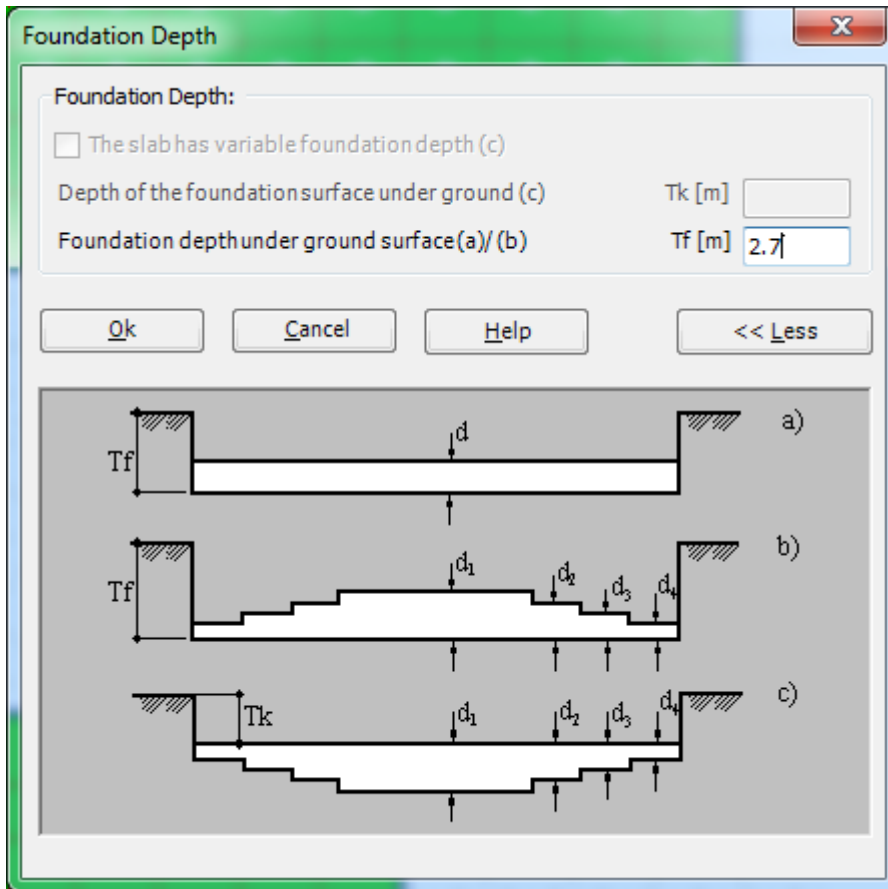


Figure 1.25 "Foundation Depth" dialog box

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

- Choose "Save" command from "File" menu in Figure 1.22 to save the foundation properties
- Choose "Close" command from "File" menu in Figure 1.22 to close the "Foundation properties" Window and return to ELPLA main window.

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" Tab. The following Tab 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. However, in the case of extreme boring arrangements, the user must define these zones.

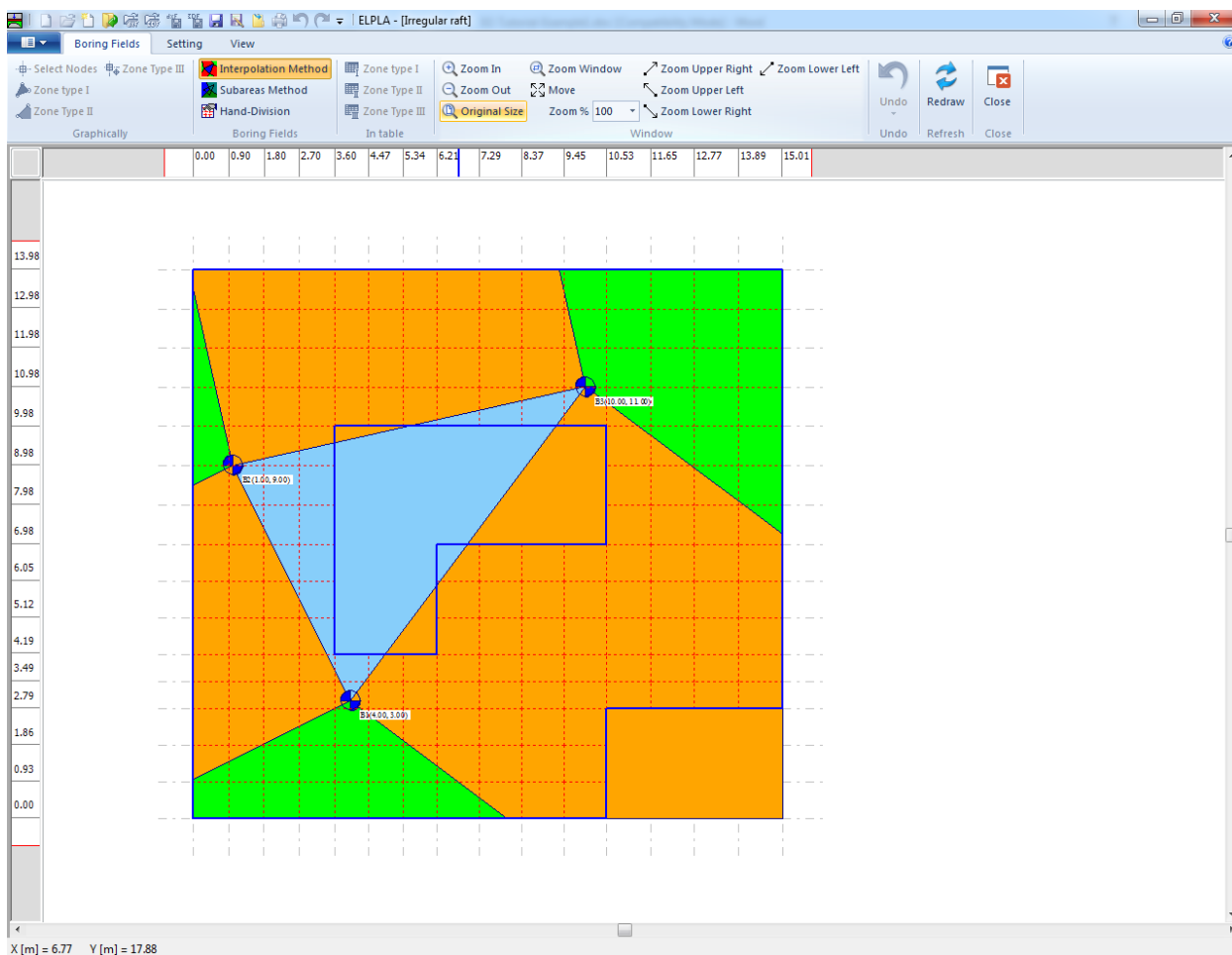


Figure 1.26 "Boring Fields" Window

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

- Choose "Save" command from "File" menu in Figure 1.26 to save the data of boring fields
- Choose "Close" command from "File" menu in Figure 1.26 to close the "Boring Fields" Tab and return to *ELPLA* main window.

