Example 9

Influence of a new neighboring building II on an old one I

Contents
Page

1 Description of the problem 3
1.1 Loads and dimensions 3
1.2 Foundation material and thickness 5
1.3 Soil properties 5
1.4 Mathematical model 5
1.5 Analysis 5
2 Creating the project of the new building II 6
2.1 Calculation method 6
2.2 Project identification 11
2.3 FE-Net data 11
2.4 Soil properties 18
2.5 Foundation properties 24
2.6 Loads 27
3 Creating the project of the Old Building I 30
3.1 Modifying the calculation methods options 31
3.2 Modifying the project identification 32
3.3 Modifying origin coordinates 33
3.4 Neighboring foundations 35
4 Carrying out the calculations 36
5 Viewing data and results 39
5.1 Viewing result graphics 39
5.2 Drawing a graph of results 42
6 Index 47

Example 9

1 Description of the problem

An example of two neighboring buildings is selected to illustrate some of the essential features of *ELPLA* for studying the influence of a new building on an existing old one.

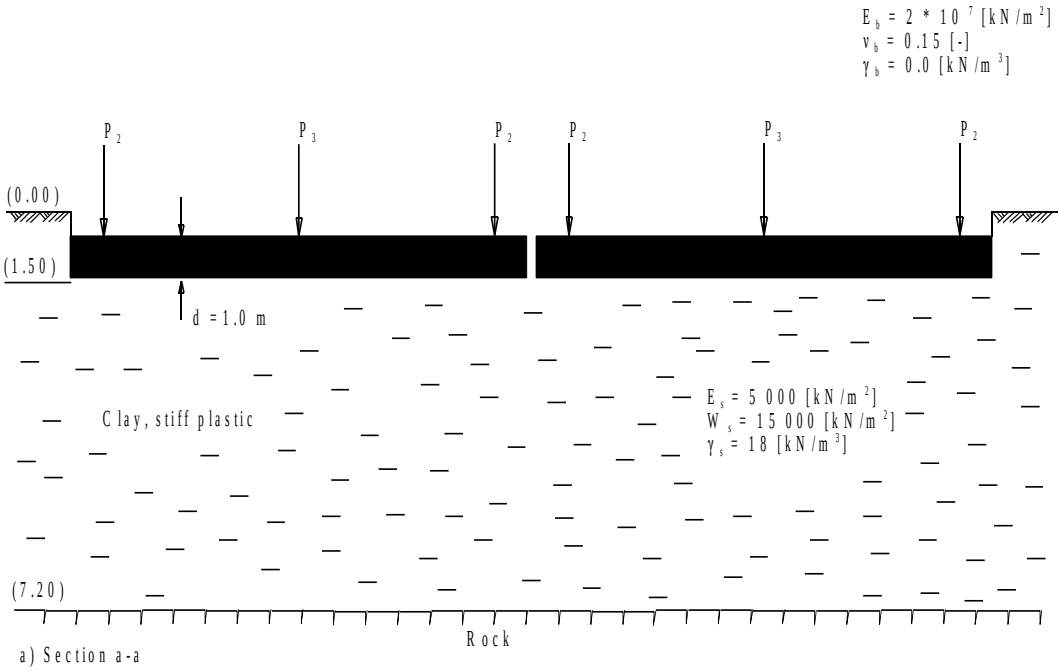
1.1 Loads and dimensions

Figure 9.1 shows plan and section of a new building II beside a similar old one I. The building II will be constructed close to the first one. The two buildings have the same construction geometry and loads. In addition, every building is symmetrical about both *x*- and *y*-axes.

Each foundation has a side of 10.08 [m] and thickness of 1.0 [m]. Loading on each foundation consists of 9 column loads as shown in Figure 9.1 and in Table 9.1. In spite of the two buildings are closed to each other, a small distance of 20 [cm] is assumed between them to avoid overlapping their nodes. The origin coordinates for the building I in the global system are (10.28, 0.0) while those for the building II are (0.0, 0.0).

Table 9.1 Point loads *P*

Load No.	Load value	X-coord.	Y-coord.
I	P	x	y
[-]	[kN]	[m]	[m]
1	500	0.72	0.72
2	500	0.72	9.36
3	500	9.36	9.36
4	500	9.36	0.72
5	1000	5.04	9.36
6	1000	5.04	0.72
7	1000	9.36	5.04
8	1000	0.72	5.04
9	2000	5.04	5.04



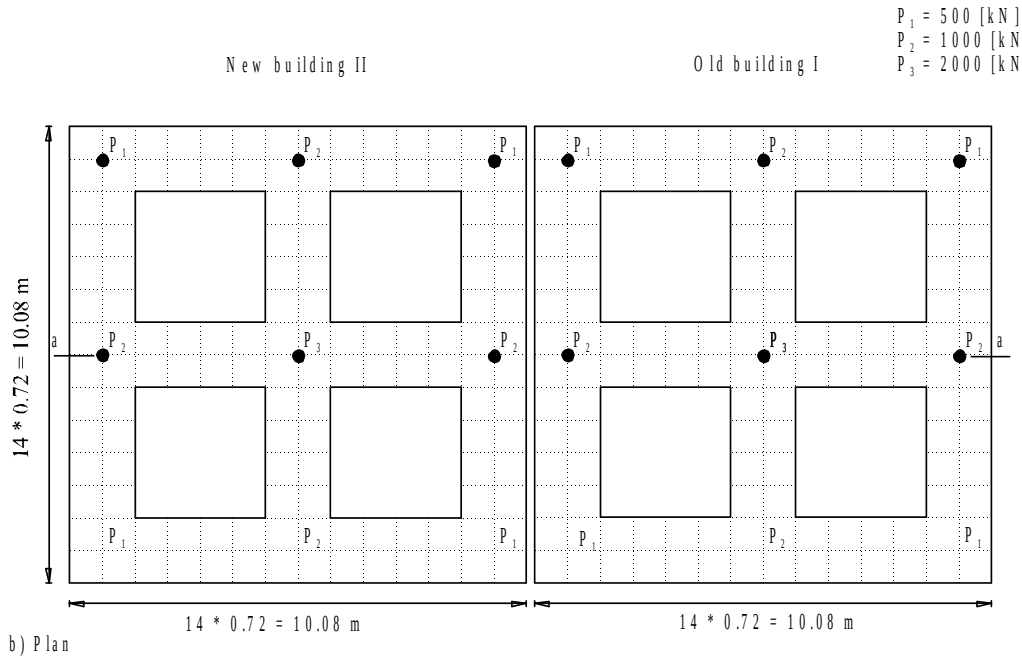


Figure 9.1 Action of new building II on the old building I

1.2 Foundation material and thickness

The foundation material and thickness are supposed to have the following parameters:

Foundation thickness	d	= 1.0	[m]
Young's modulus	E_b	= 2×10^7	[kN/m^2]
Unit weight	γ_b	= 0.0	[kN/m^3]
Poisson's ratio	ν_b	= 0.15	[-]

Unit weight of the foundation material is assumed zero to neglect its own weight in the analysis.

1.3 Soil properties

The subsoil under the buildings consists of a layer of stiff plastic clay with 7.2 [m] thickness, overlying a rigid base (Figure 9.1). The soil is supposed to have the following parameters:

Modulus of compressibility for loading	E_s	= 5000	[kN/m^2]
Modulus of compressibility for reloading	W_s	= 15000	[kN/m^2]
Unit weight	γ_s	= 18	[kN/m^3]

1.4 Mathematical model

The influence of surrounding structures and external loads can be taken into consideration only for the Continuum model (methods 4 to 9). The Continuum model based on, the settlement at any node is affected by the forces at all the other nodes. In this example, the Modulus of compressibility method (method 7) is chosen to analyze both of the two buildings.

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*.

1.5 Analysis

To analyze foundations, each foundation is subdivided into elements with 132 nodes as shown in Table 9.1. Two independent names define the data of the two buildings are chosen. The data are quite similar for the two buildings except the origin coordinates, which are chosen
to be (x_o, y_o) = (10.28, 0.0) and (0.0, 0.0) for buildings I and II, respectively. In spite of the two buildings are closed to each other, a small distance of 20 [cm] is assumed between them to avoid overlapping their nodes.

The analysis of the new building II is carried out first to obtain the contact pressures under it. Due to these contact pressures, settlements will occur not only under Building II, but also outside under Building I. Then, assuming that a new building will be built to the left of the old

building, the contact pressures, settlement and internal forces of the old building are determined.

2 Creating the project of the new building II

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

2.1 Calculation method

To create the project, Choose the "New Project" command from the "File" menu. The "Calculation Method" wizard appears, Figure 9.2. This wizard will guide you through the steps required to create the project. As shown in Figure 9.2, the first form of the wizard is the "Analysis Type" form.
In this form, 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 page.

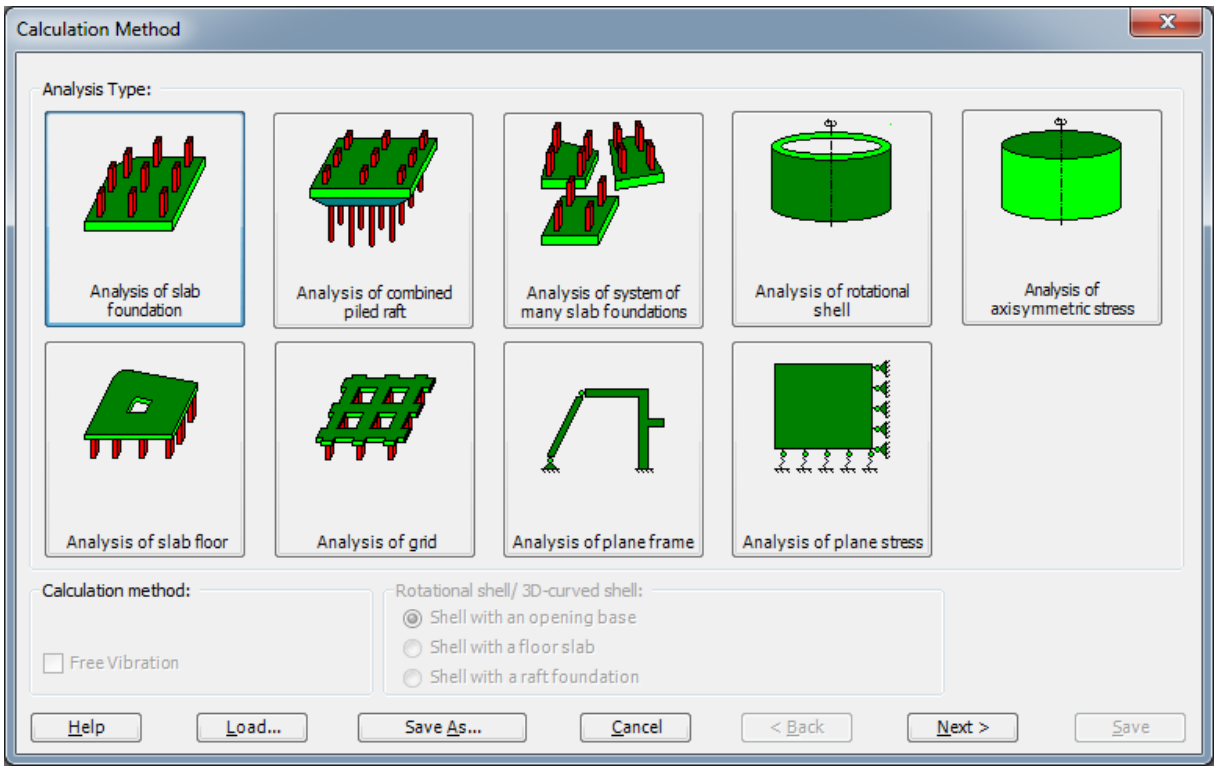


Figure 9.2 "Calculation Method" wizard with "Analysis Type" form

After clicking "Next" button, the "Calculation Method" form appears, Figure 9.3.

To define the calculation method

- Select the calculation method "7-Modulus of Compressibility (Elimination)"
- Click "Next" button to go to the next form

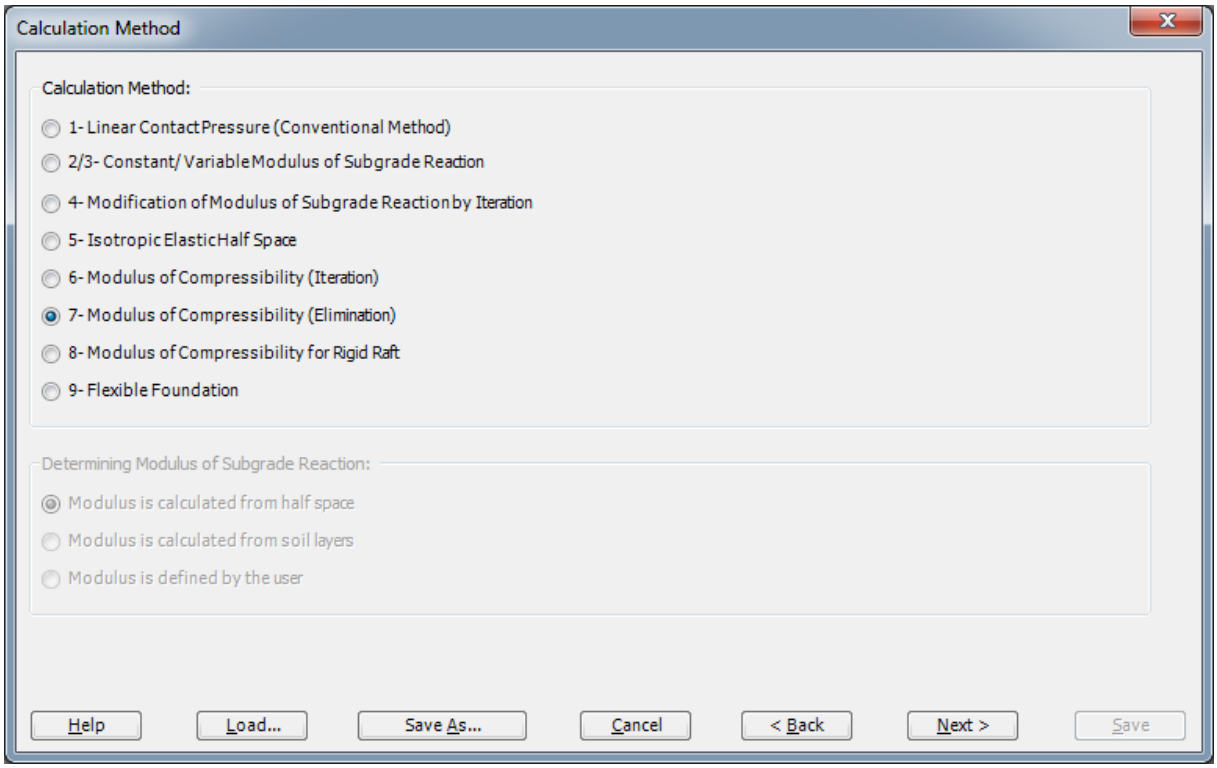


Figure 9.3 "Calculation Method" form

The next form is the "System Symmetry" (Figure 9.4). In this form choose "Unsymmetrical System" and then click "Next" button.

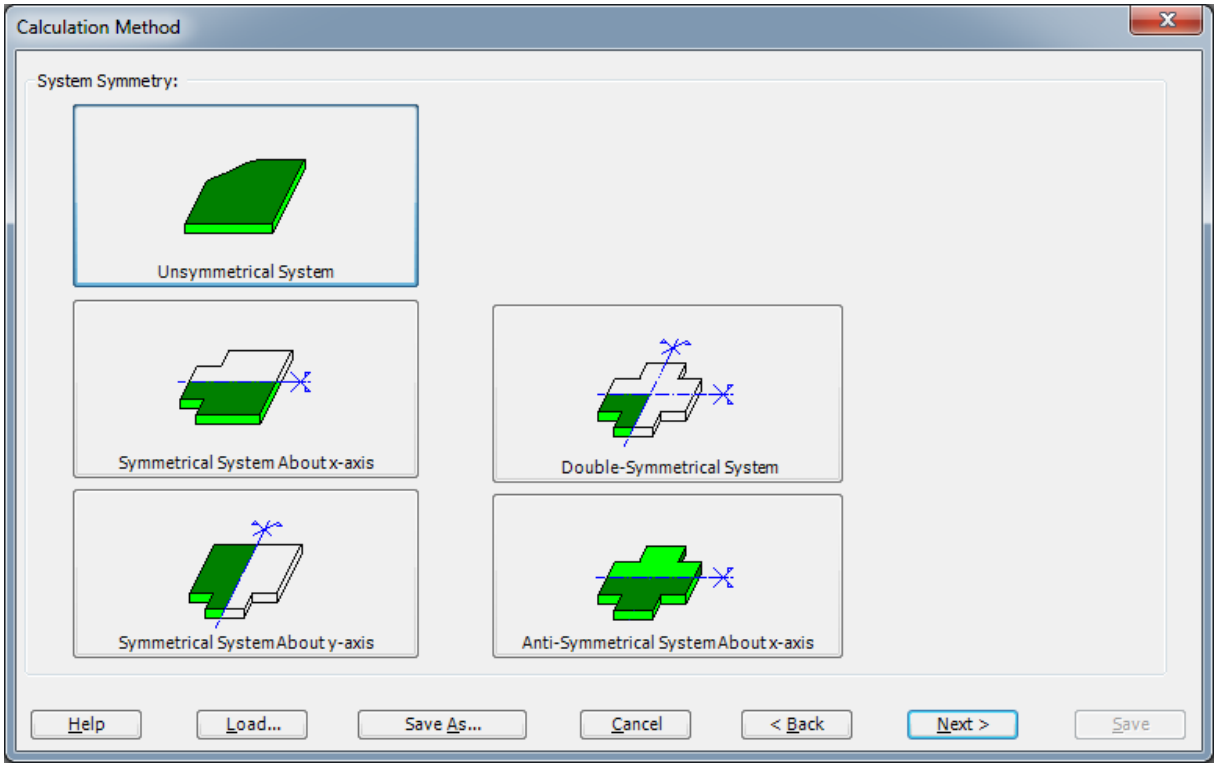


Figure 9.4 "System Symmetry" form

The last form of the wizard assistant contains the "Options" list, Figure 9.5. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other. There is not any requested choices, so click the "Save" button.