Example 1

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" Tab. The following Window in Figure 1.27 appears.

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 1.27. In this example, the user will learn the definition of loads numerically.

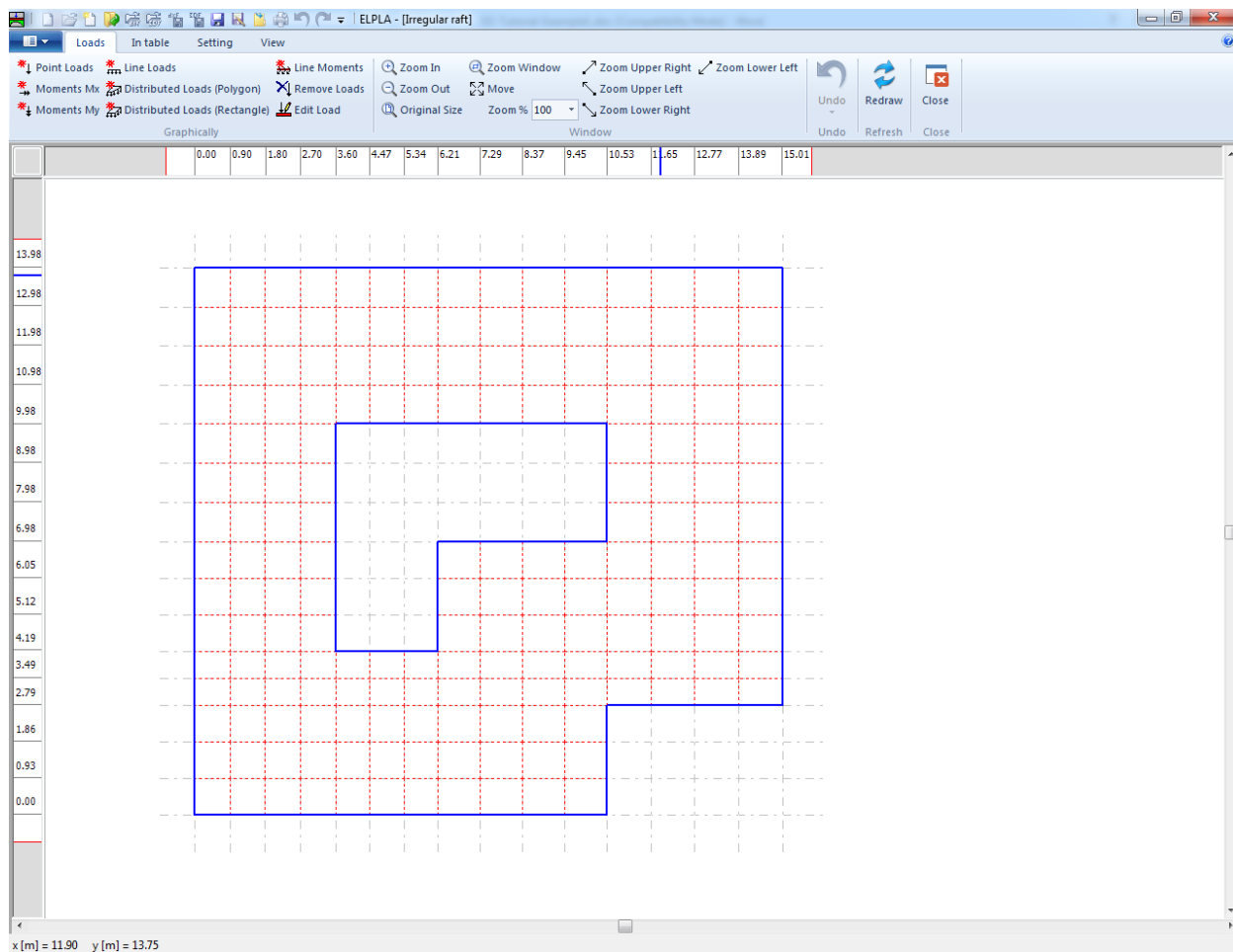


Figure 1.27 "Loads" Window

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 M_x , moments M_y , line loads and distributed loads using "Moments M_x ", "Moments M_y ", "Line loads" and "Distributed loads" commands from "Loads" Tab 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]	Column label CZ
1	1	1265	1.50	1.40	C1
2	1	1600	1.50	5.50	C2
3	1	1350	1.50	9.90	C3
4	1	1368	1.50	12.60	C4
5	1	1560	5.00	1.40	C5
6	1	1538	5.00	12.60	C6
7	1	800	9.20	1.40	C7
8	1	750	9.20	5.50	C8
9	1	1565	9.20	12.60	C9
10	1	2150	13.40	5.50	C10
11	1	1450	13.40	9.90	C11
12	1	1254	13.40	12.60	C12

Figure 1.28 "Point loads P " List box

Example 1

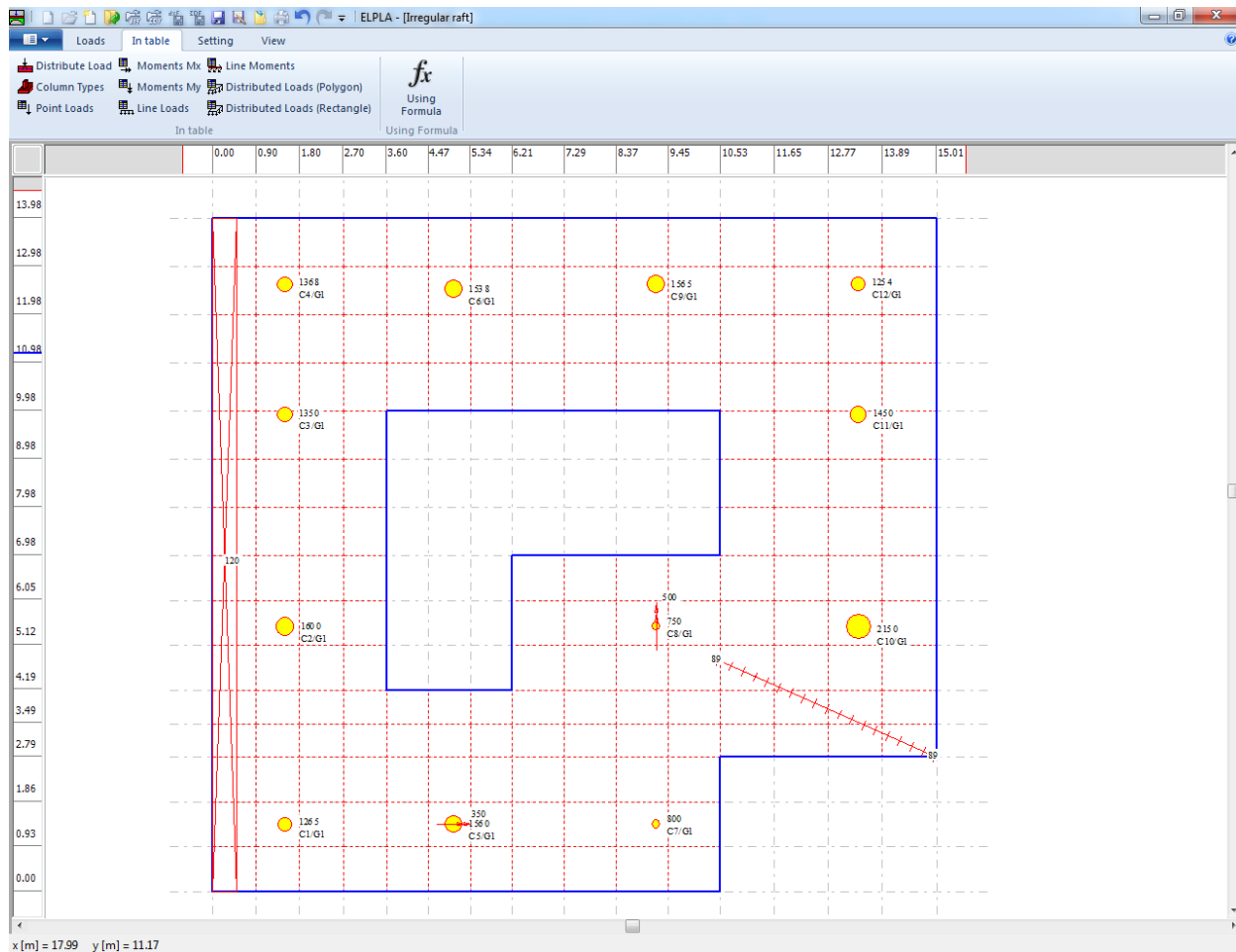


Figure 1.29 Loads

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

- Choose "Save" command from "File" menu in Figure 1.29 to save the load data
- Choose "Close" command from "File" menu in Figure 1.29 to close the "Loads" Window and return to *ELPLA* main window.

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

To analyze the problem, switch to "Solver" Tab, Figure 1.30.

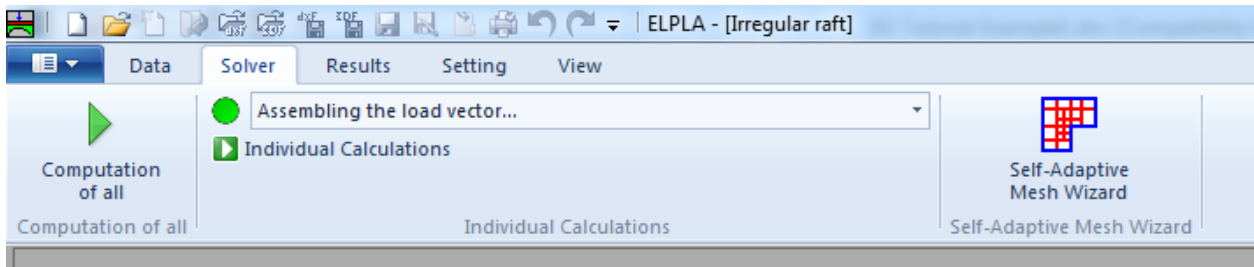


Figure 1.30 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which 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

Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab 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

Example 1

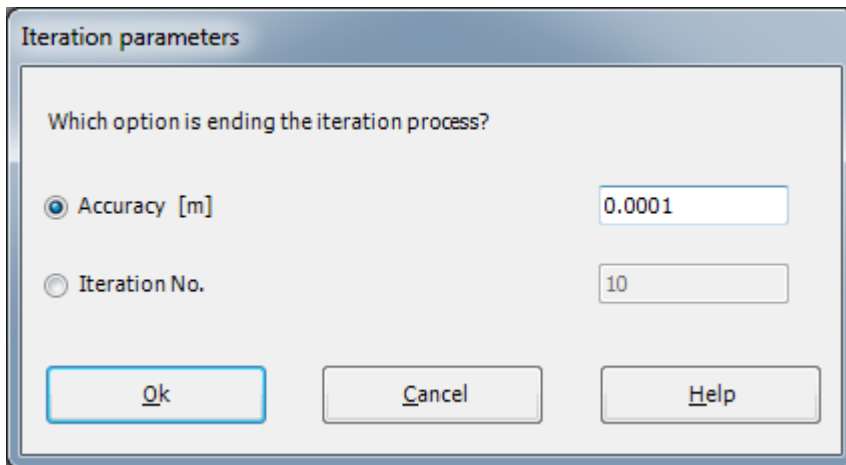


Figure 1.31 "Iteration parameters" option box

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. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

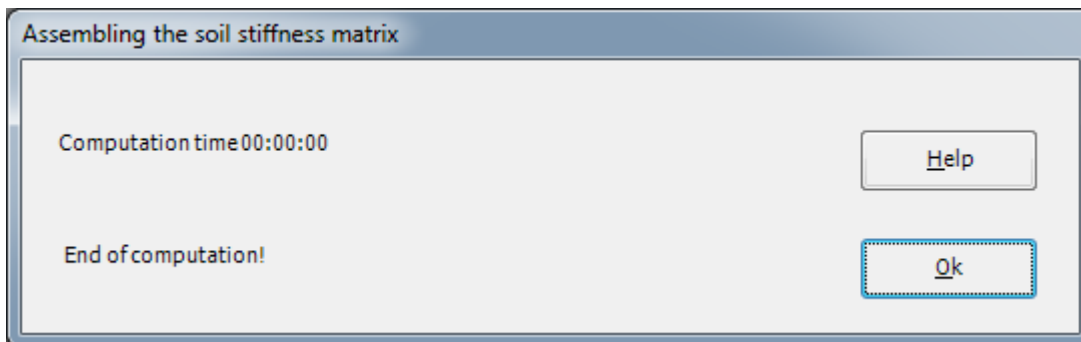


Figure 1.32 Analysis progress menu

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.