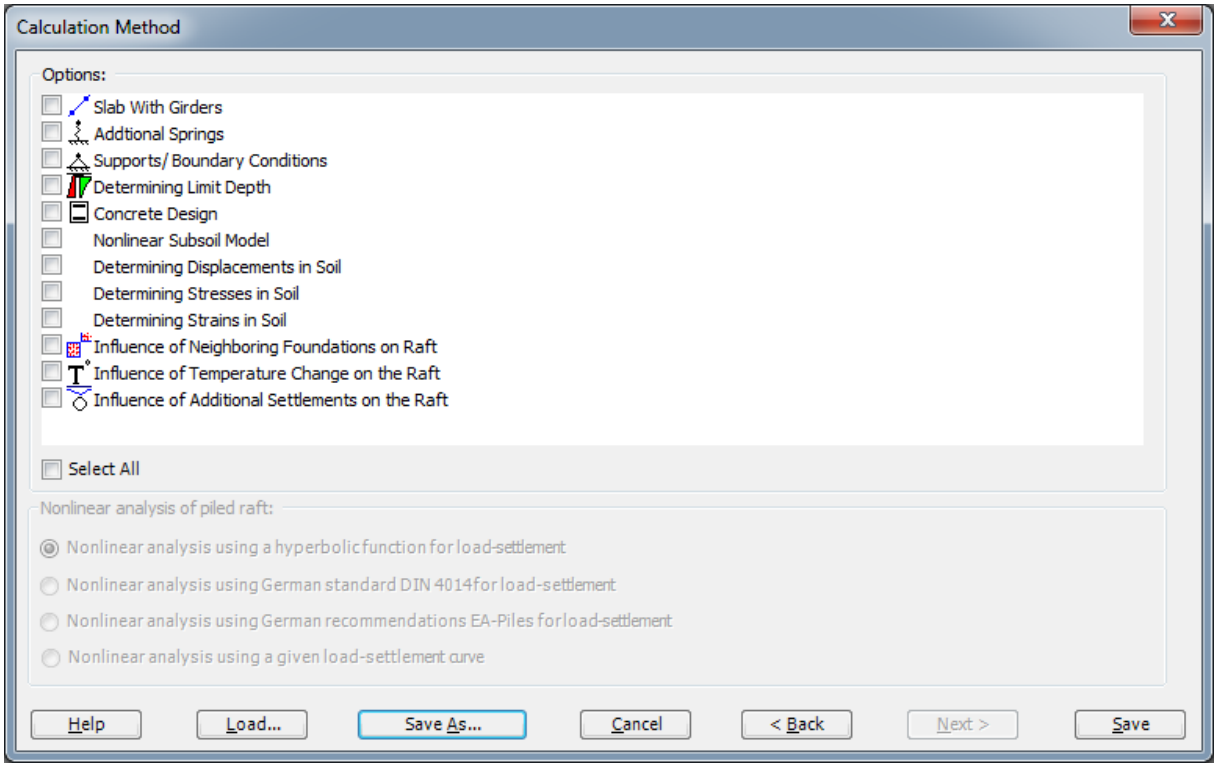


Figure 9.5 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 9.6.

In this dialog box

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

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

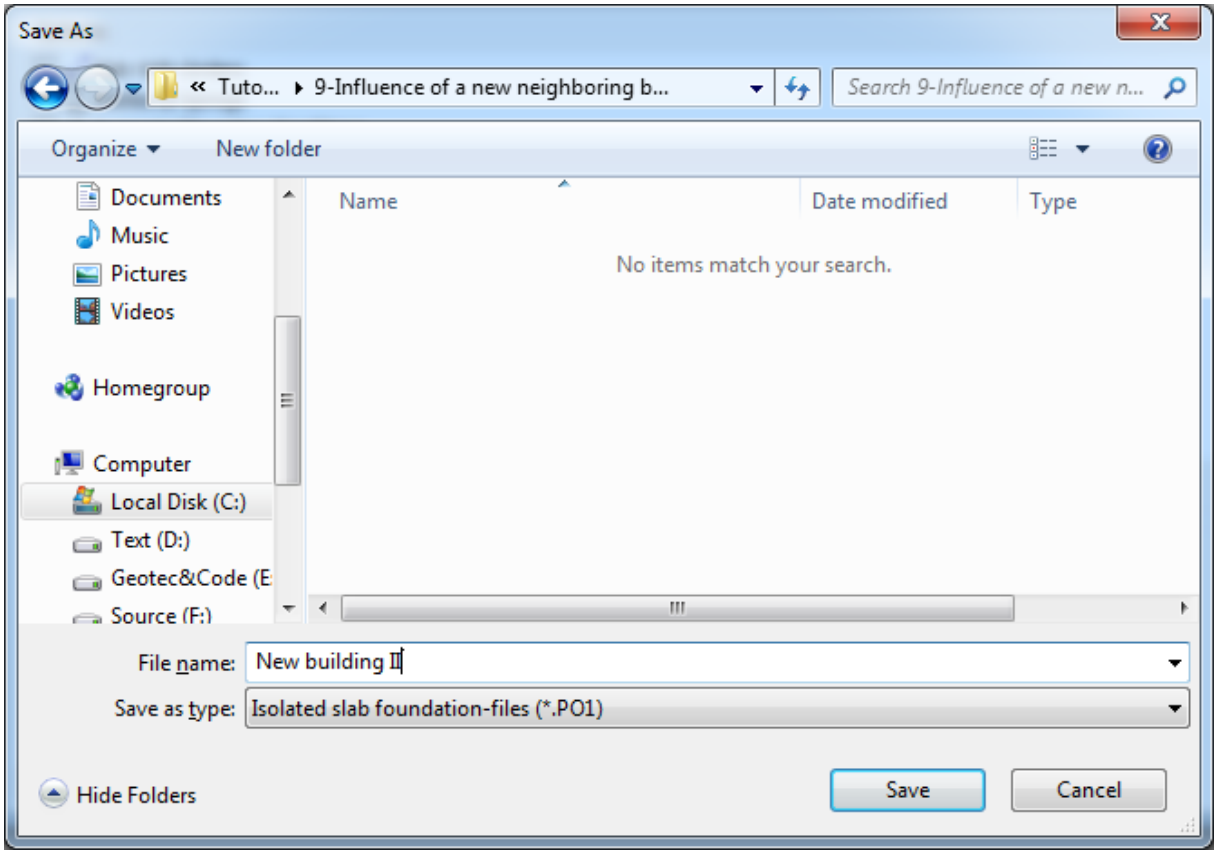


Figure 9.6 "Save as" dialog box

2.2 Project identification

To identify the project, choose "Project Identification" command from “Data” Tab. The dialog box in Figure 9.7 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Influence of a new neighboring building II on an old one I"
- Type the date of the project in the "Date" edit box
- Type "New Building II" in the "Project" edit box
- Click "Save" button

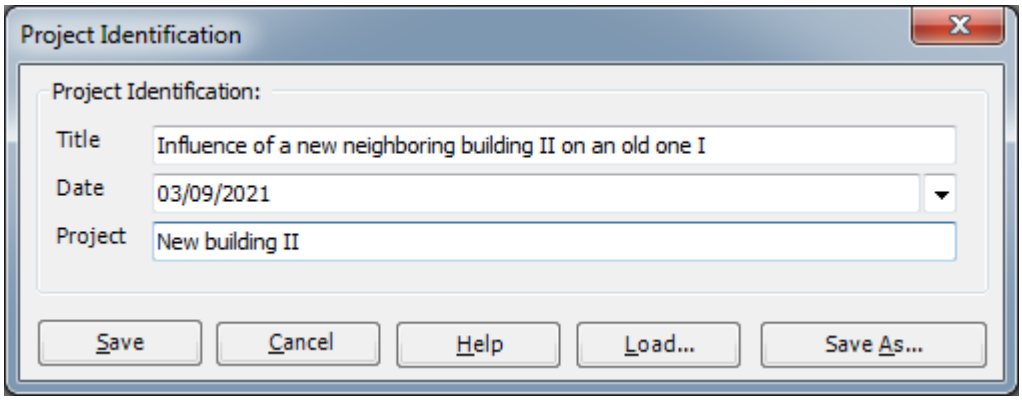


Figure 9.7 "Project Identification" dialog box

2.3 FE-Net data

Choose "FE-Net data" command from “Data” Tab. The "FE-Net Generation" wizard appears as shown in Figure 9.8. This wizard will guide you through the steps required to generate the FE-Net. 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 a square shape.

To generate the FE-Net

- In the "Slab Type" options, choose the square slab option
- Type 10.08 in the "Length/ width of square slab" edit box
- Click "Next" button to go to the next form

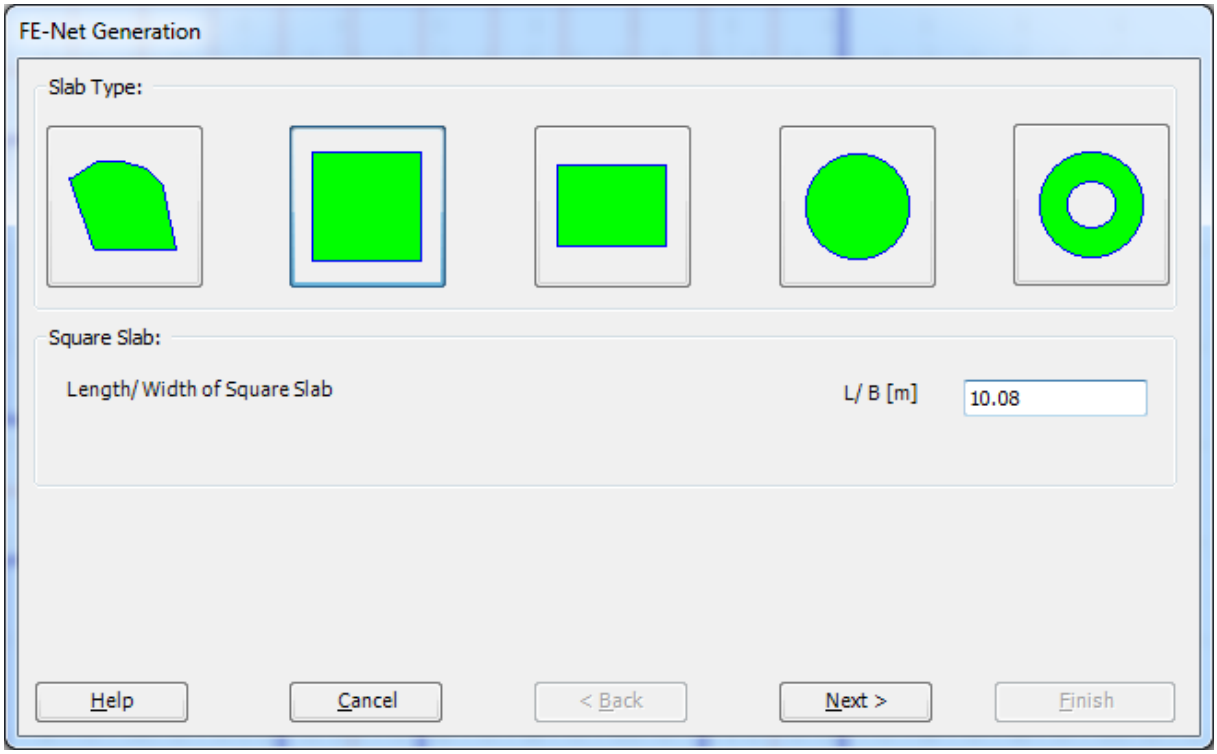


Figure 9.8 "Slab Type" form

After clicking "Next" in "Slab Type" form, the "Generation Type" form appears, Figure 9.9. *ELPLA* can deal with various type of generations with rectangular elements. Choose the rectangular elements option as generation type. Then click "Next" button to go to the next form.

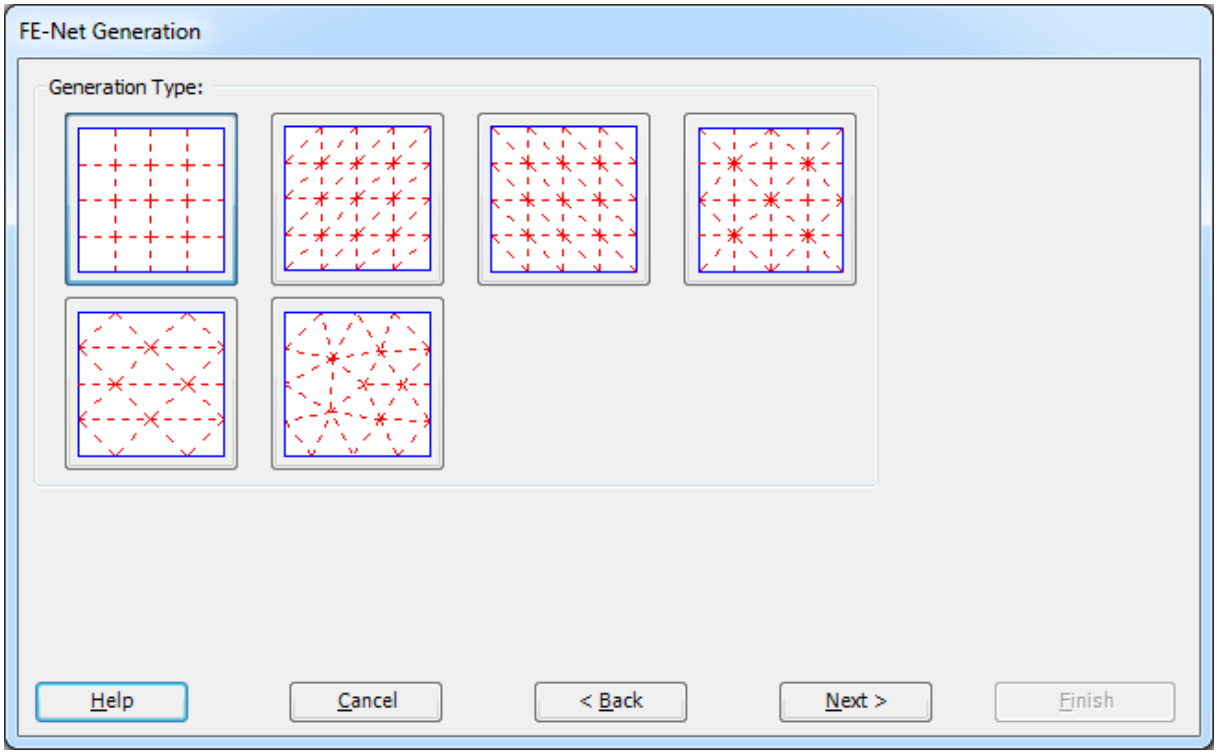


Figure 9.9 "Generation Type" form

The next form of the "FE-Net Generation" wizard is the "Grid Definition" dialogue box. In this dialogue box, the default values of constant element size appear, Figure 9.10.

In this "Grid definition" dialog box

- Type 14 in the "No. of grid intervals" edit box for grids in *x*-direction
- Type 14 in the "No. of grid intervals" edit box for grids in *y*-direction
- Type 0.72 in the "Grid interval D_x [m] " edit box for grids in *x*-direction
- Type 0.72 in the "Grid interval D_y [m] " edit box for grids in *y*-direction
- Click "Finish" button

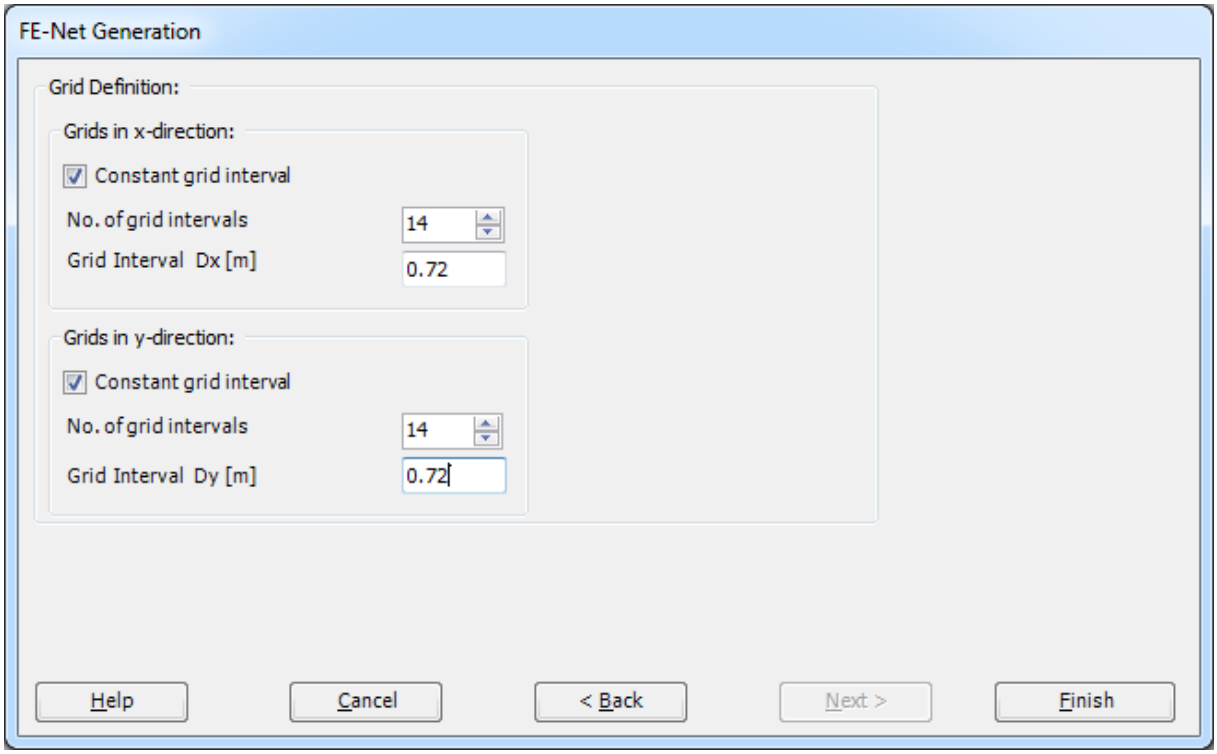


Figure 9.10 "Fe-Net Generation" form

Click "Finish" button in the "Generation Parameters" form. *ELPLA* will generate a suitable FE-Net for the square foundation of 10.08 [m] length with rectangular elements, which have equal areas. The following Window in Figure 9.11 appears with the generated net.

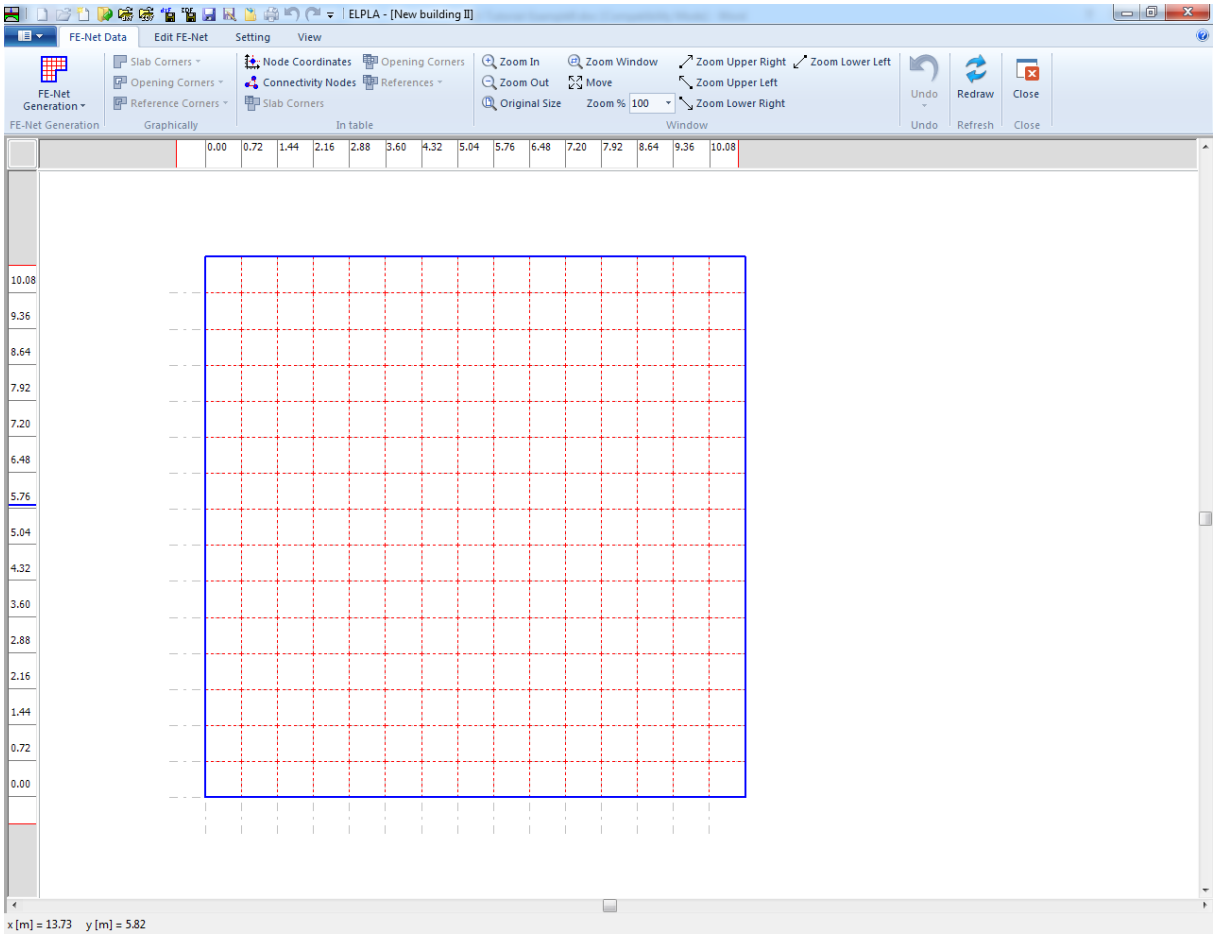


Figure 9.11 Imaginary net of a square area

Deleting nodes from the FE-Net

To select the unnecessary nodes, that are required to be removed from the net, first choose "Select Nodes" command from "Edit FE-Net" menu in Figure 9.11. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair. 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 9.12. A group of nodes can be selected by holding the left mouse button down at the corner of the region. Then, dragging the mouse until a rectangle encompasses the required group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

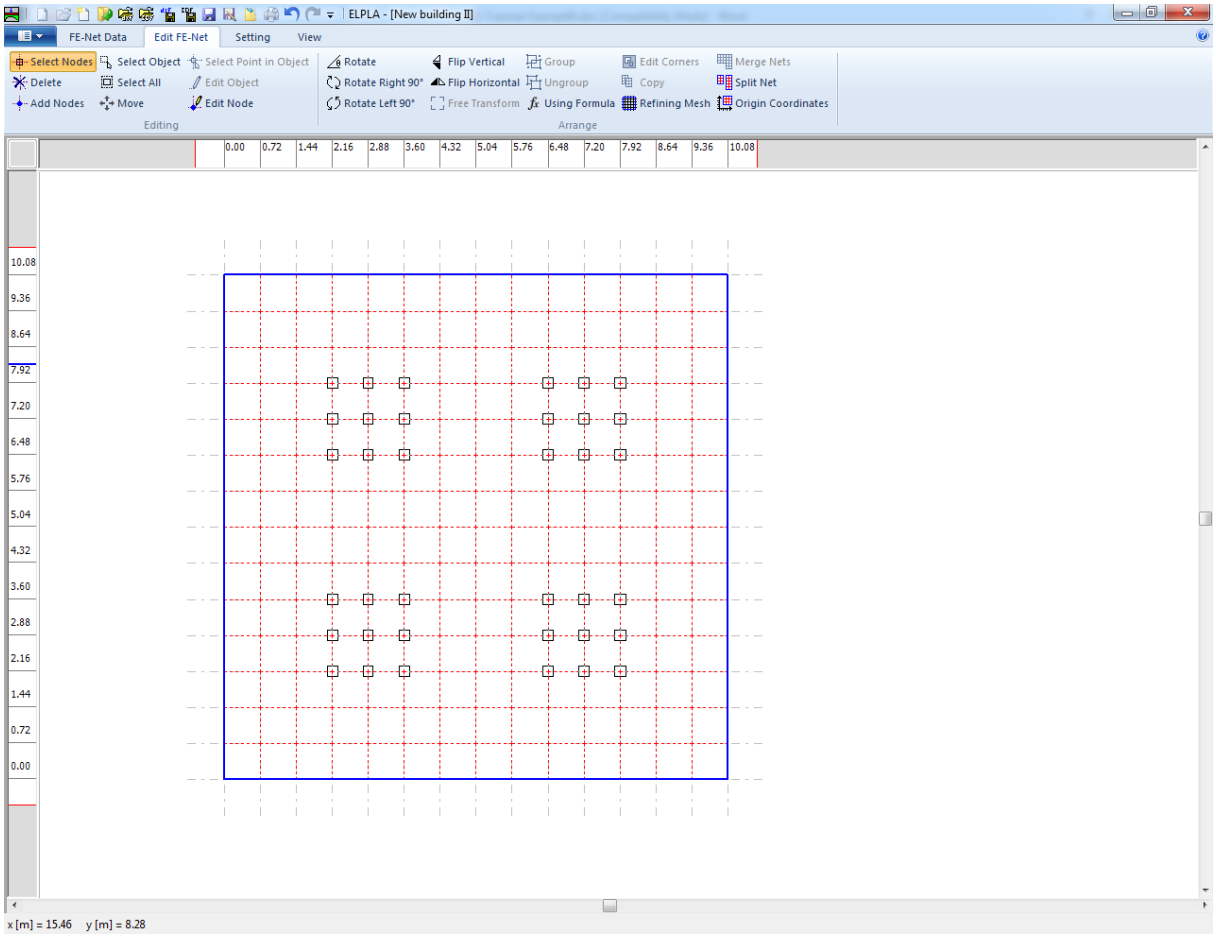


Figure 9.12 Generated FE-Net after selecting the unnecessary nodes

To remove the selected nodes, choose "Delete" command from "Edit FE-Net" menu. The action of this command is indicated in Figure 9.13. To leave the graphic mode, press "Esc" key.

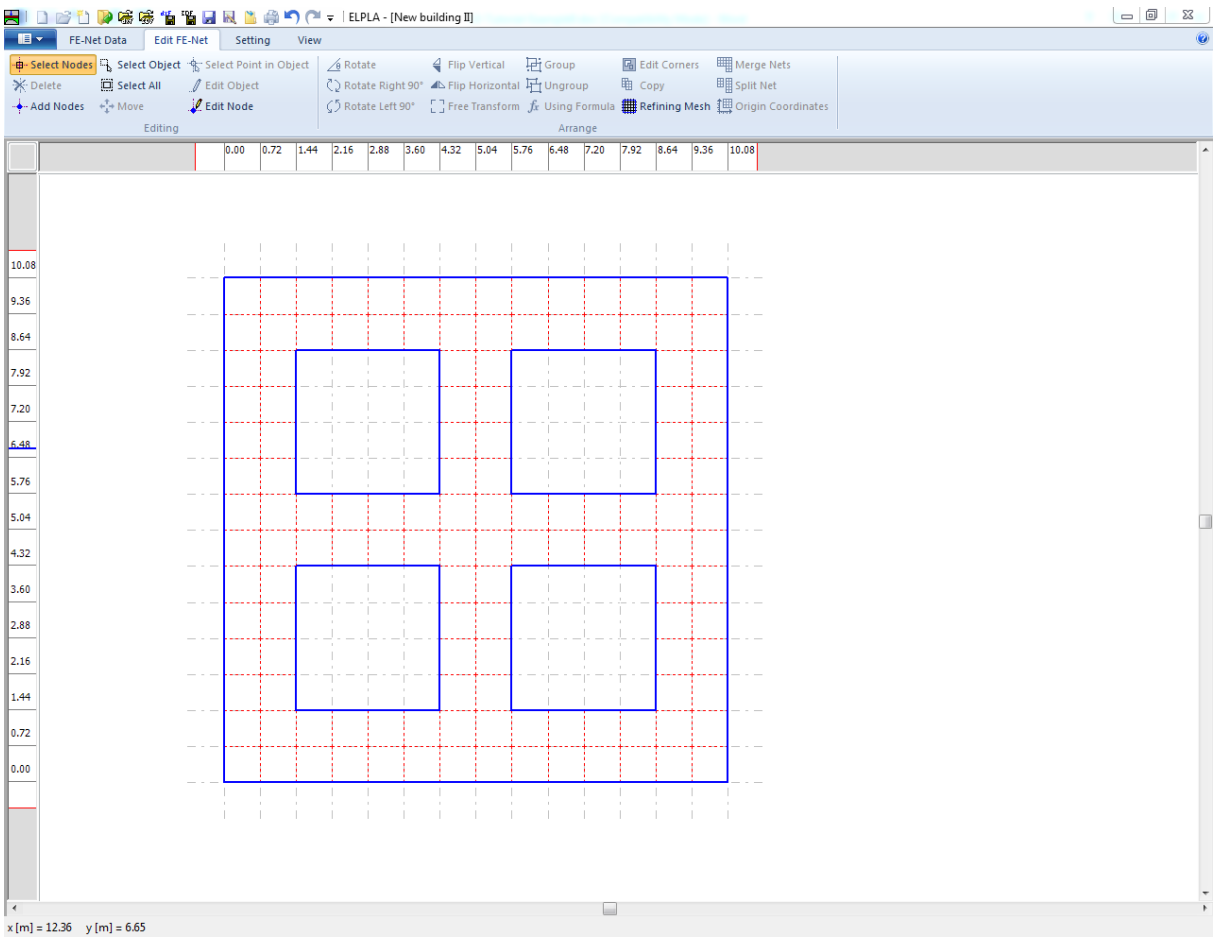


Figure 9.13 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save FE-Net" command from "File" menu in Figure 9.13 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 9.13 to close "FE-Net" Window and to return to *ELPLA* main window.

2.4 Soil properties

To define the soil properties, choose "Soil Properties" command from “Data” Tab. The following Window in Figure 9.14 appears with a default-boring log.

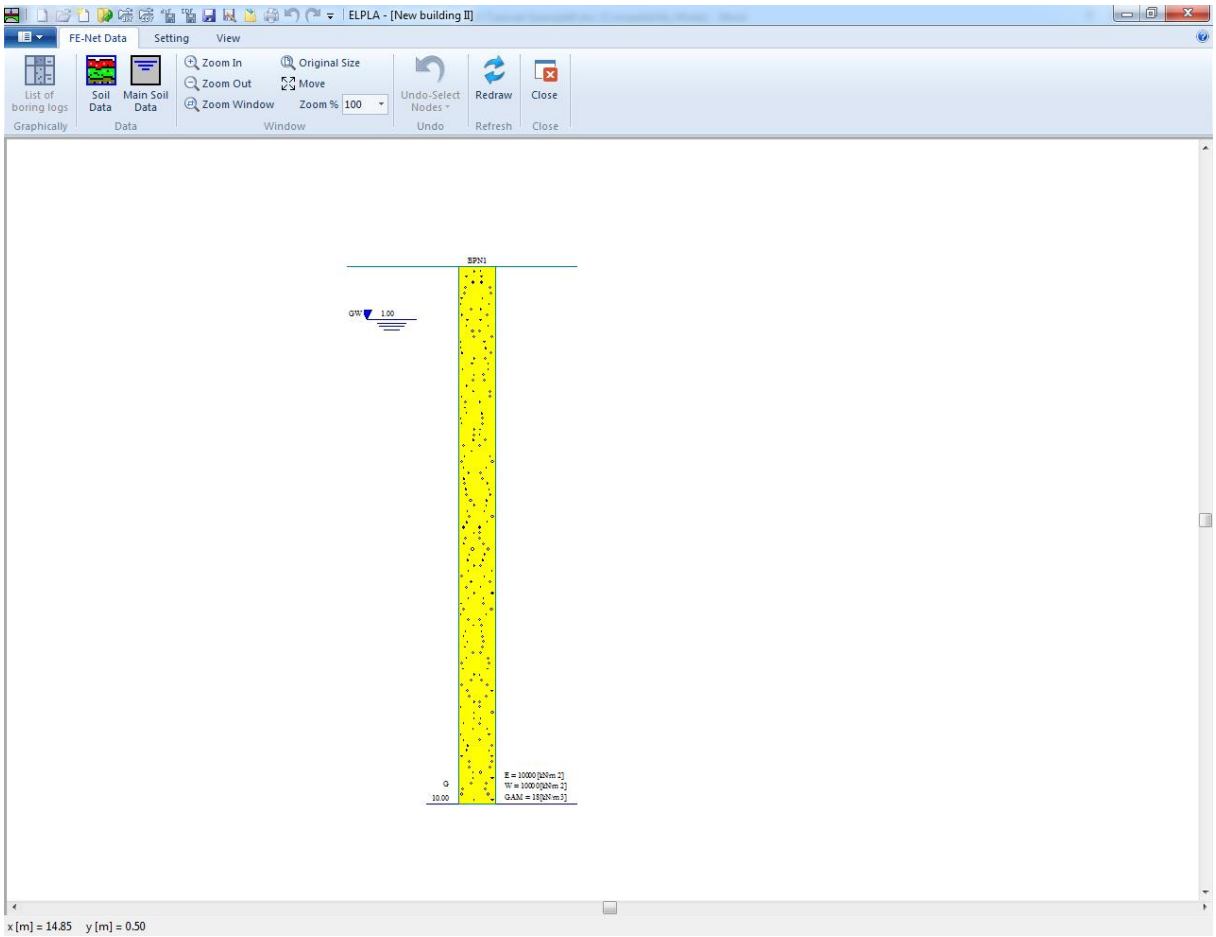


Figure 9.14 "Soil Properties" window with a default-boring log

Modifying data of boring log graphically

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

To enter the geotechnical data of the soil layer

- Double-click on the geotechnical data of the soil layer. The corresponding dialog box appears allowing you to modify the geotechnical data of the soil layer,Figure 9.15
- In the dialog group box "Geotechnical data of the layer" in Figure 9.15, define the geotechnical data of the soil layer as follows:

E_s	= 5000	[kN/m^2]
W_s	= 15000	[kN/m^2]
γ_s	= 18	[kN/m^3]

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 $\gamma_s \times d_f$. In the current example d_f = 1.5. 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.