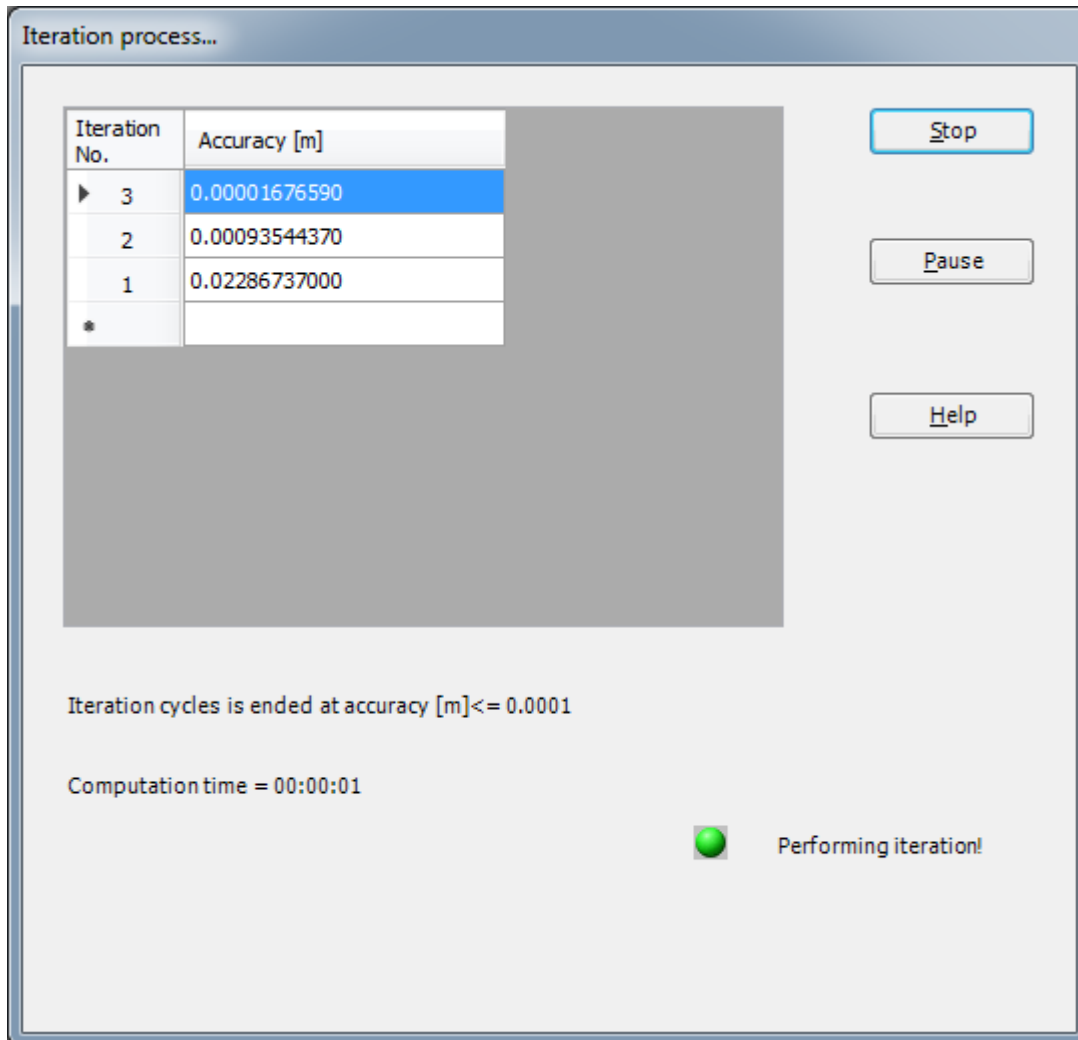


Figure 1.33 "Iteration process" list box

Example 1

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.

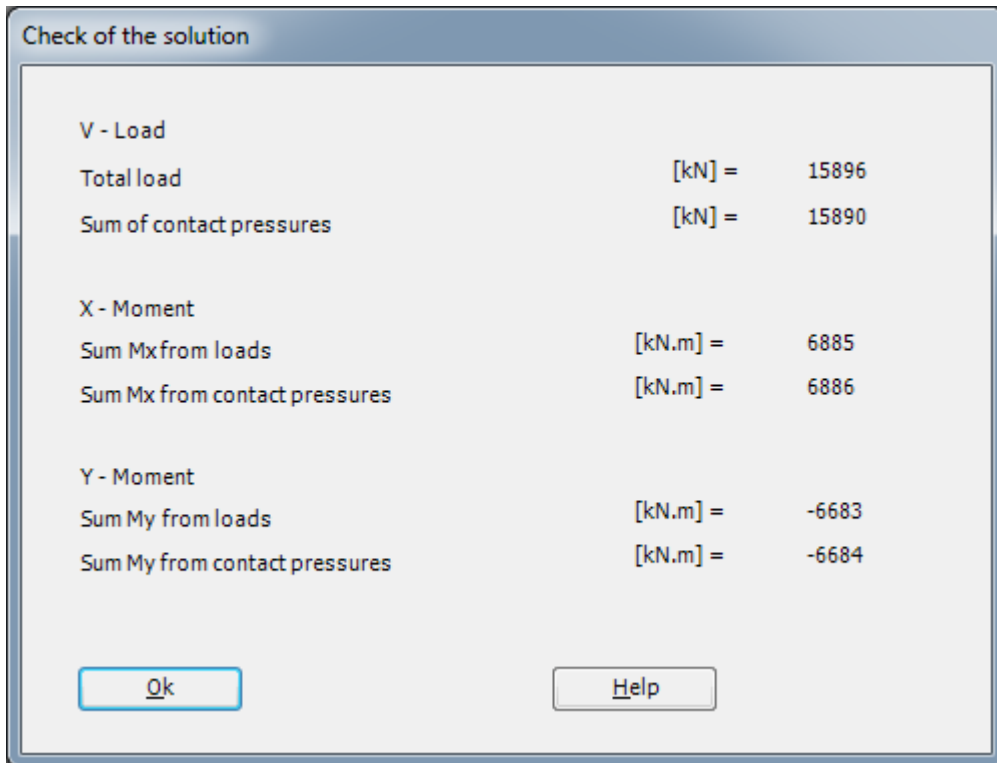


Figure 1.34 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

4 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab.