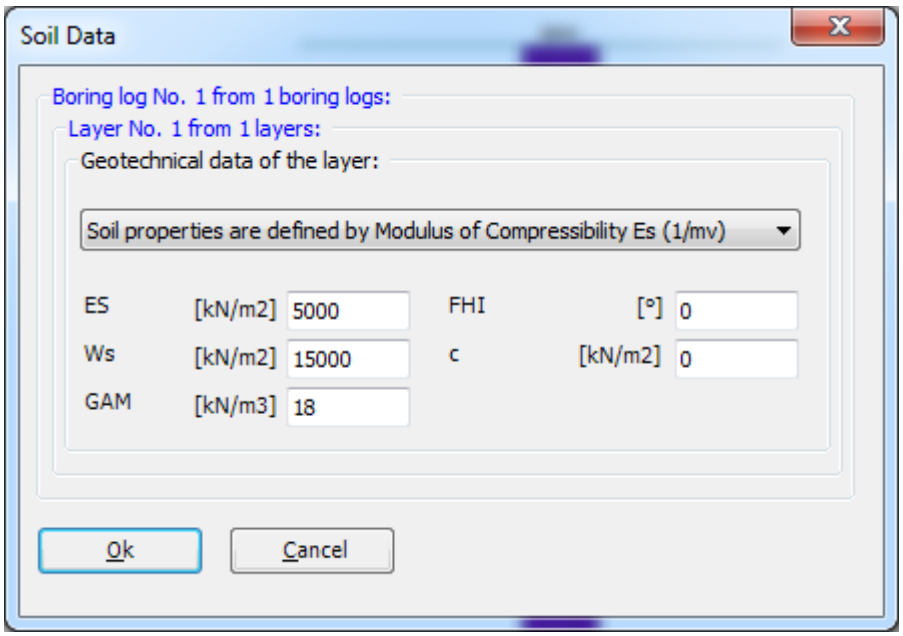


Figure 9.15 "Geotechnical data of the soil layer" dialog box

To define the soil type and color for the layer

- Double-click on the soil symbol of the soil layer. The corresponding dialog box appears allowing you to modify the soil symbols of that layer, Figure 9.16
- Select "T, Clay" as the soil type in the "Main soil type 1" combo box in the dialog group box "Soil and rock symbols" in Figure 9.16. The color of the clay and a short text "T" according to the German specification code DIN 4023 will be automatically created
- Click "OK" button

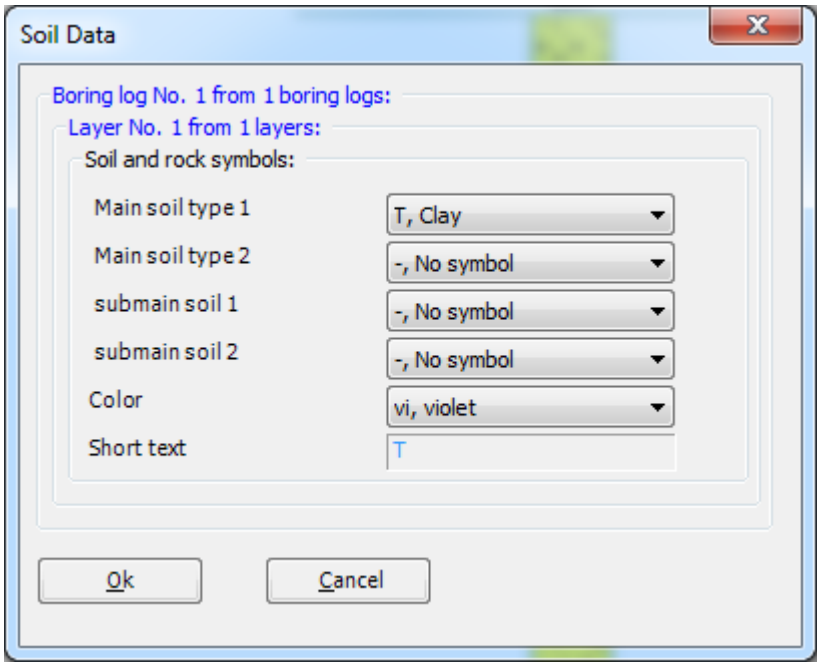


Figure 9.16 "Soil and rock symbols" dialog box

To modify the groundwater depth under the ground surface

- Double-click on the groundwater level. The corresponding edit box appears allowing you to modify the groundwater depth under the ground surface, Figure 9.17 To neglect the uplift pressure on the foundation, groundwater level is chosen at anywhere under the foundation basement
- Type 7.20 in the "Groundwater depth under the ground surface" edit box
- Click "OK" button

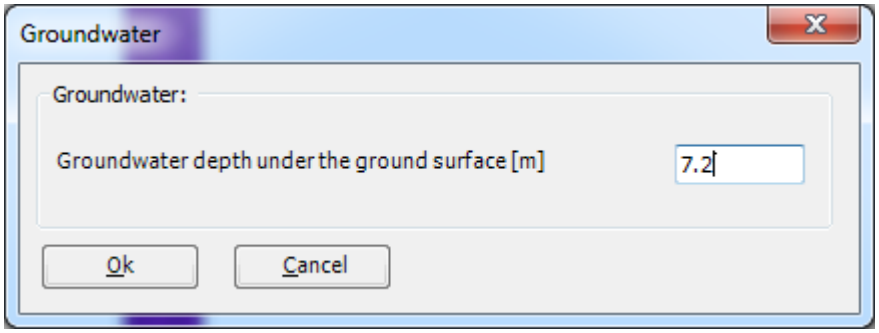


Figure 9.17 "Groundwater depth under the ground surface" edit box

To modify a layer depth

- Double-click on the layer depth. The corresponding edit box appears allowing you to modify the layer depth under the ground surface, Figure 9.18
- Type 7.20 in the "Layer depth under the ground surface" edit box
- Click "OK" button

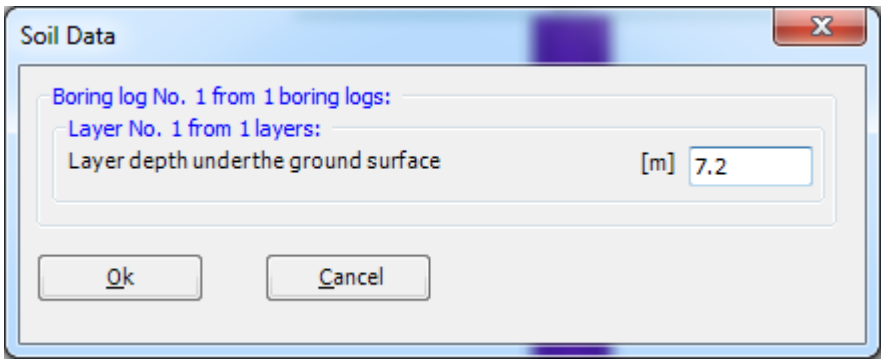


Figure 9.18 "Layer depth under the ground surface" edit box

To modify the label of a boring log

- Double-click on the label of the boring log. The corresponding edit box appears allowing you to modify the label of the boring log, Figure 9.19
- Type "B1" in the edit box of Figure 9.19
- Press "Enter" key to consider the text

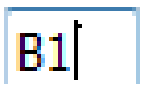


Figure 9.19 "Label of the boring log" edit box

To enter the main soil data for the layer

- Choose "Main Soil Data" from “Data” menu in Figure 9.20. The following dialog group box in Figure 9.20 appears
- In this dialog group box, enter the settlement reduction factor α and the groundwater depth under the ground surface G_w [m] as indicated in Figure 9.20 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 9.20

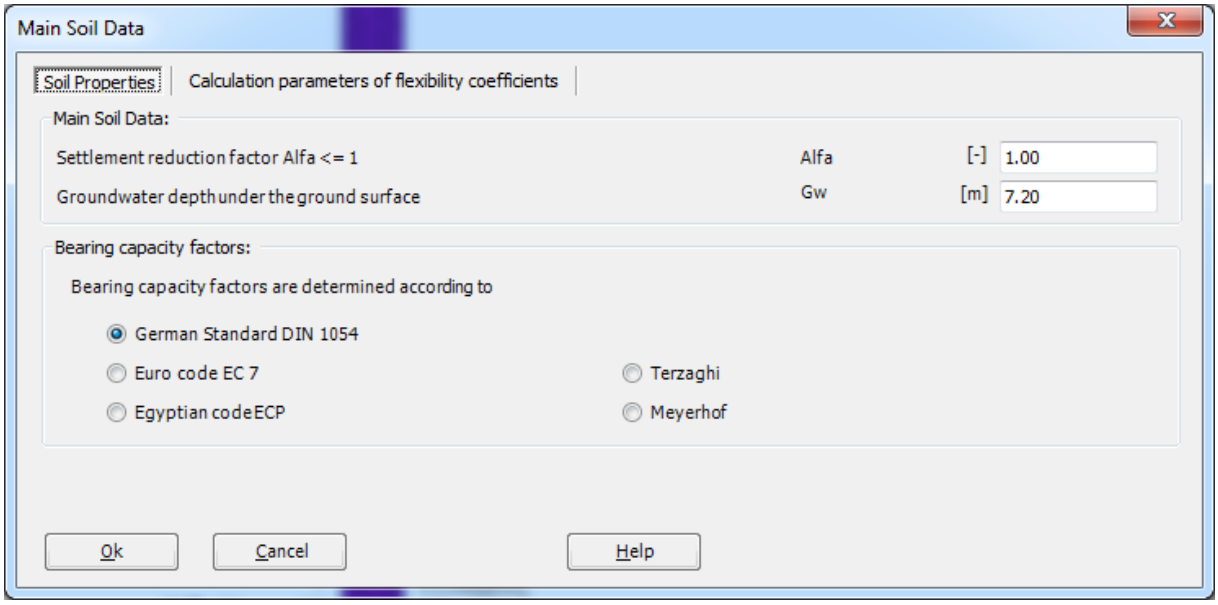


Figure 9.20 "Main Soil Data" dialog group box

After the user has completed the definition of all soil properties and parameters, the screen should look like the following Figure 9.21.

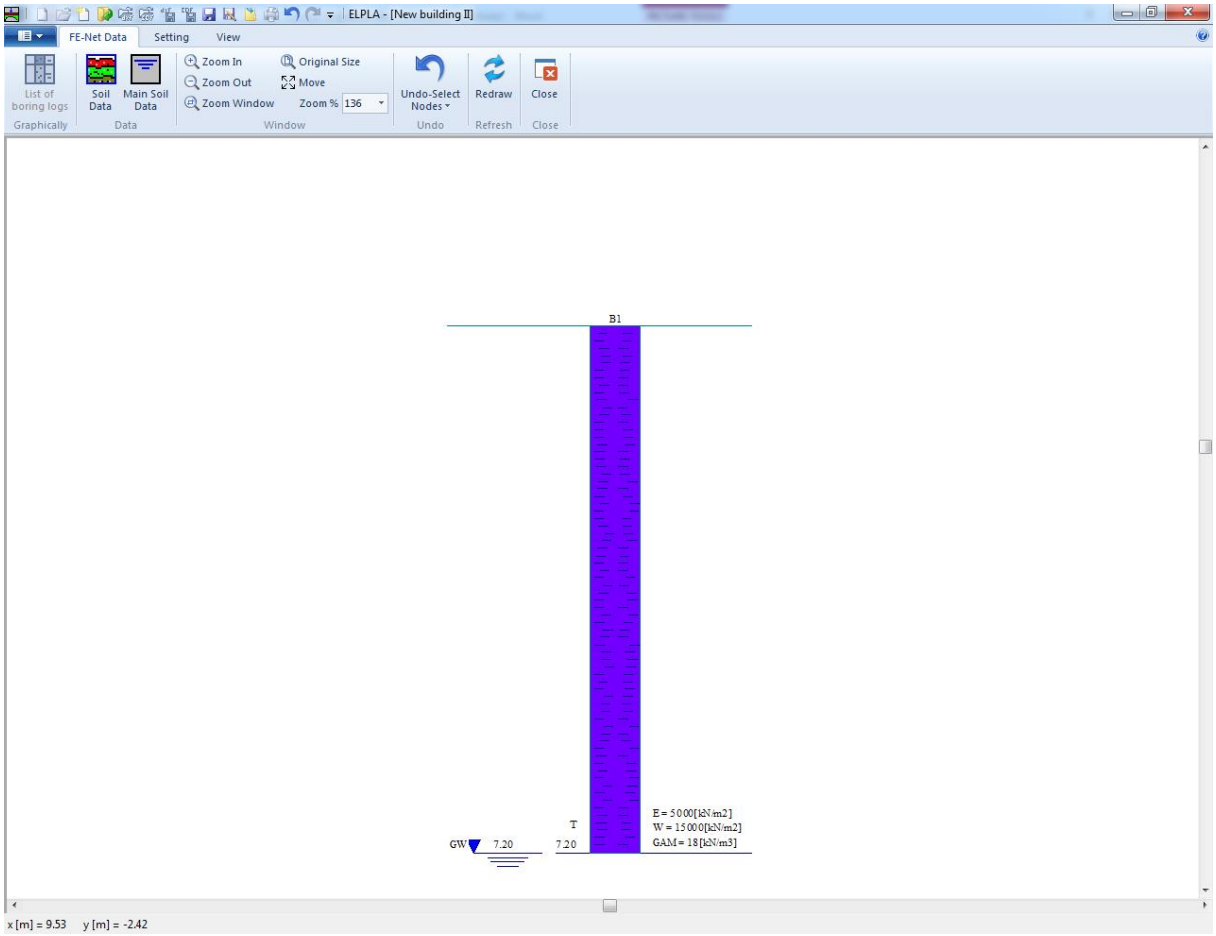


Figure 9.21 Boring log

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

- Choose "Save" command from "File" menu in Figure 9.21 to save the data of boring log
- Choose "Close" command from "File" menu in Figure 9.21 to close the "Soil Properties" Window and to return to *ELPLA* main window.

2.5 Foundation properties

To define the foundation properties, choose "Foundation Properties" command from “Data” Tab. The following Window in Figure 9.22 appears with default foundation properties. The data of foundation properties for the current example, which are required to define, are foundation material, foundation thickness and foundation level. 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.

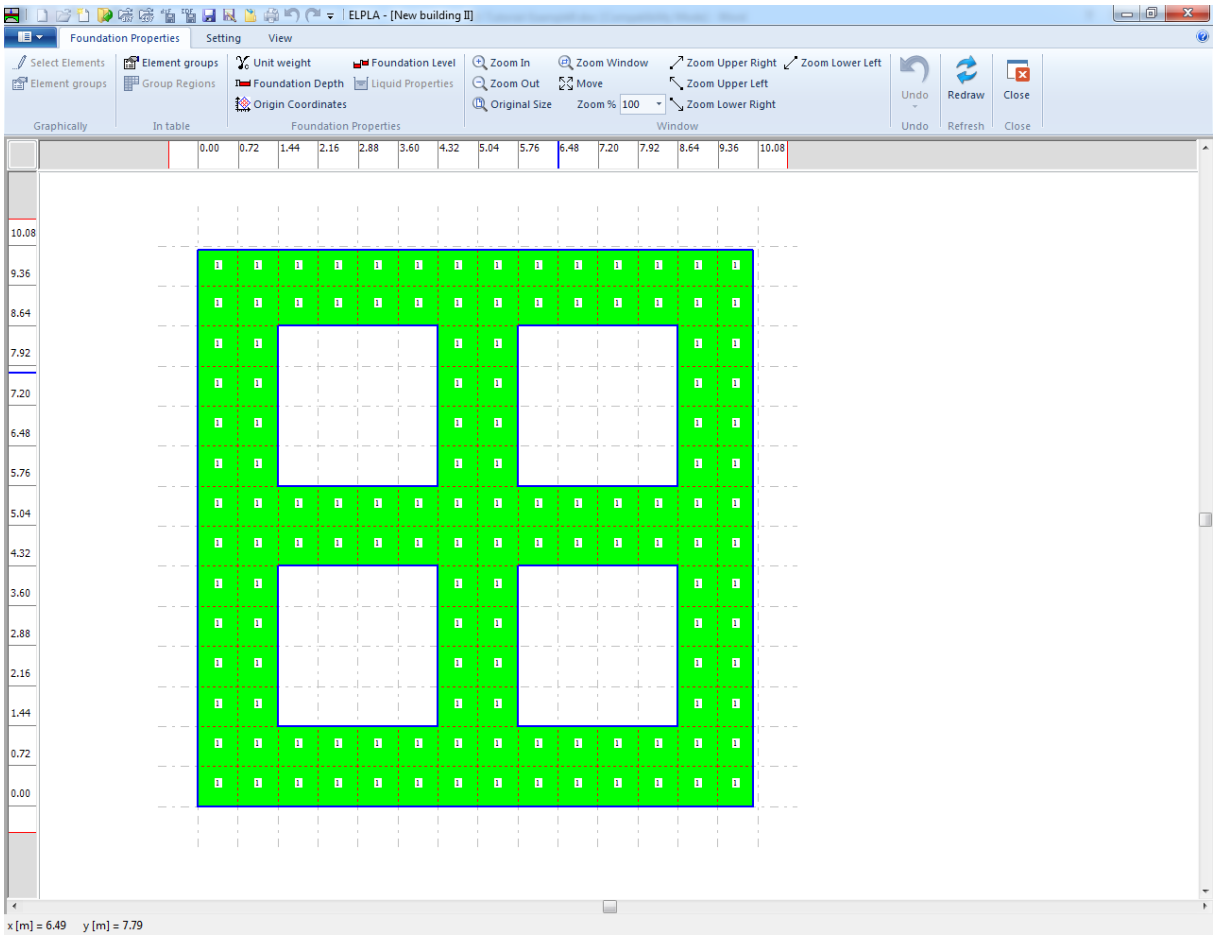


Figure 9.22 "Foundation Properties" Window

To enter the foundation material and thickness

- Choose "Element groups" command from "In table" menu Figure 9.22. The following list box in Figure 9.23 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 9.23, enter E-Modulus of the foundation, *Poisson's* ratio of the foundation and foundation thickness

- Click "OK" button

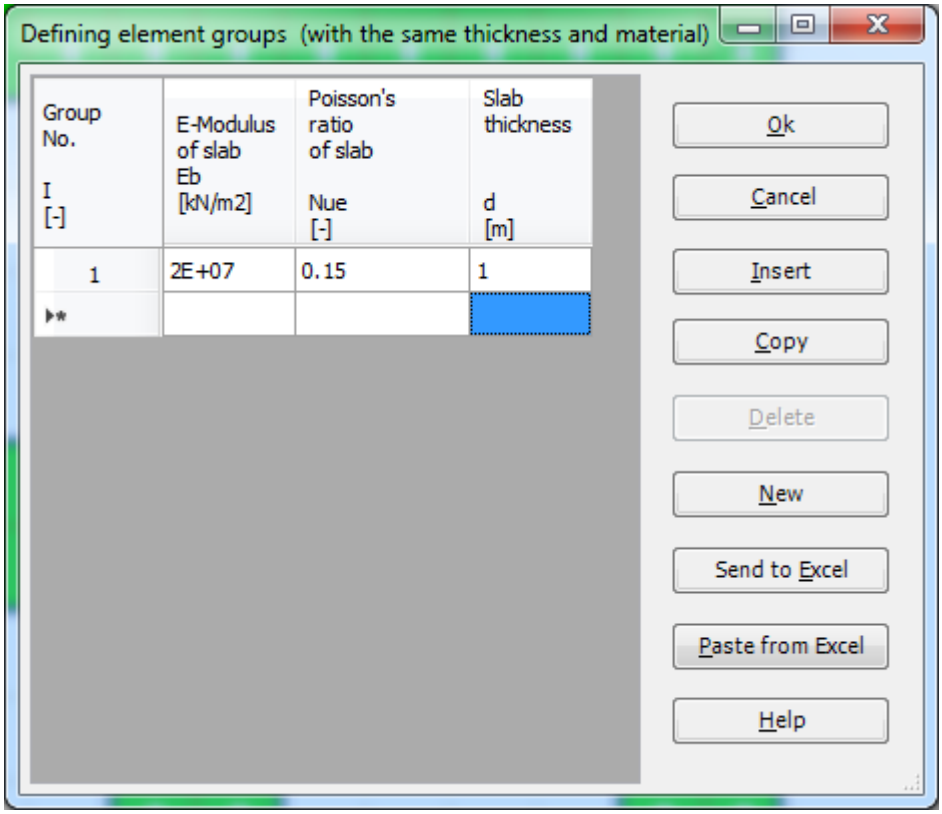


Figure 9.23 "Defining element groups" list box

To enter the unit weight of the foundation

- Choose "Unit weight" command from "Foundation properties" menu in the window of Figure 9.22. The following dialog box in Figure 9.24 with a default unit weight of 25 [kN/m^3] appears. To neglect the self-weight of the foundation in the analysis, type 0 in the edit box "Unit weight"
- Click "OK" button

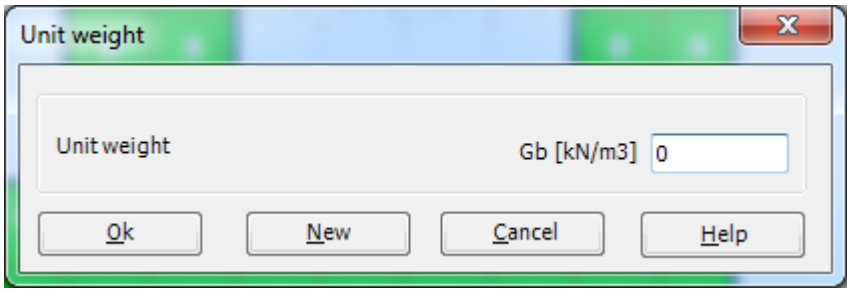


Figure 9.24 "Unit weight of the foundation" dialog box

-

To enter the foundation level

- Choose "Foundation Depth" command from "Foundation properties" menu Figure 9.22.
  The following dialog box in Figure 9.25 appears
- Type 1.5 in the "Foundation depth under ground surface (a)/ (b)" edit box
- Click "OK" button

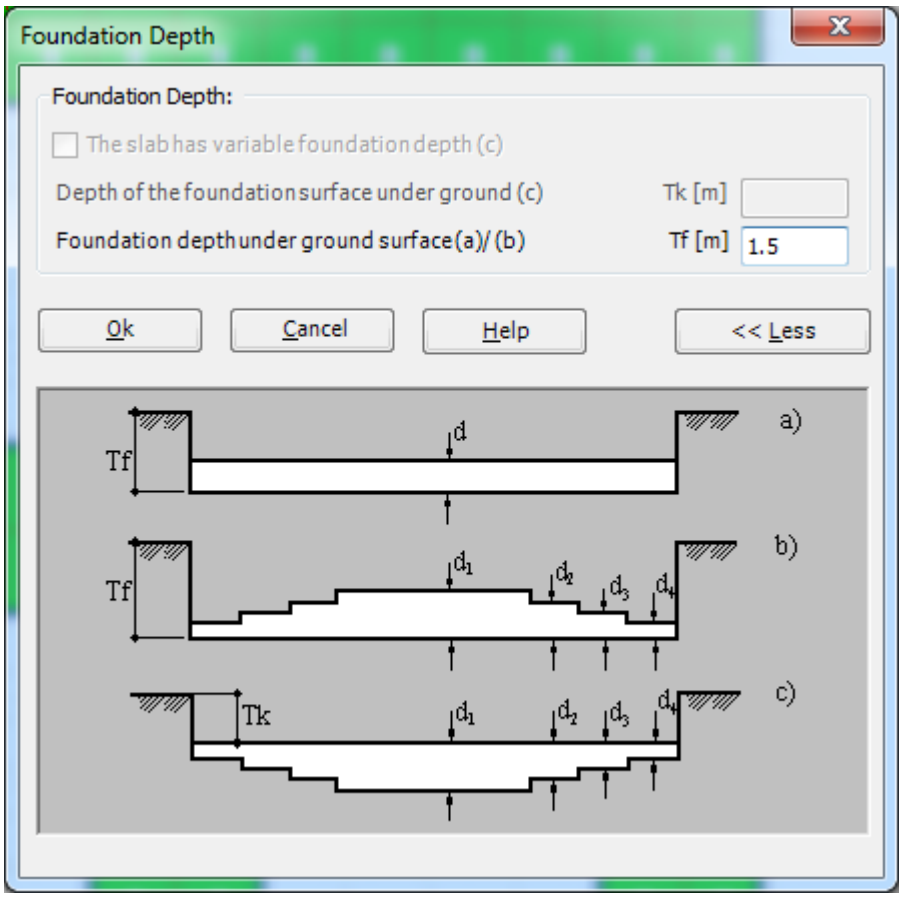


Figure 9.25 "Foundation Depth" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 9.22 to save
  the foundation properties
- Choose "Close" command from "File" menu in Figure 9.22 to close the "Foundation properties" Window and to return to *ELPLA* main window.

2.6 Loads

To define the loads, choose "Loads" command from “Data” Tab The following Window in Figure 9.26 appears.

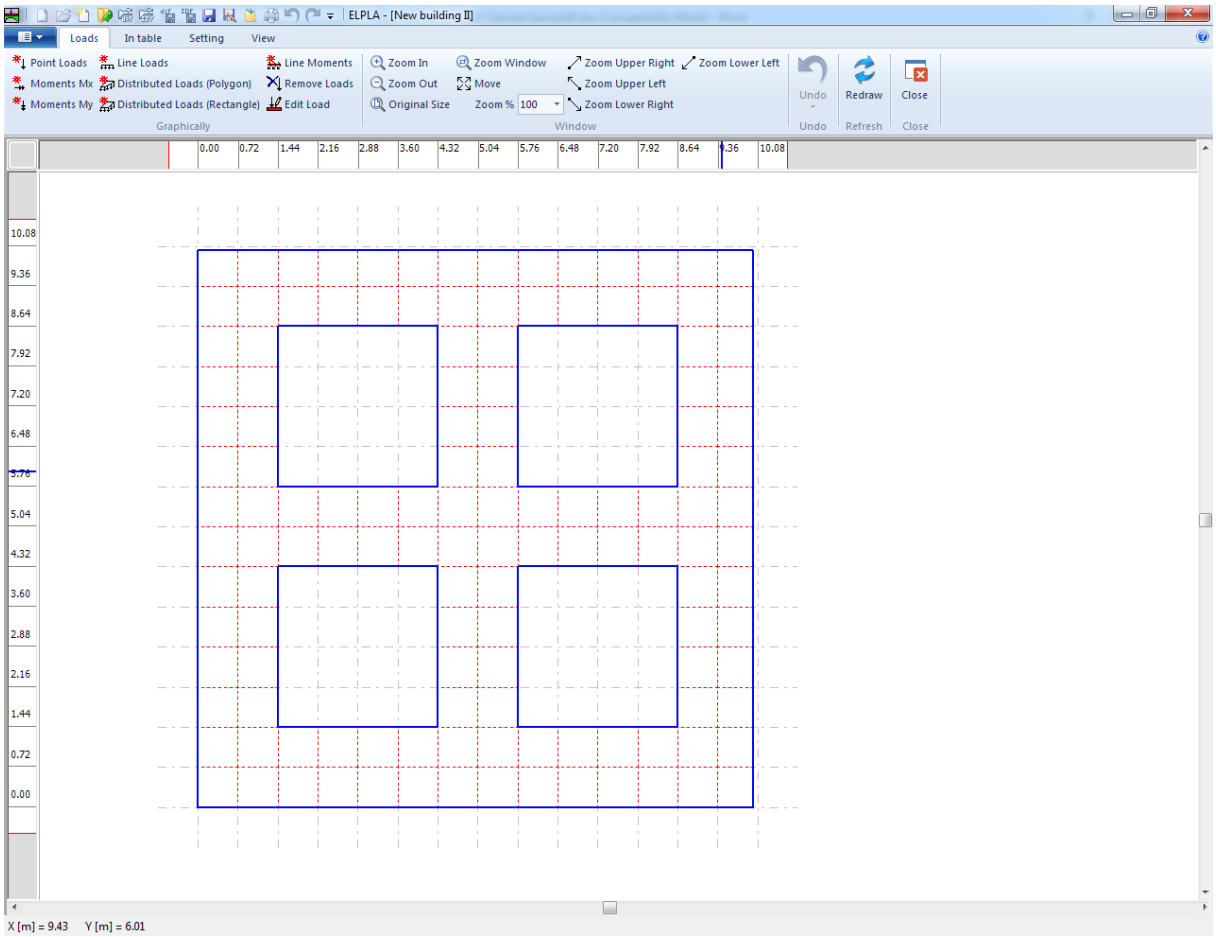
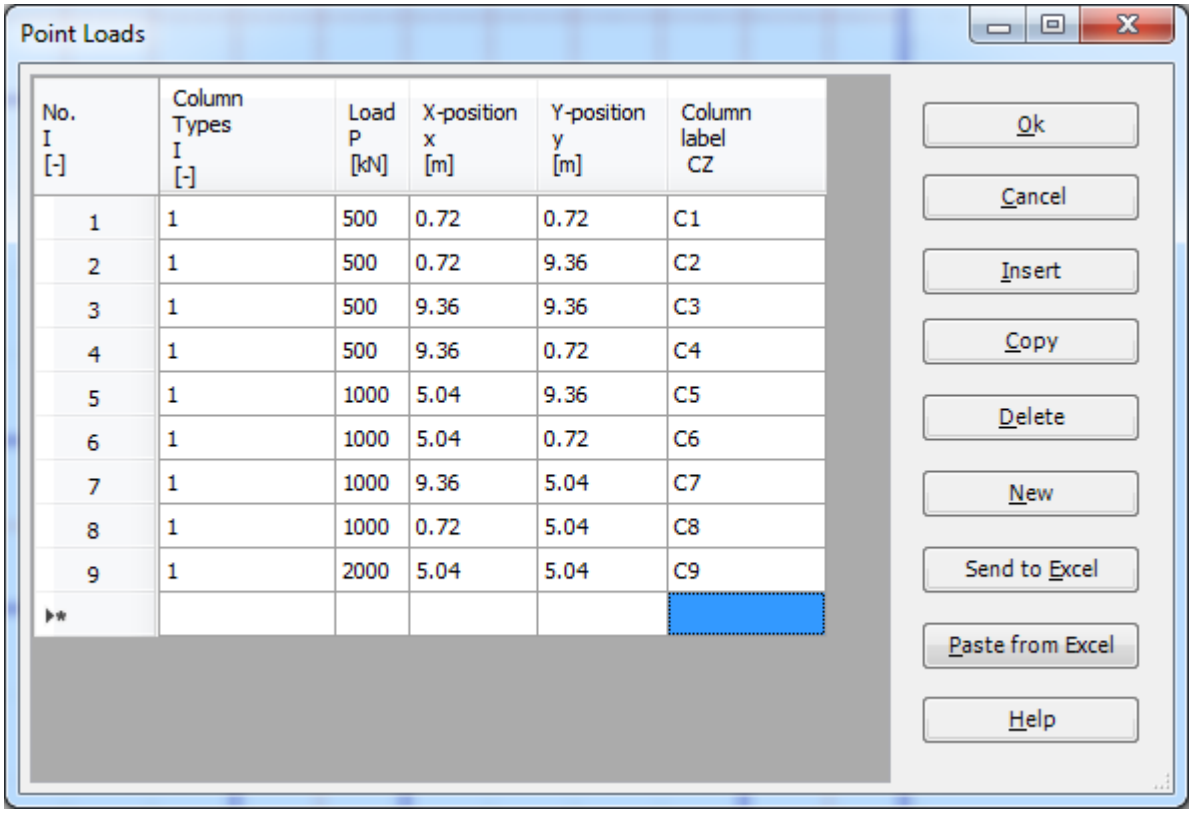


Figure 9.26 "Loads" Window

To enter loads

- Choose "Point Loads" command from "In Table" menu in the window of Figure 9.26. The following list box in Figure 9.27 appears
- Enter the external point loads *P* [kN] and their corresponding coordinates (*x*, *y*) in the list box of Figure 9.27. 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]	Column label CZ
1	1	500	0.72	0.72	C1
2	1	500	0.72	9.36	C2
3	1	500	9.36	9.36	C3
4	1	500	9.36	0.72	C4
5	1	1000	5.04	9.36	C5
6	1	1000	5.04	0.72	C6
7	1	1000	9.36	5.04	C7
8	1	1000	0.72	5.04	C8
9	1	2000	5.04	5.04	C9
▸*					

Figure 9.27 "Point Loads" list box

After you have completed the definition of all load data, the screen should look like the following Figure 9.28.

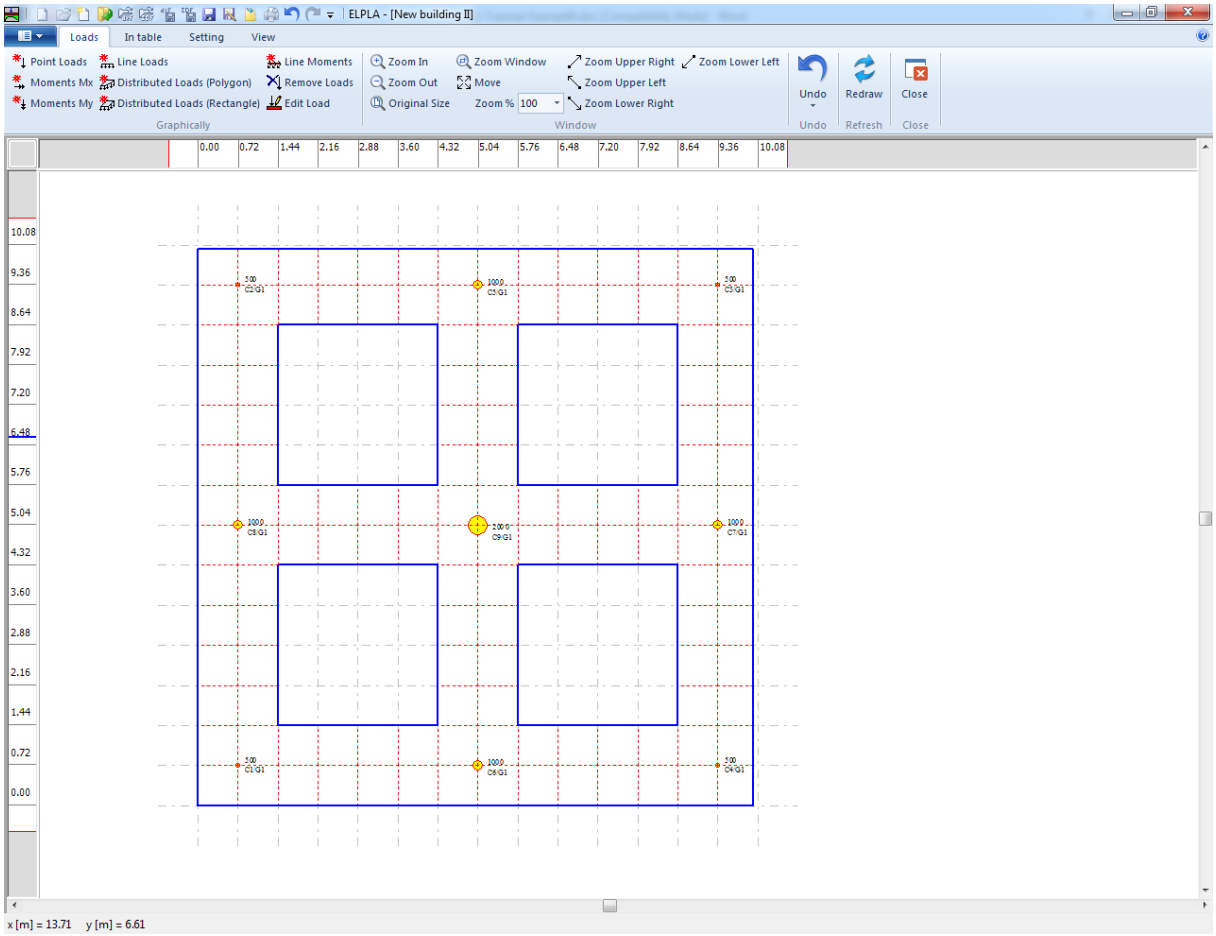


Figure 9.28 Loads

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

- Choose "Save " command from "File" menu in Figure 9.26 to save the load data
- Choose "Close" command from "File" menu in Figure 9.26 to close the "Loads" Window and return to *ELPLA* main window

The project creation of the New Building II is now completed.