4.1 Viewing data and result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 1.35).

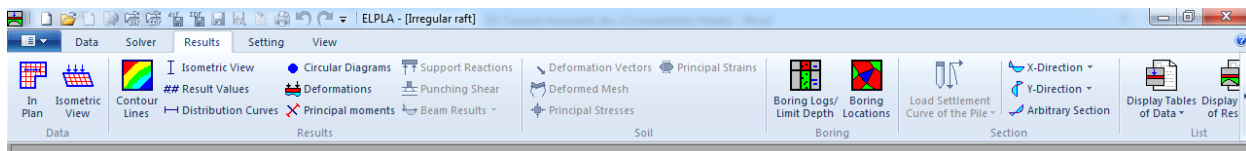


Figure 1.35 "Results" Tab

The "Result" Tab 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:

- Isometric view
- Contour lines
- Result Values
- Distribution Curves
- Circular diagrams
- Deformations
- Principal moments
- Data in isometric view
- Data in the plan
- Boring locations
- Boring logs

Only the first command of the "Result" Tab is explained here. In the same way, the user can carry out the remaining commands of the previous list.

To view the results in isometric view

- Choose "Isometric View" command from "Result" menu. The following option box in Figure 1.36 appears
- In the "Results in Isometric View" option box, select "Settlements" as an example for the results to be displayed
- Click "OK" button

The Settlements are now displayed in an isometric view as shown in Figure 1.37.

Example 1

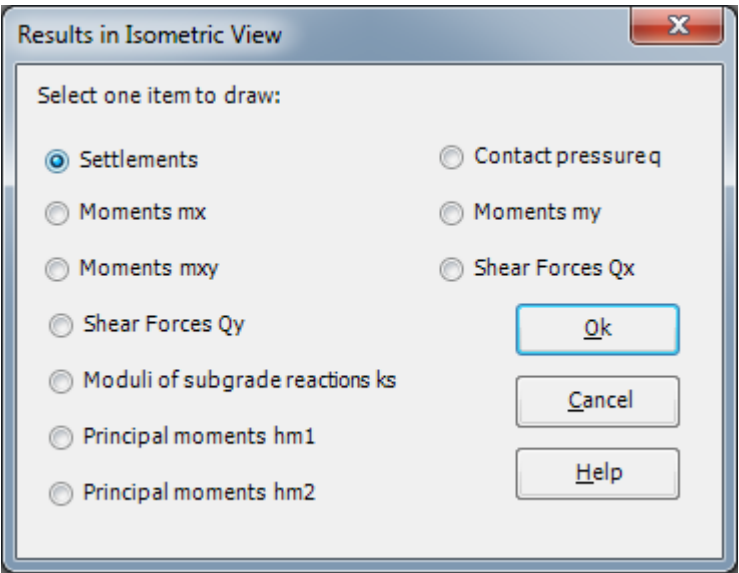


Figure 1.36 "Results in isometric view" option box

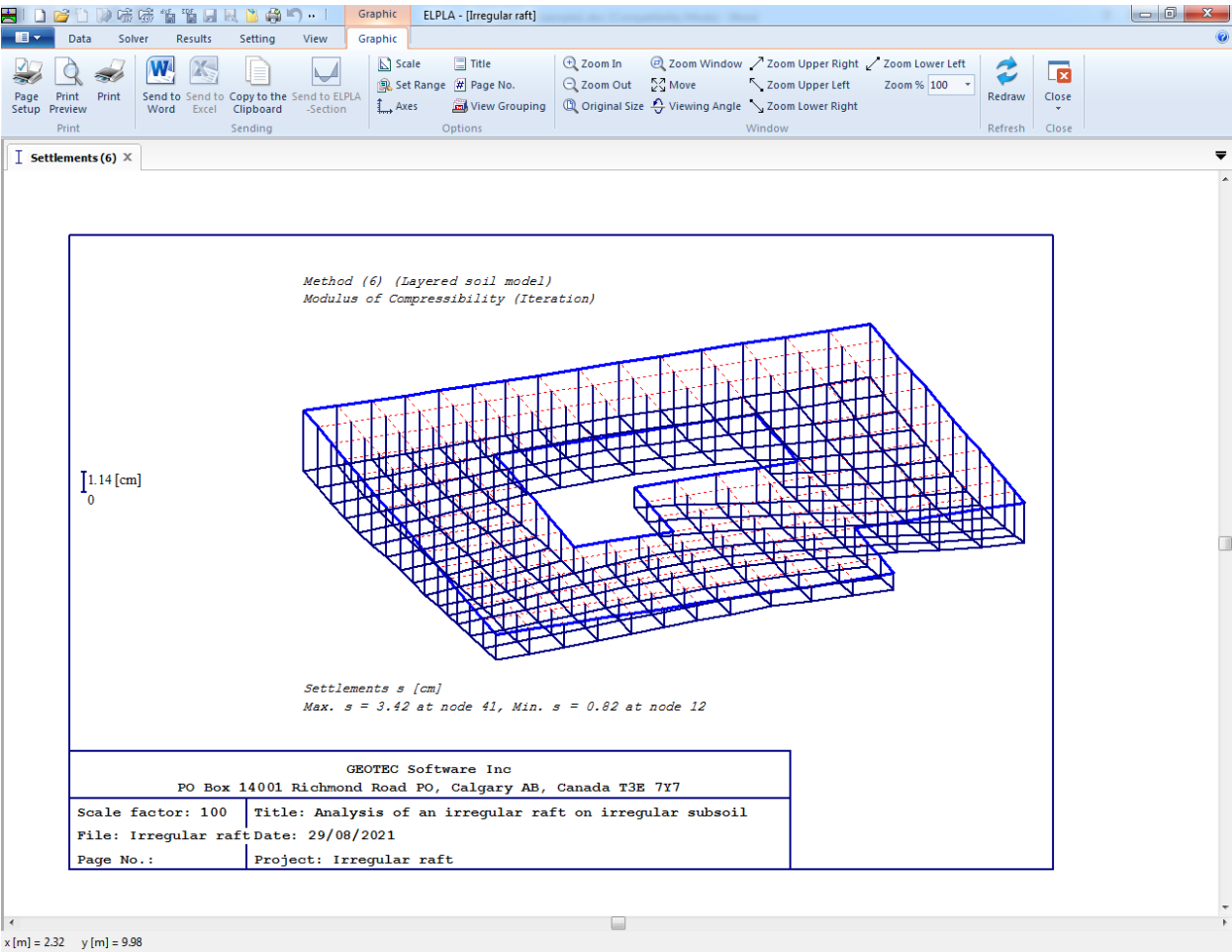


Figure 1.37 Settlements in an isometric view

4.2 Plot a diagram of the results at a specified section

The "Result" Tab 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 sections in the "Result" Tab is explained here. In the same way, the user can carry out the remaining commands of the previous list.