3 Creating the project of the Old Building I

The data of the two foundations are quite similar except the origin coordinates of the global system, which are chosen to be (0.0, 0.0) and (10.28, 0.0) for buildings I and II respectively. Project identification is entered so that you can distinguish between the two projects. The data of Old Building I are created by first saving the data of the New Building II 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 as" command from "File" menu of *ELPLA*. The following "Save as" dialog box appears, Figure 9.29

In this dialog box

- Type a file name for the project of the Old Building I in the file name edit box. For example, type "Old Building I"
- Click "Save" button

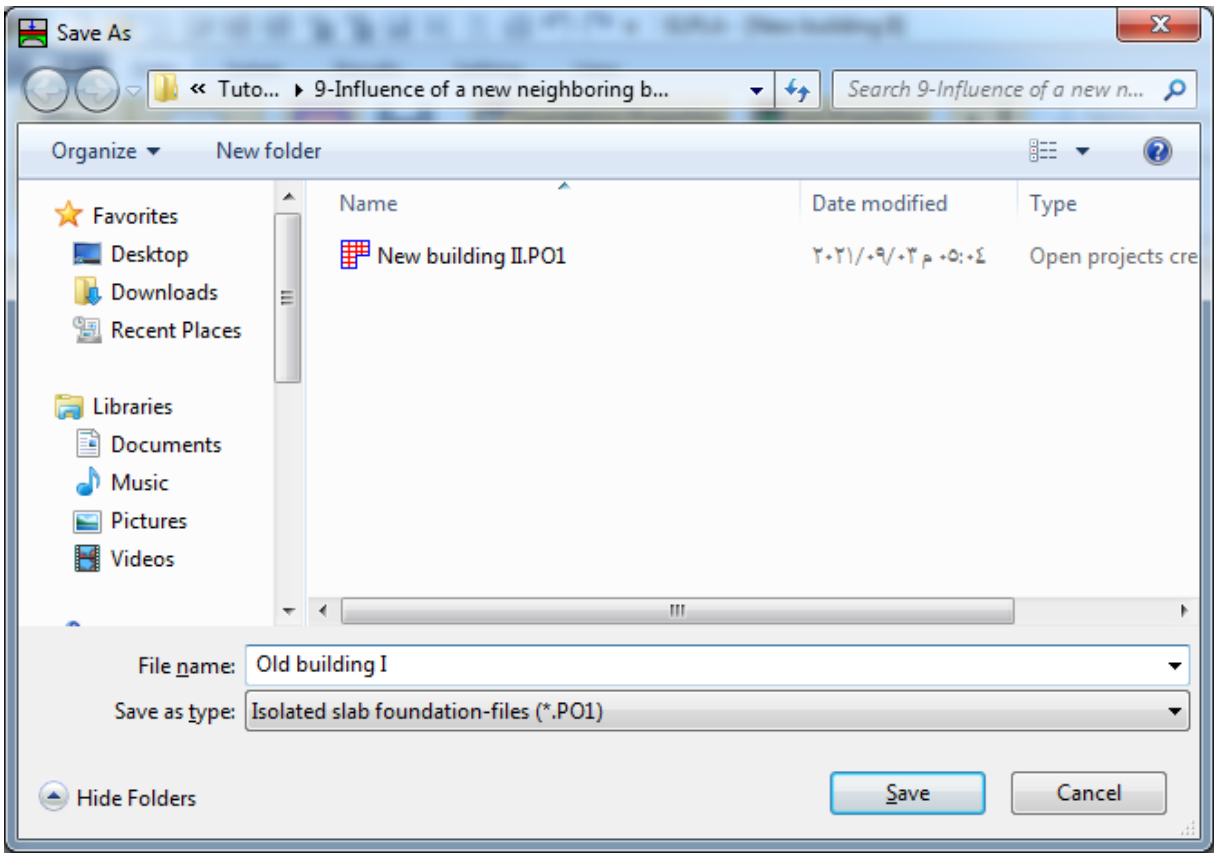


Figure 9.29 "Save as" dialog box

3.1 Modifying the calculation methods options

To add the option of neighboring foundations, do the following steps:

- Choose "Calculation Method" command from “Data” Tab
- The first form of the wizard "Calculation Method" is the "Analysis Type" form, Figure 9.2
- In this form, click "Next" button to go to the next page
- After clicking "Next" button, the "Calculation Method" form appears, Figure 9.3. Click "Next" button to go to the next form
- The next form is the "System Symmetry", Figure 9.4. In this form click "Next" button
- The last form of the wizard assistant contains the "Options" list, Figure 9.30. In this list, check the option "Influence of neighboring foundations on the slab", then click the "Save" button
- *ELPLA* will activate the “Data” Tab. Also the file name of the current project [Old Building I] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

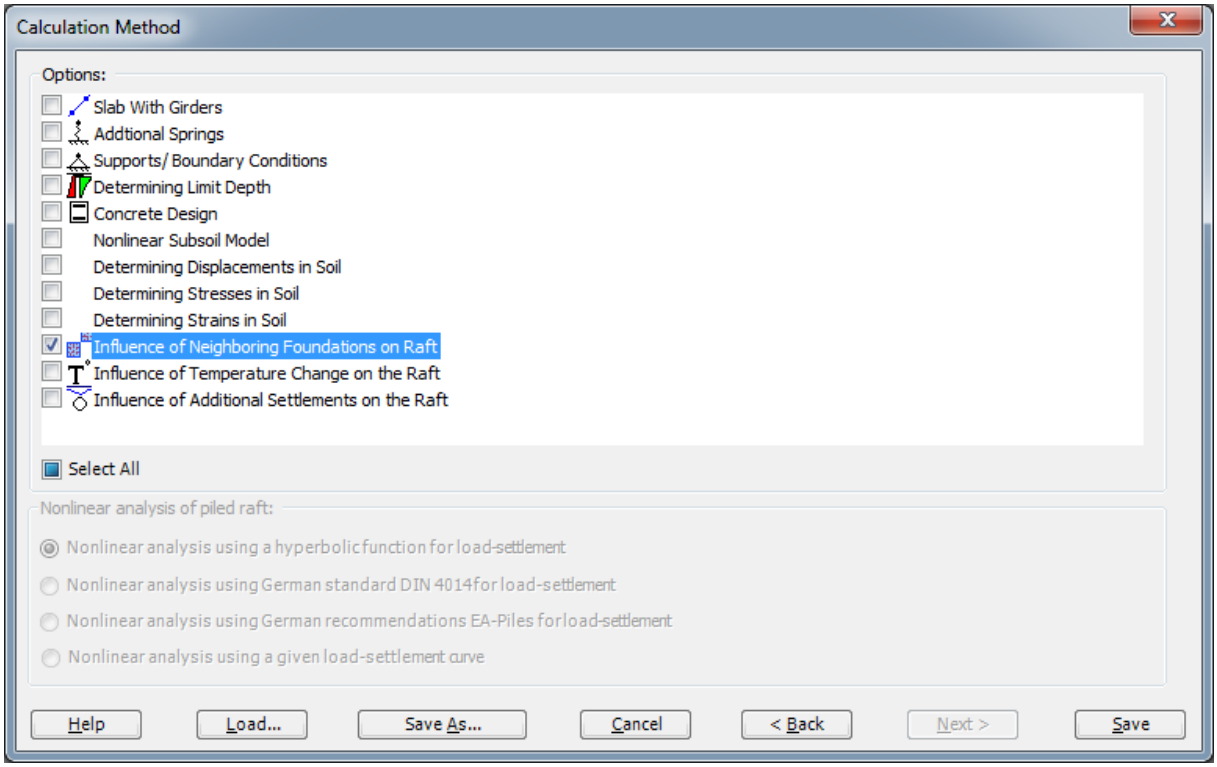


Figure 9.30 "Options" list

3.2 Modifying the project identification

To modify the project identification of building II, choose "Project Identification" command from “Data” Tab. The dialog box in Figure 9.31 appears. In this dialog, type "Old building I" in the "Project" edit box then click "Save" button.

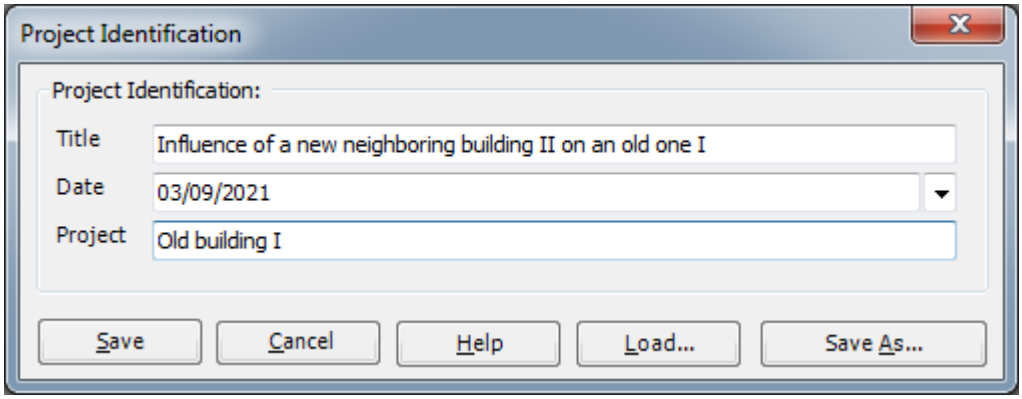


Figure 9.31 "Project Identification" dialog box

3.3 Modifying origin coordinates

To modify the origin coordinate of building II, choose "Foundation Properties" command from “Data” Tab of *ELPLA*. The following Window in Figure 9.32 appears.

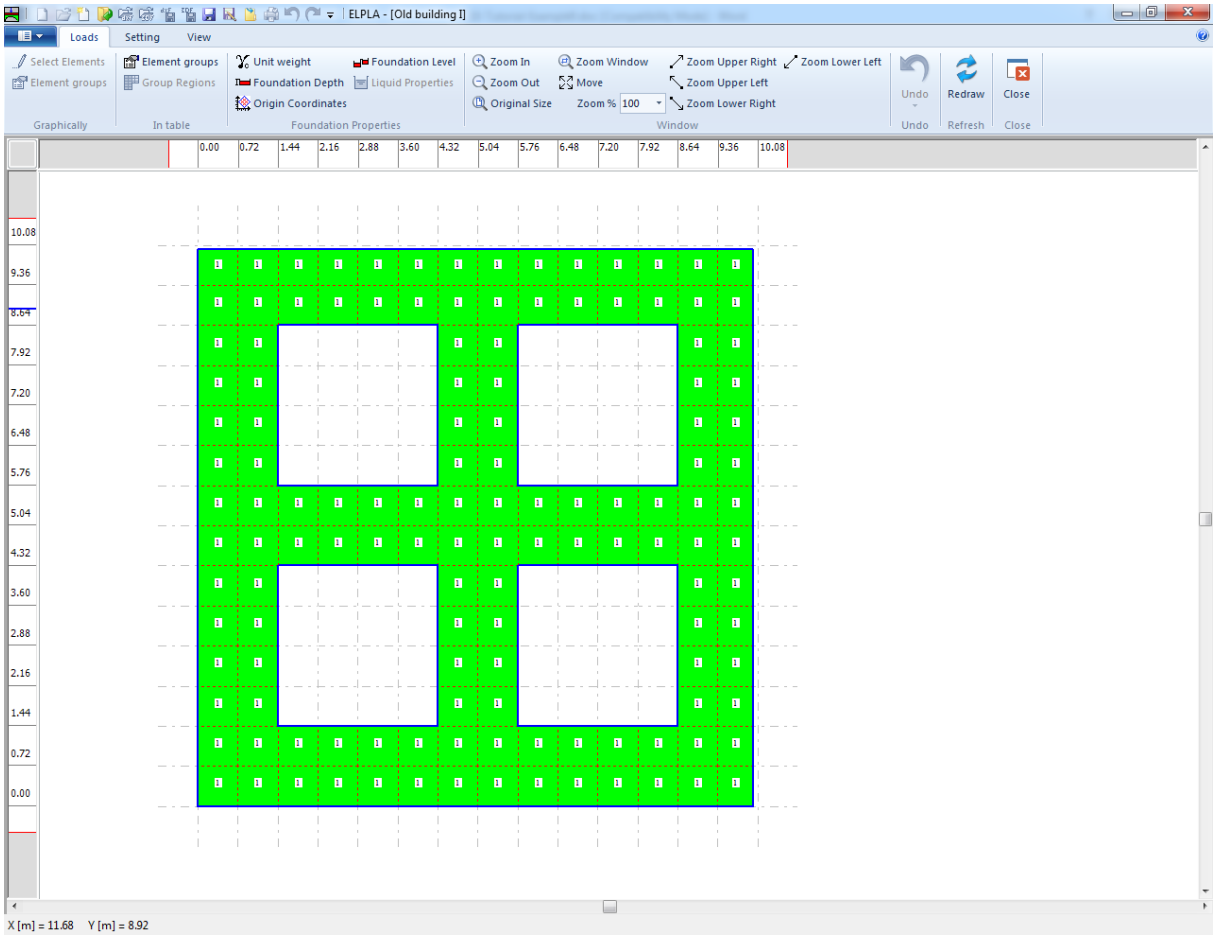


Figure 9.32 "Foundation Properties" Window

To enter the origin coordinates

- Choose "Origin Coordinates" command from "Foundation Properties" menu in the window of Figure 9.32. The following dialog box in Figure 9.33 appears
- In this dialog box type 10.28 in the "*x*-coordinate" edit box
- Click "OK" button

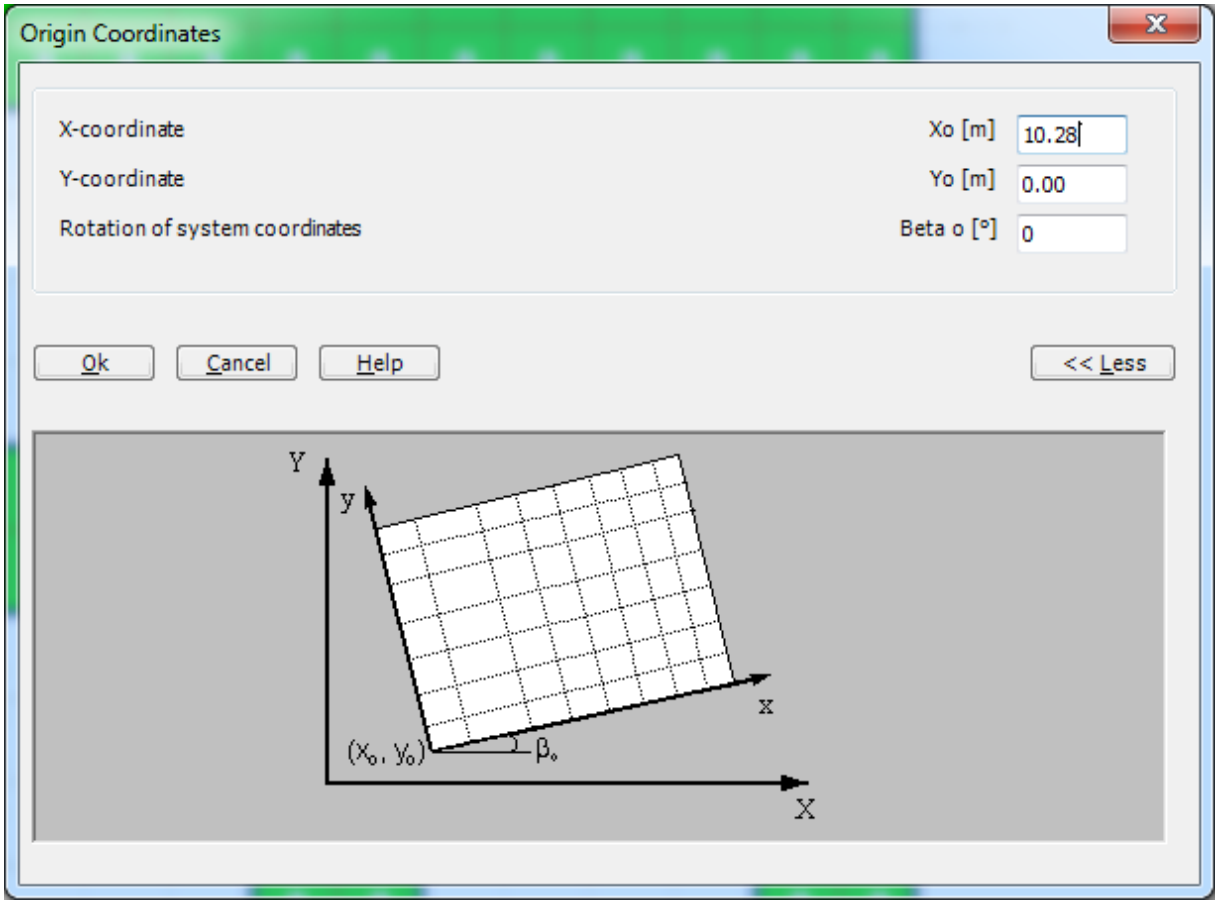


Figure 9.33 "Origin Coordinate" dialog box

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

- Choose "Save" command from "File" menu Figure 9.32 to save the foundation properties
- Choose "Close" command from "File" menu Figure 9.32 to close the "Foundation properties" Window and return to *ELPLA*

3.4 Neighboring foundations

To enter the neighboring foundations, do the following steps:

- Choose "Neighboring Foundations" command from “Data” Tab, “Neighboring foundations” Wizard Figure 9.34 appears
- Click "Add neighboring foundations" button in “Neighboring Foundations” menu in Figure 9.34 and open the project "New Building II"
- Click "Save" button

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

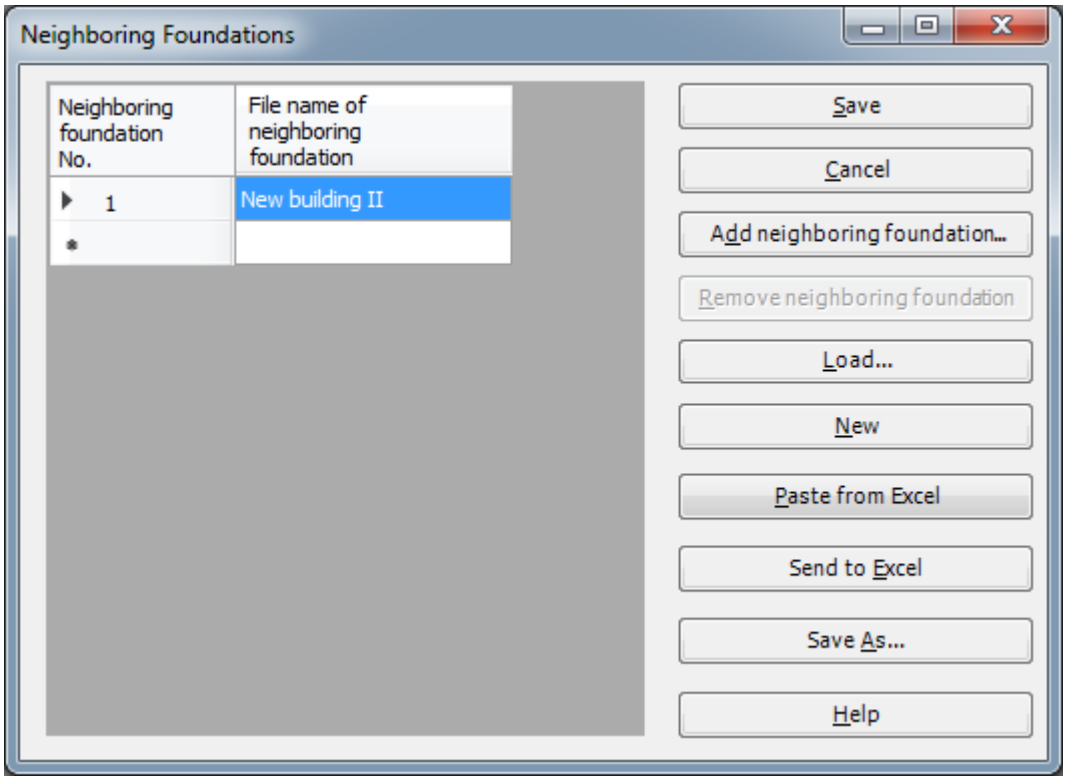


Figure 9.34 "Neighboring Foundations" menu

4 Carrying out the calculations

The calculations of the “New Building II” should be first done to get the contact pressure under its foundation, then the calculation of the “Old Building I” with the influence of building II is carried out.

To analyze the problem open the file "New Building II" from "File" menu from *ELPLA* then switch to "Solver" Tab, Figure 9.35).

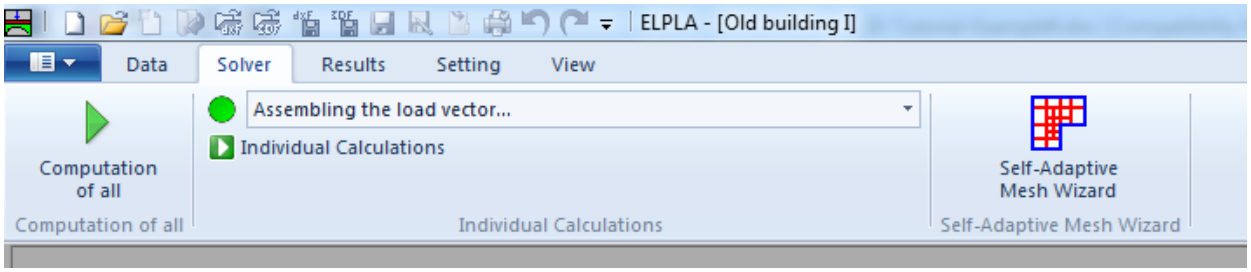


Figure 9.35 "Solver" Tab

The "Individual Calculations" list contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis.

For the project of the New building II, the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Assembling the slab stiffness matrix
- Solving the system of linear equations
- Determining deformation, internal forces, contact pressures

While for the project of the Old building I, the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Influence of neighboring foundations on settlements
- Assembling the slab stiffness matrix
- Solving the system of linear equations
- 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 "Calculation" Tab in "Solver" Tab

Analysis progress

Analysis progress menu in Figure 9.36 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, the status bar down of the "Solver" Tab displays information about the progress of calculation.

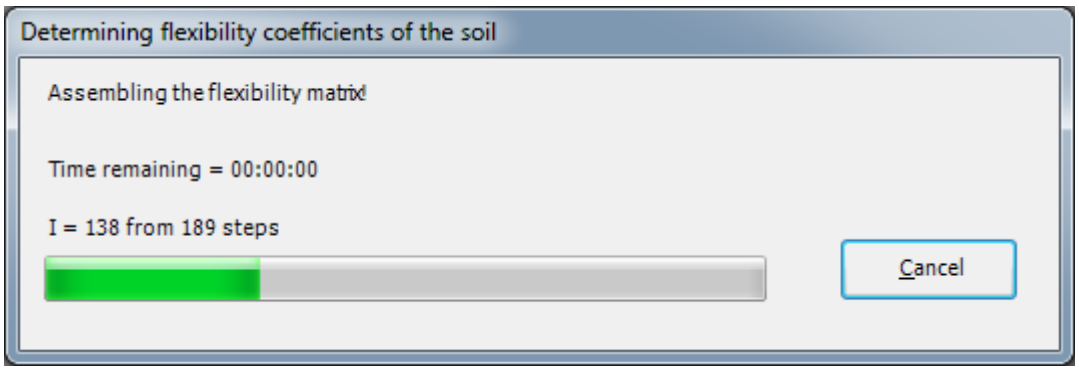


Figure 9.36 Analysis progress menu

Check of the solution

Once the analysis is complete, a check menu of the solution appears, Figure 9.37. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

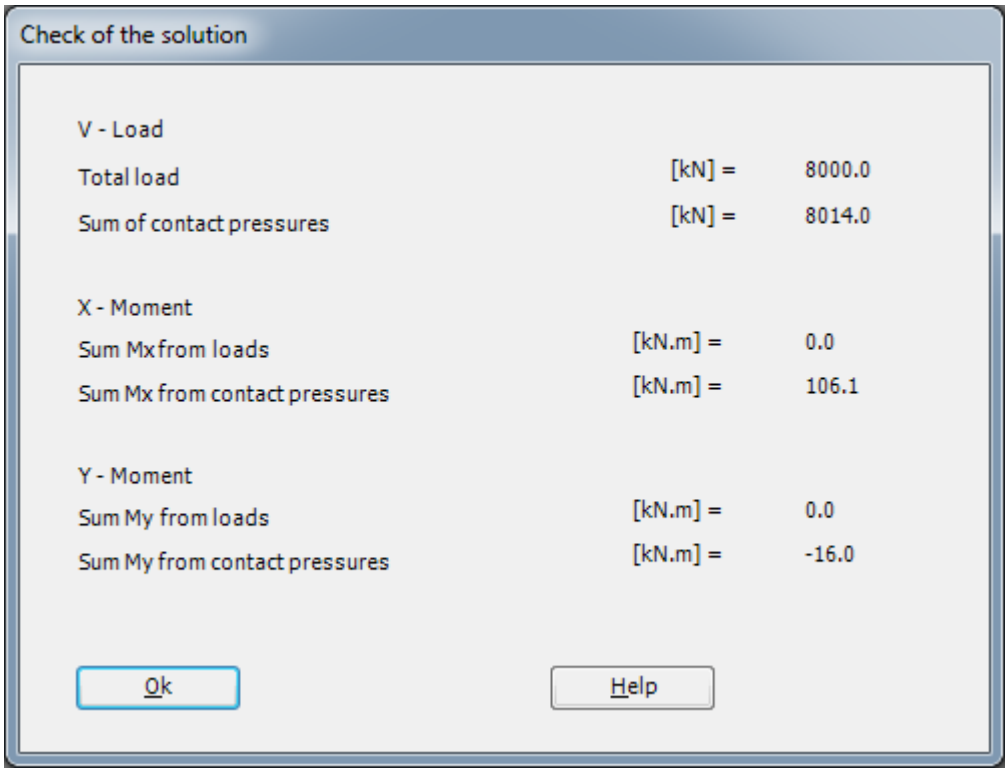


Figure 9.37 Menu "Check of the solution"

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

Then open "Old Building I" from "File" menu from *ELPLA*, to solve the problem of the Old Building I with the influence of building II is carried out.

5 Viewing data and results

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

5.1 Viewing result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 9.38).

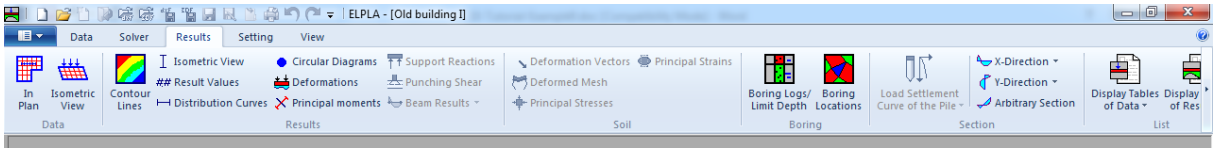


Figure 9.38 "Results" Tab

To view the results for the new foundation only as contour lines

- Choose "Contour lines" command from "Results" menu.
  The following option box in Figure 9.39 appears
- In "Results as Contour Lines" option box, select "Settlements" as a sample for the results
  to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for old foundation only as shown in Figure 9.40.

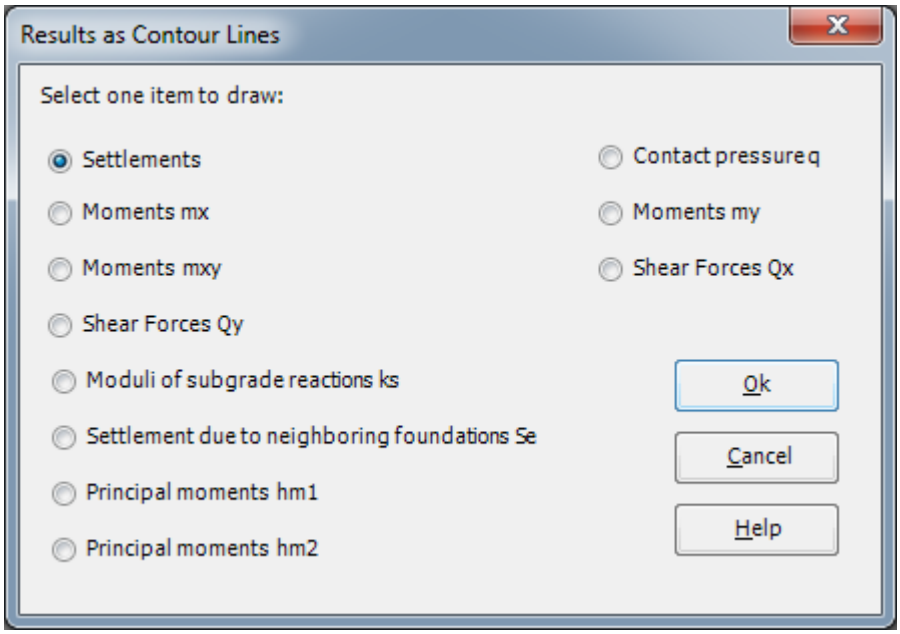


Figure 9.39 "Results as Contour Lines" option box

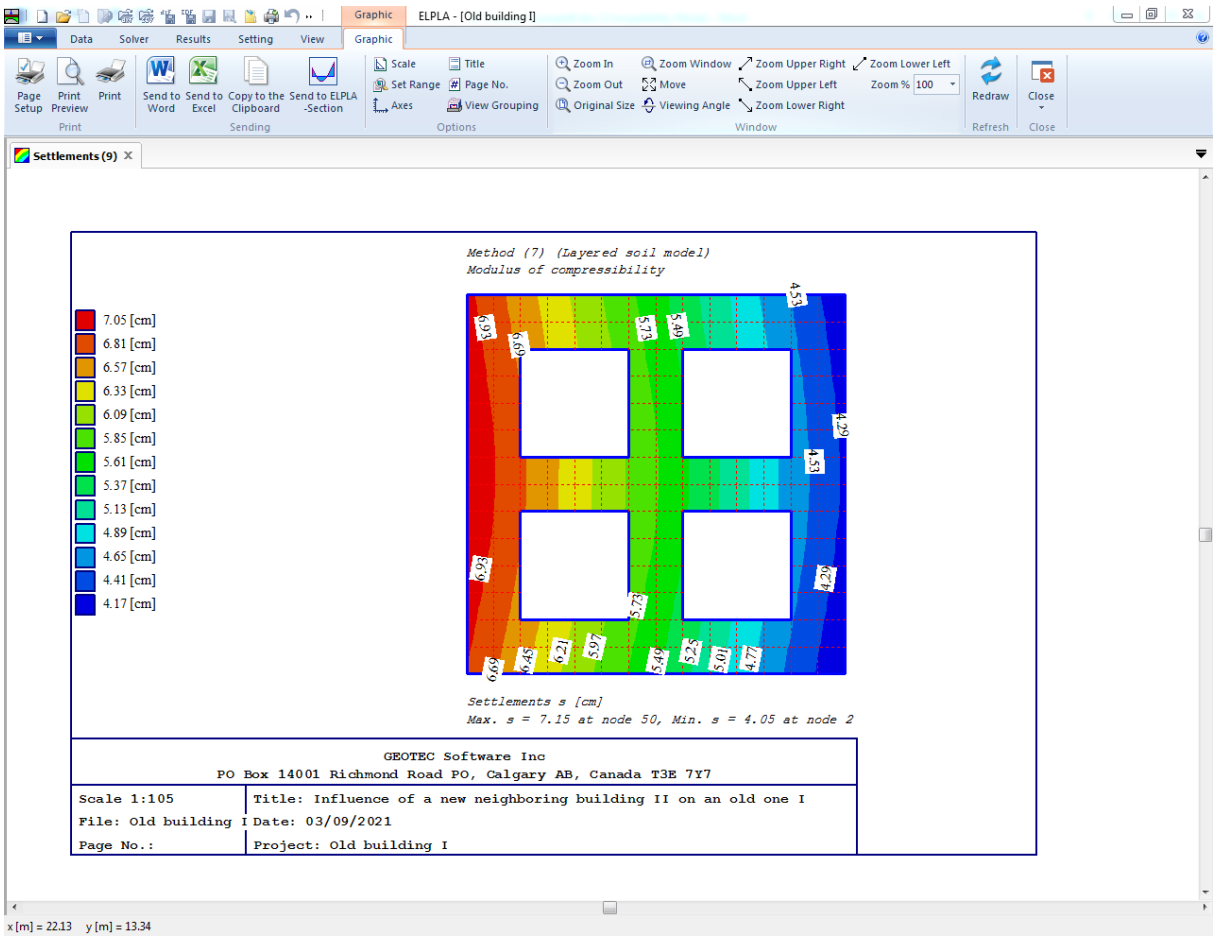


Figure 9.40 Settlements as contour lines for the old foundation only

To view the settlement results for the two foundations together as contour lines

- Choose "Plot Parameters" command from "Setting" Tab. "Plot Parameters" dialog box in Figure 9.41 appears
- check the option "Draw neighboring foundations" from "General plot Parameters" menu
  as shown in Figure 9.41
- Click "OK" button

The settlements are now displayed as contour lines for the two foundations together as shown
in Figure 9.42.