To plot a section in x -direction

- Choose "X-Direction" command from "Section" menu. The following option box in Figure 1.38 appears
- In the "Section in x -direction" option box, select "Settlements" as an example for the results to be plotted in a diagram
- Click "OK" button

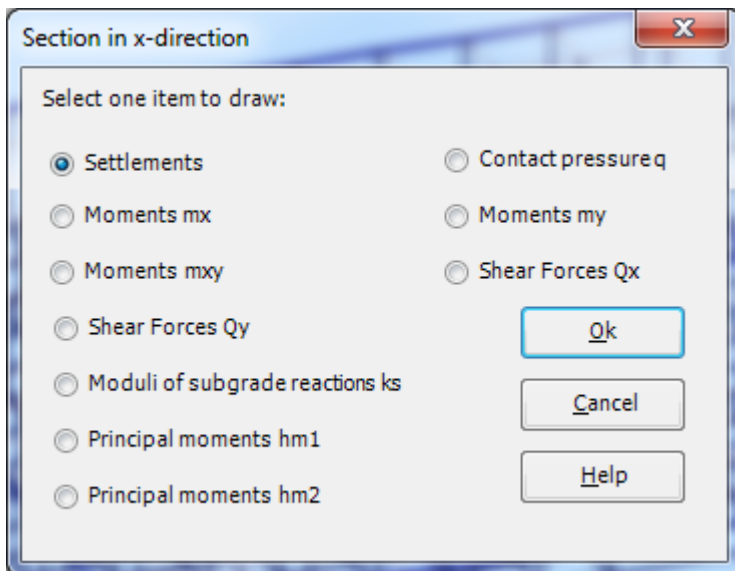


Figure 1.38 "Section in x -direction" option box

Example 1

The following dialog box in Figure 1.39 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 1.40.