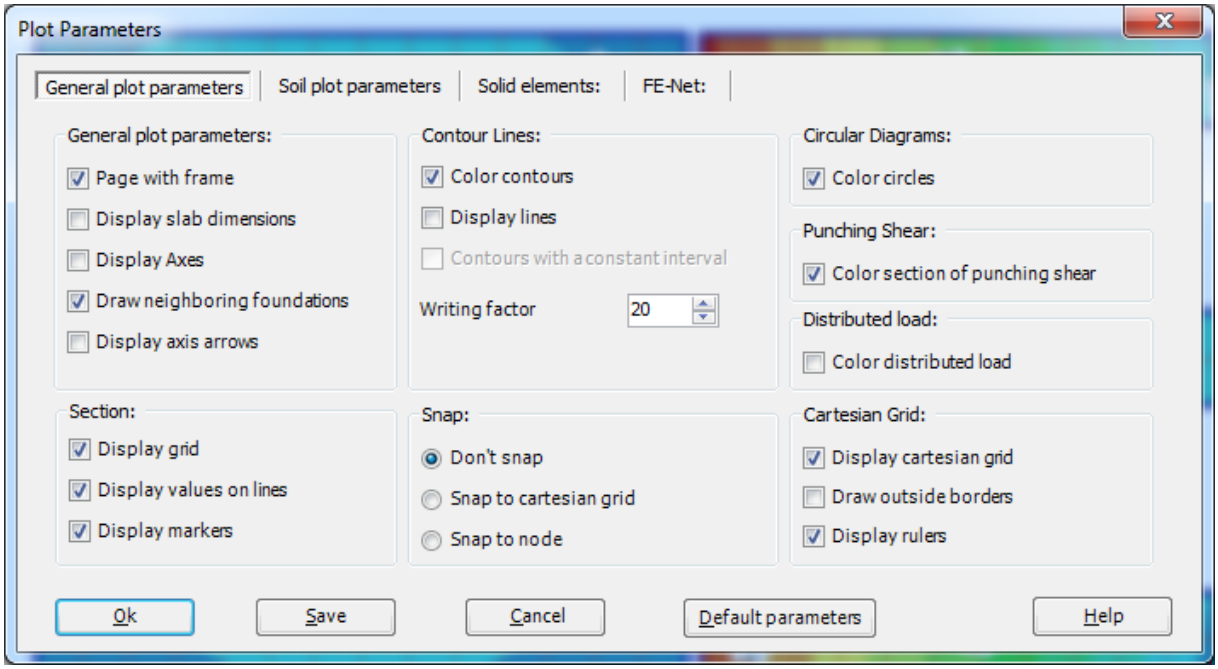


Figure 9.41 "Plot Parameters" dialog box

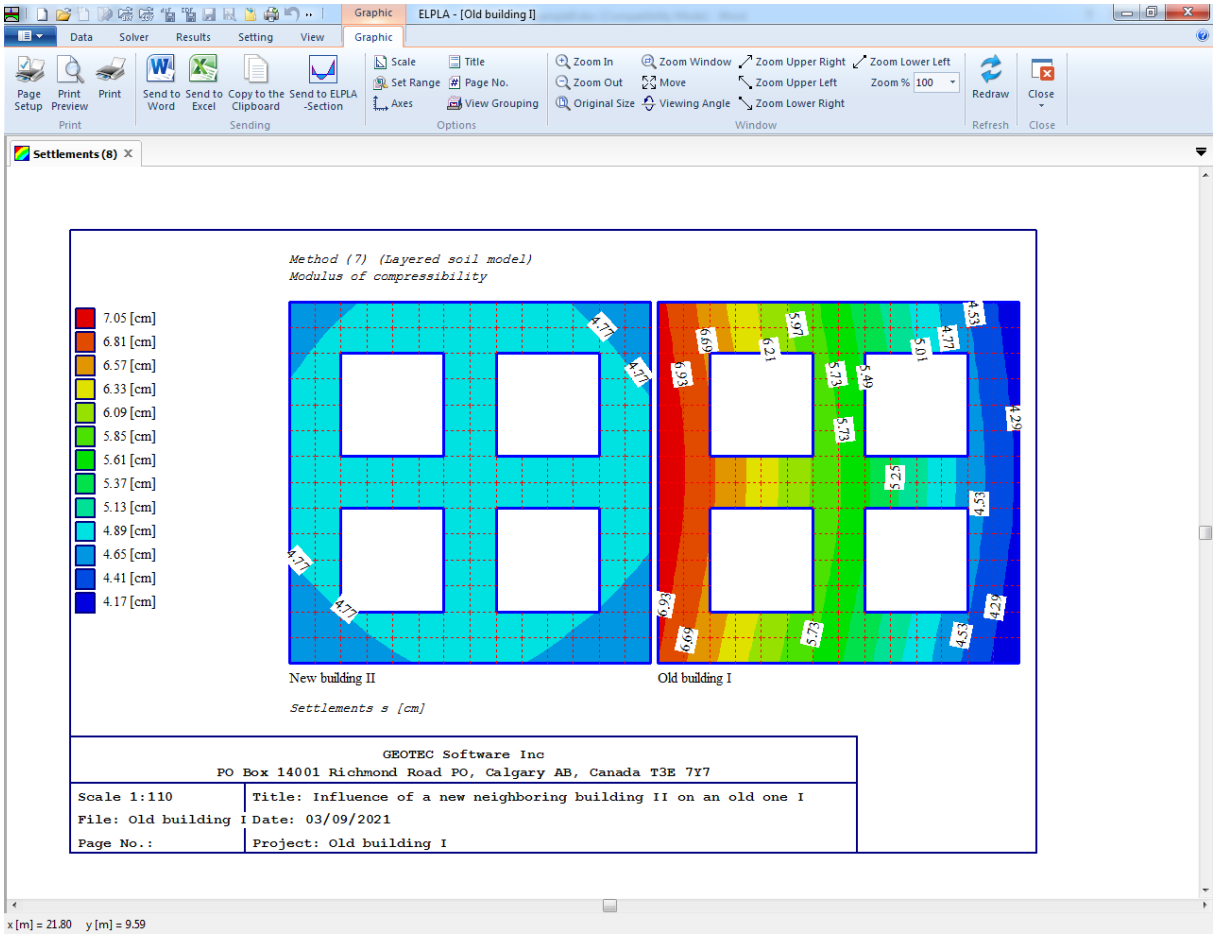


Figure 9.42 Settlements as contour lines for the two foundations together

5.2 Drawing a graph of results

ELPLA can plot diagram of the results of both foundations together, this is done by switching to "View" Tab Figure 9.43

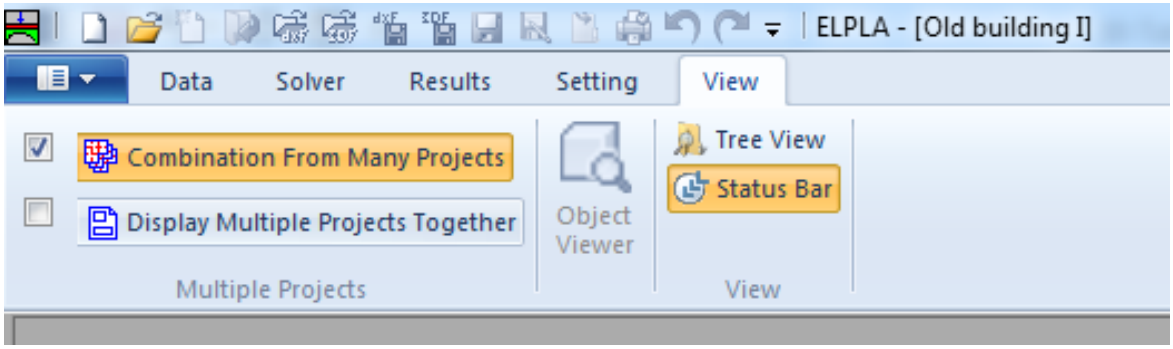


Figure 9.43 "View"

- Select and check “Combination From Many Projects” Figure 9.44 appears
- In this List box select "Add Project", then choose "New building II"

- Click "Save" button

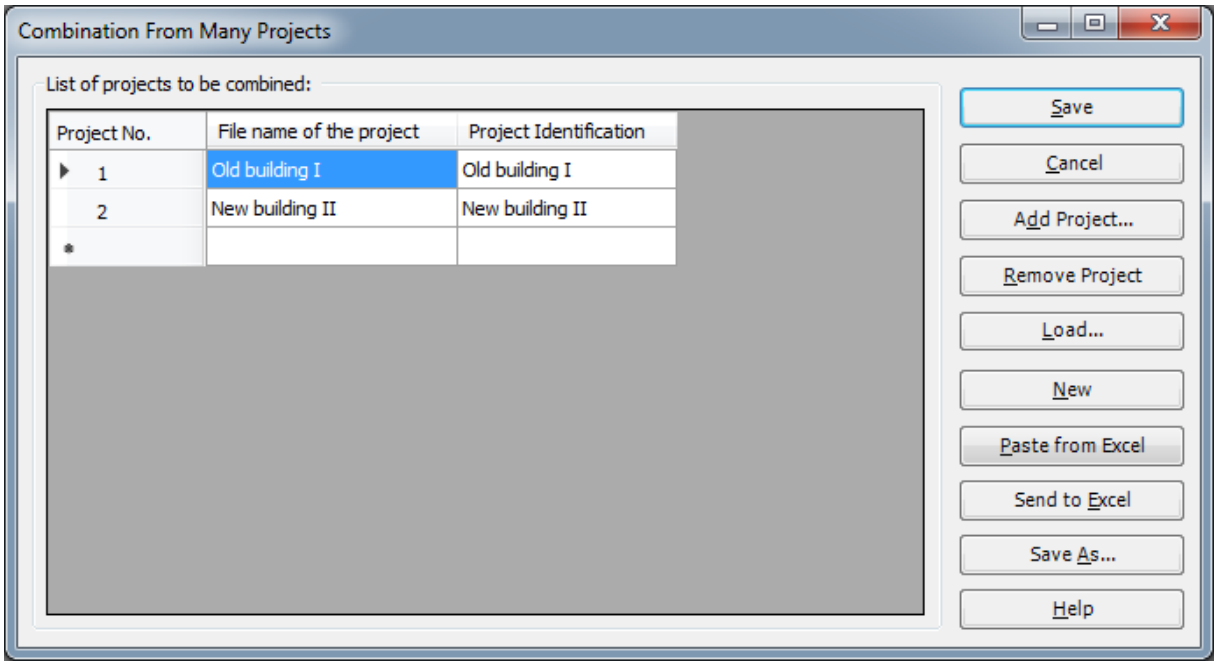


Figure 9.44 "Combination From Many Projects" list box

To plot a diagram of the results. 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 "Section" Menu in the "Result" Tab. The following option box in Figure 9.45 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

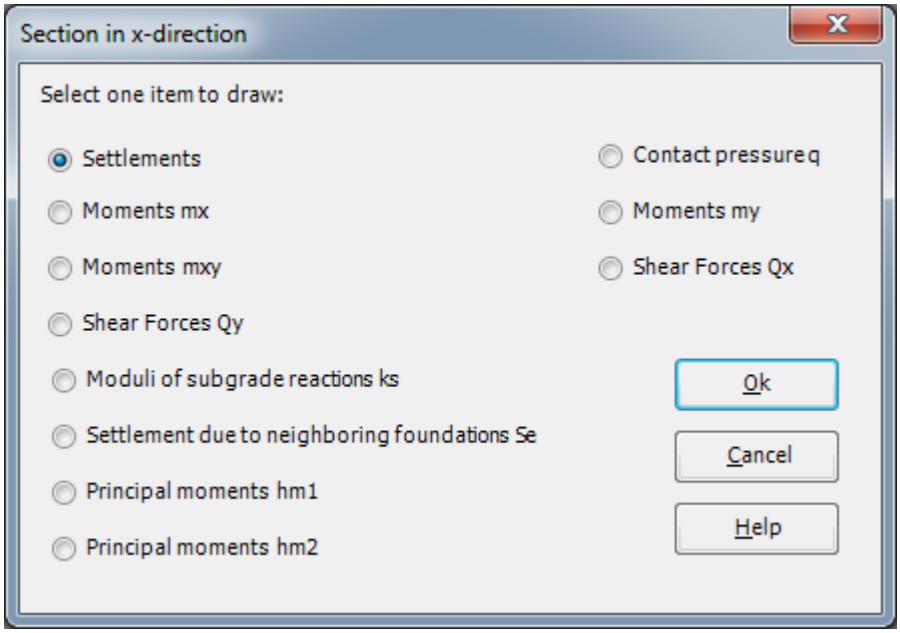


Figure 9.45 "Section in x-direction" option box

The following option box in Figure 9.46 appears to specify the section in *x*-direction that is required to be plotted.

In this dialog box

- Type 5.04 in the "Section at *y*-coordinate" edit box to plot a diagram at the middle of the foundation
- Type 10.08 in the "Section at *x*-coordinate" edit box
- Click "OK" button

The settlements of both foundations are now plotted in a diagram as shown in Figure 9.47

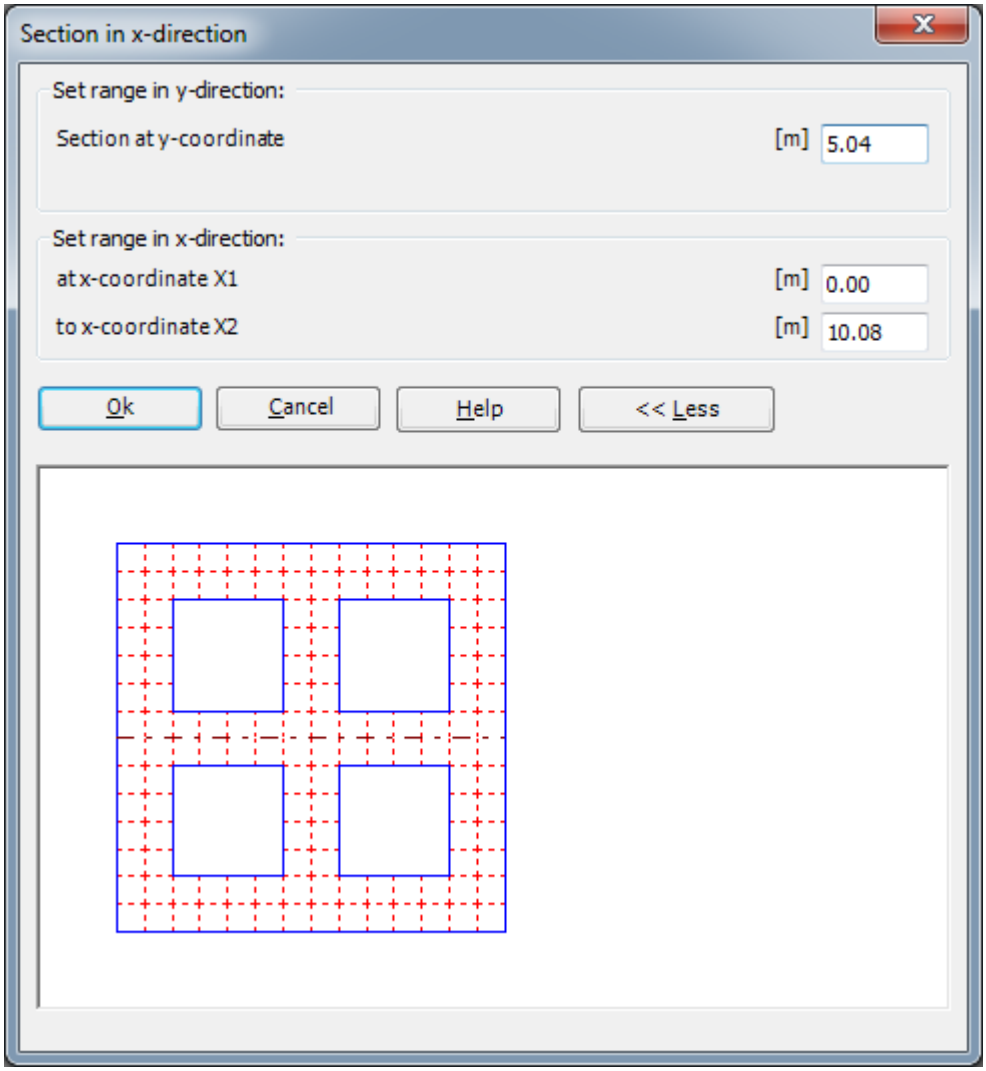


Figure 9.46 "Section in x-direction" dialog box

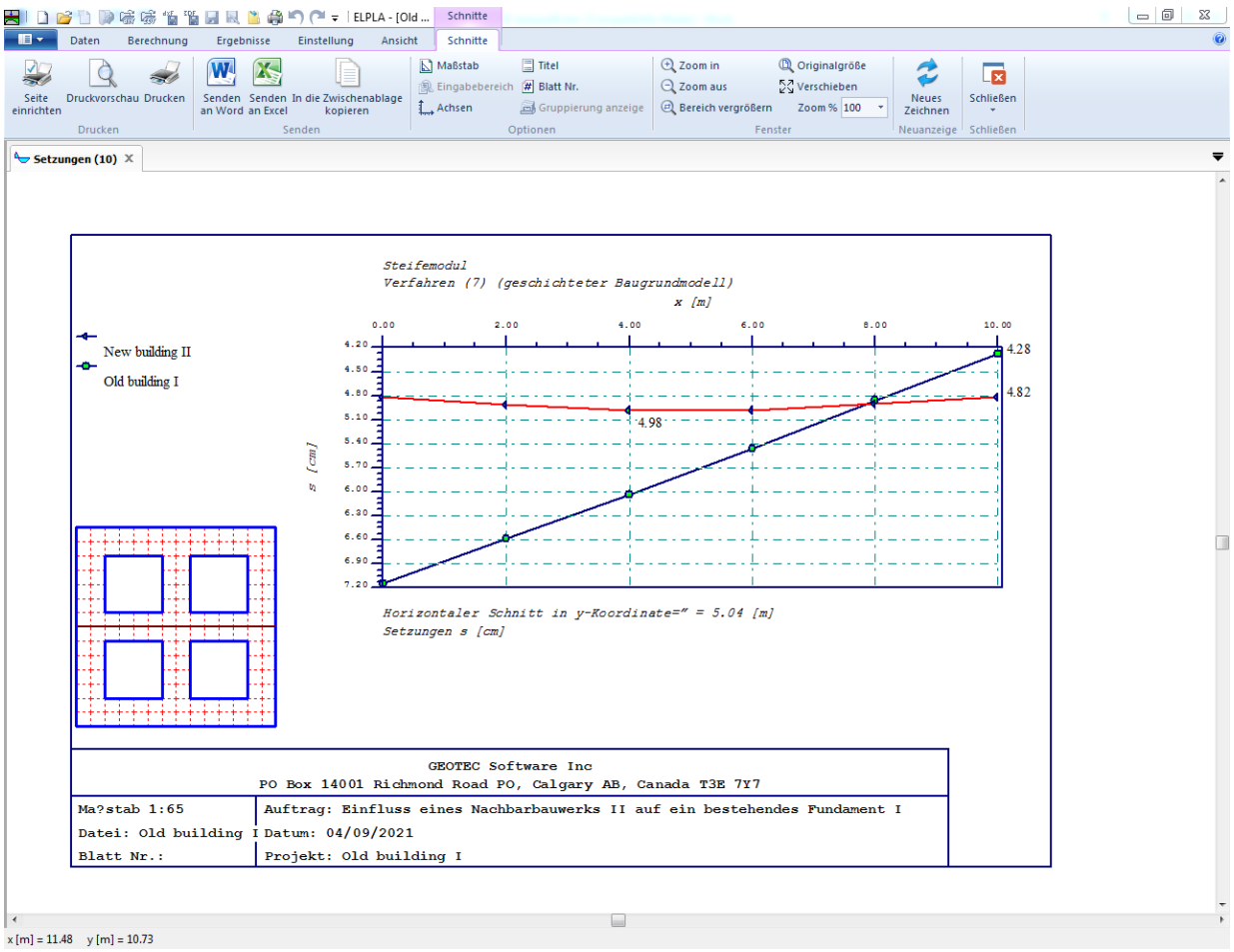


Figure 9.47 Diagram of settlements in x-direction at the middle of both foundations

6 Index

A

Analysis · 5, 6, 31, 37

B

boring log · 18, 21, 23

C

Calculation · 36
Calculation methods · 6, 31
check of the solution · 38
Computation of all · 37
contact pressures · 36
contour lines · 39, 40
coordinates · 3, 5, 27, 30, 33

D

deformation · 36
diagram · 42, 43
diagrams · 39

E

E-Modulus · 25
Ergebnissen · 42

F

FE-Net · 11
File · 6, 9, 17, 23, 26, 29, 30, 34, 36, 38
flexibility coefficients · 36
foundation · 3, 5, 6, 11, 14, 20, 24, 25, 26, 27, 34, 36, 39, 40
foundation properties · 24

G

Generation parameters · 13
German specification code · 20
Graphic · 39, 40
graphics · 39
groundwater · 20, 22

I

internal forces · 36

K

Key · 16

L

load vector · 36
Loads · 27

M

Main soil data · 22
Modulus of compressibility for loading · 5
Modulus of compressibility for reloading · 5

N

nodes · 3, 5, 15, 16

O

Options · 43
origin coordinate · 33

P

Point loads · 3, 27, 28
Poisson's ratio · 25
Project identification · 11, 32

R

raft · 44
Remove nodes · 15
Results · 42

S

Save · 8
Save project as · 9, 30
Select nodes · 15
Slab type · 12

soil properties · 18
soil stiffness matrix · 36
System symmetry · 7, 31

U

Unit weight · 25
Unsymmetrical system · 7
User's Guide · 43