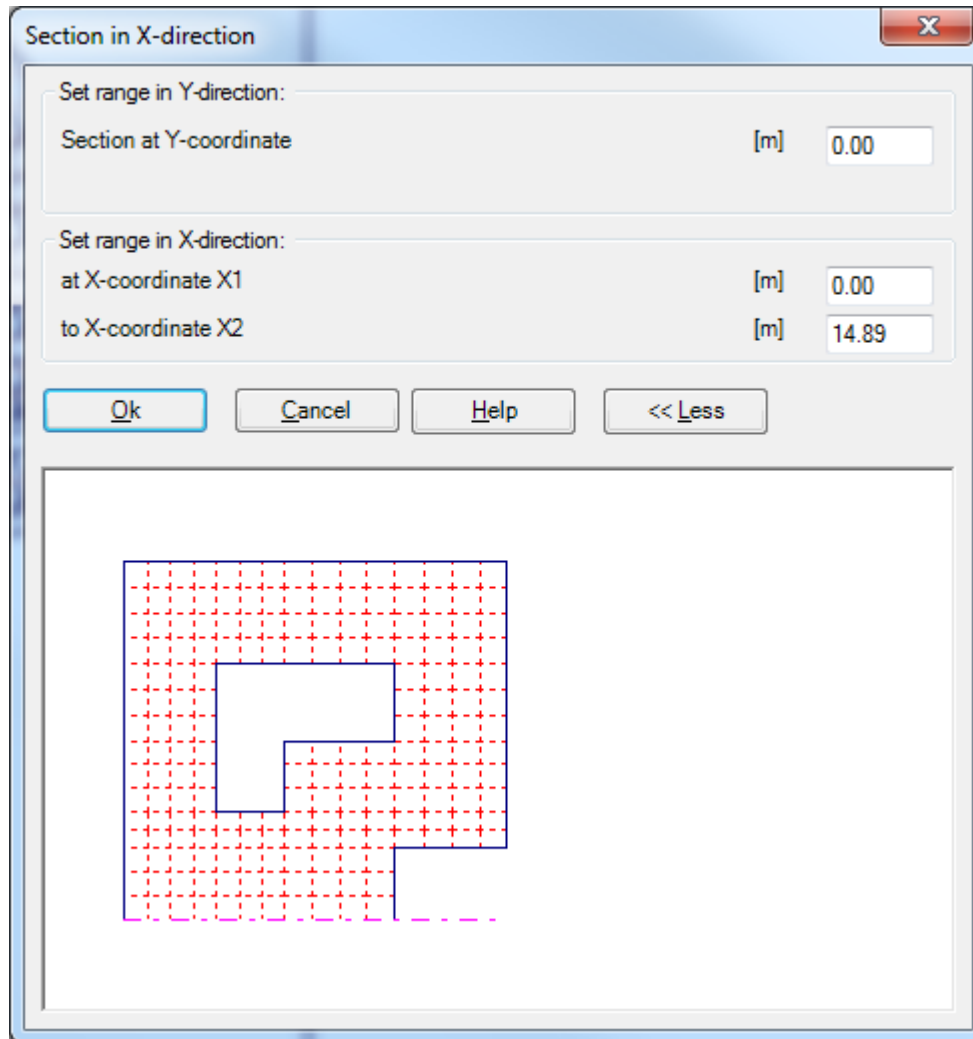


Figure 1.39 "Section in x -direction" dialog box with a default section

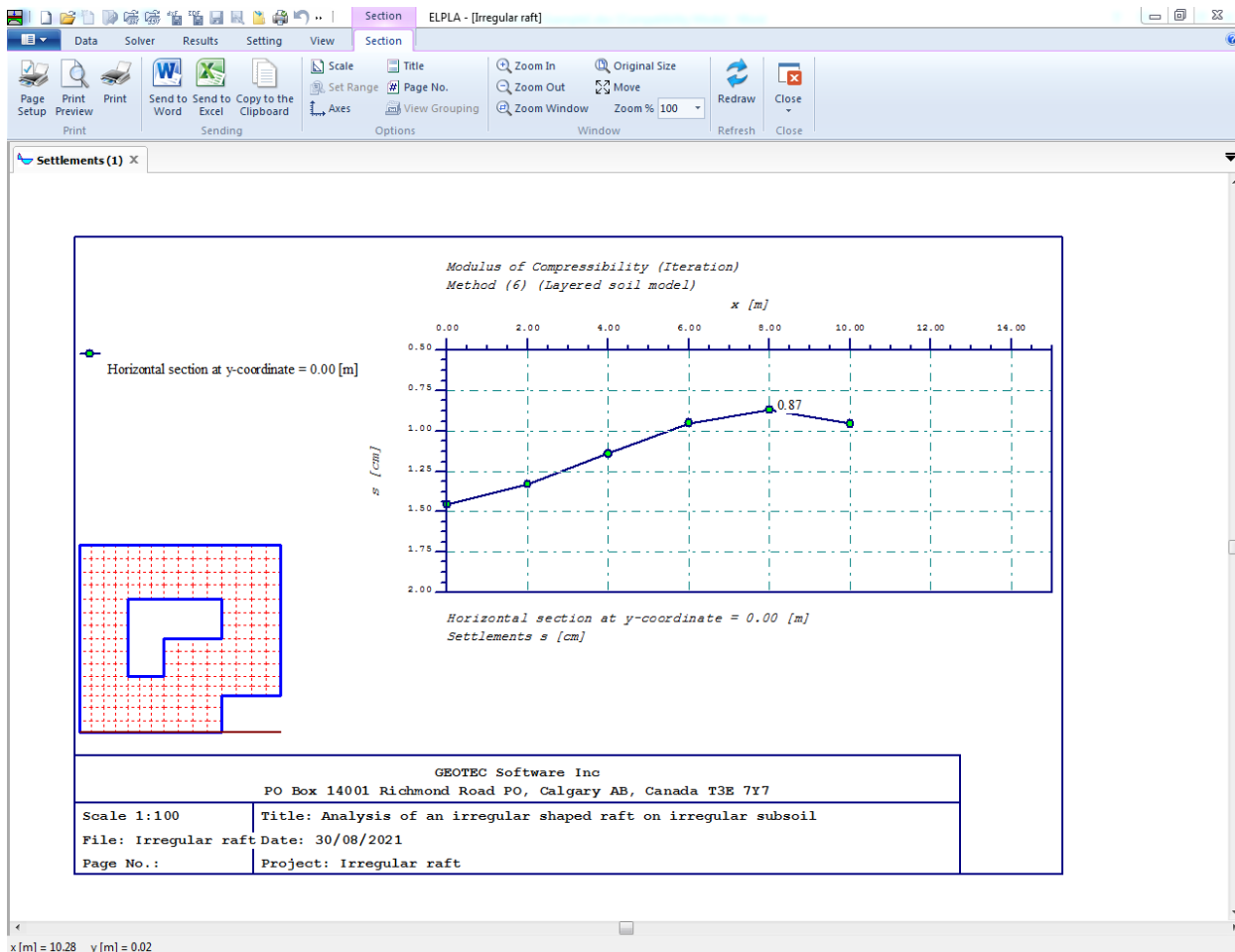


Figure 1.40 Diagram of settlements in x-direction

4.3 Print the drawing

To print the drawing

- Choose "Print" command from "Print" menu in the current drawing Window. The following "Print" dialog box in Figure 1.41 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.

Example 1

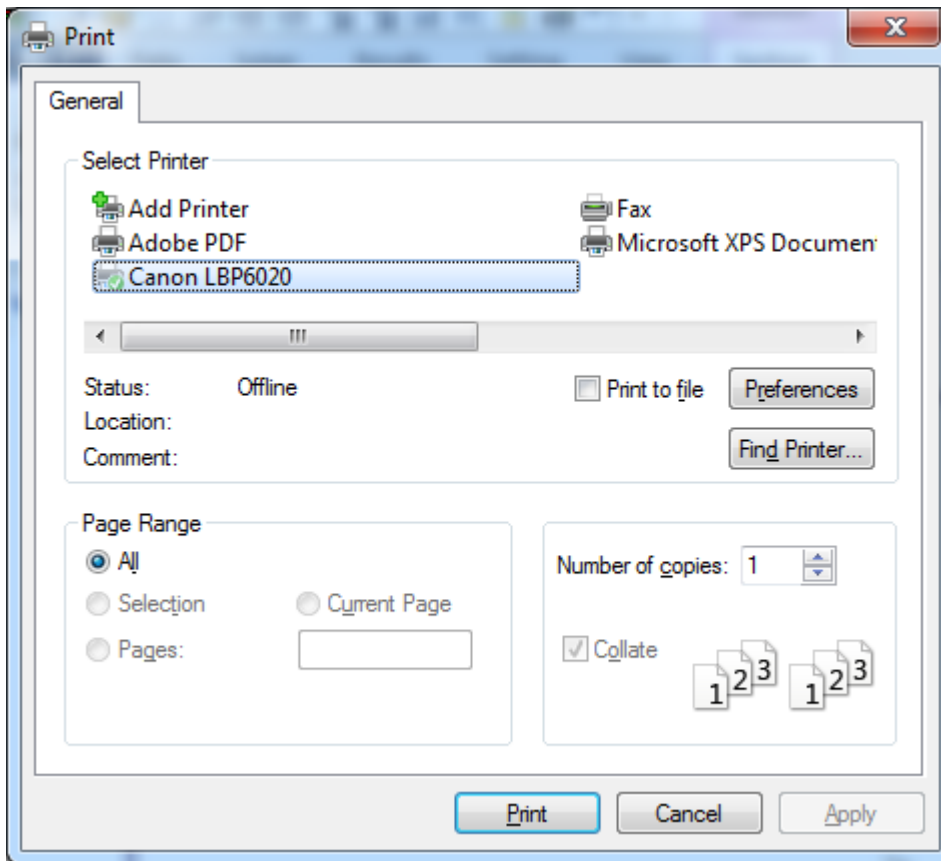


Figure 1.41 "Print" dialog box

4.4 Listing data and results in tables

The "Result" Tab contains the commands of listing data and results. The commands for listing data and results in tables are:

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

Only the first command of listing in "Result" Tab is explained here. In the same way, the user can carry out the remaining commands of the previous list.

To list data in a table

- Choose "Display tables of data" command from "Result" Tab. The following option box in Figure 1.42 appears
- In the "Display tables of data" option box, select "Loads" as an example for the data to be listed in a table
- Click "OK" button. The loading data are now listed (Figure 1.43)
- Choose "Send to Word" from "Result" Tab if you wish to export the table to a MS Word application, Figure 1.44



Figure 1.42 "Display Tables of Data" option box

Example 1

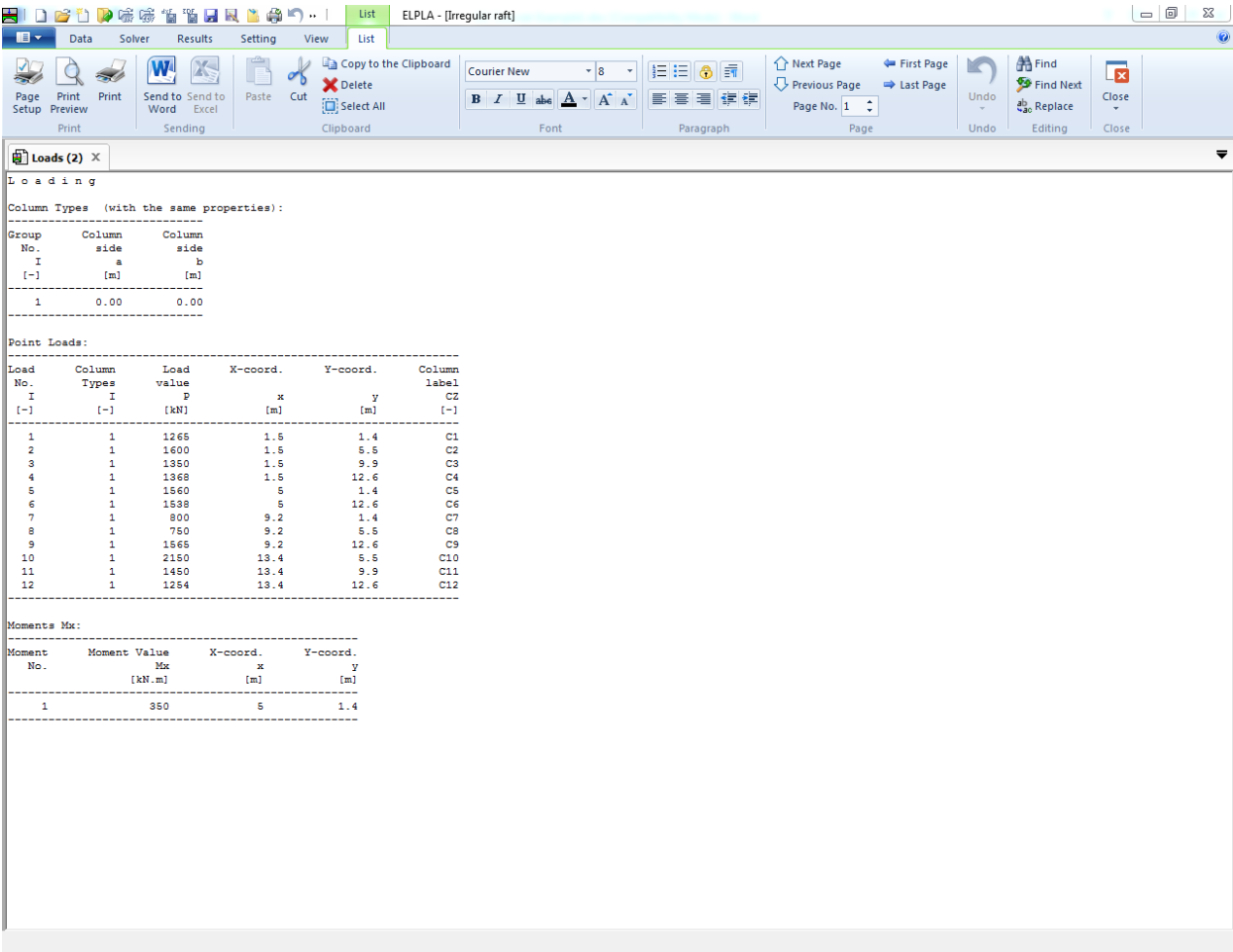


Figure 1.43 List of loading data

Document1 - Word

FILE HOME INSERT DESIGN PAGE LAYOUT REFERENCES MAILINGS REVIEW VIEW

Clipboard Font Paragraph Styles Editing

Sign in

Find Replace Select

Column Types (with the same properties):

Group No.	Column side	Column side
I	a	b
[~]	[m]	[m]
1	0.00	0.00

Point Loads:

Load No.	Column Types	Load value	X-coord. x	Y-coord. y	Column label
I	I	P	x	y	CZ
[~]	[~]	[kN]	[m]	[m]	[~]
1	1	1265	1.5	1.4	C1
2	1	1600	1.5	5.5	C2
3	1	1350	1.5	9.9	C3
4	1	1368	1.5	12.6	C4
5	1	1560	5	1.4	C5
6	1	1538	5	12.6	C6
7	1	800	9.2	1.4	C7
8	1	750	9.2	5.5	C8
9	1	1565	9.2	12.6	C9
10	1	2150	13.4	5.5	C10
11	1	1450	13.4	9.9	C11
12	1	1254	13.4	12.6	C12

Moments Mx:

Moment No.	Moment Value	X-coord. x	Y-coord. y
[~]	[kN.m]	[m]	[m]
1	350	5	1.4

Figure 1.44 Exported data in MS Word

5 Index

A

Analysis type 7

B

Boring fields 32
boring logs 4, 32

C

Calculation 36
Calculation methods 7
check of the solution 39
Computation of all 36

D

deformation 36
diagrams 40

F

FE-Net 12
File 35
Foundation depth 31
foundation properties 29

G

Generation type 13
graphics 40
Grid spacing 14

I

internal forces 36
Iteration process 36, 38

L

loads 3, 33

M

Main soil data 27
Max./ Min. values in x-direction 42
Max./ Min. values in y-direction 42
Moments 34

N

New project 7

O

Overlapping in x-direction 42

P

Point loads 34
project 36
Project identification 12

R

raft material 3
Remove nodes 17
results 40

S

Save 35
Save project as 10
Section in x-direction 42
Section in y-direction 42
Select nodes 17
soil model 4
soil properties 20
soil stiffness matrix 36
System symmetry 8

U

unit weight of the raft 30