Example 2

Analysis of a slab floor

Content Page

1 Description of the problem

An example of a slab floor with girders is selected to illustrate some features of *ELPLA* for analyzing slab floors.

1.1 Loads and dimensions

The slab floor has a thickness of 10 [cm] and carries uniform loads with different intensities as shown in Figure 2.1. All girders have the same dimensions of 15 [cm] × 60 [cm]. Own weight of the girder is 1.875 [kN/m].

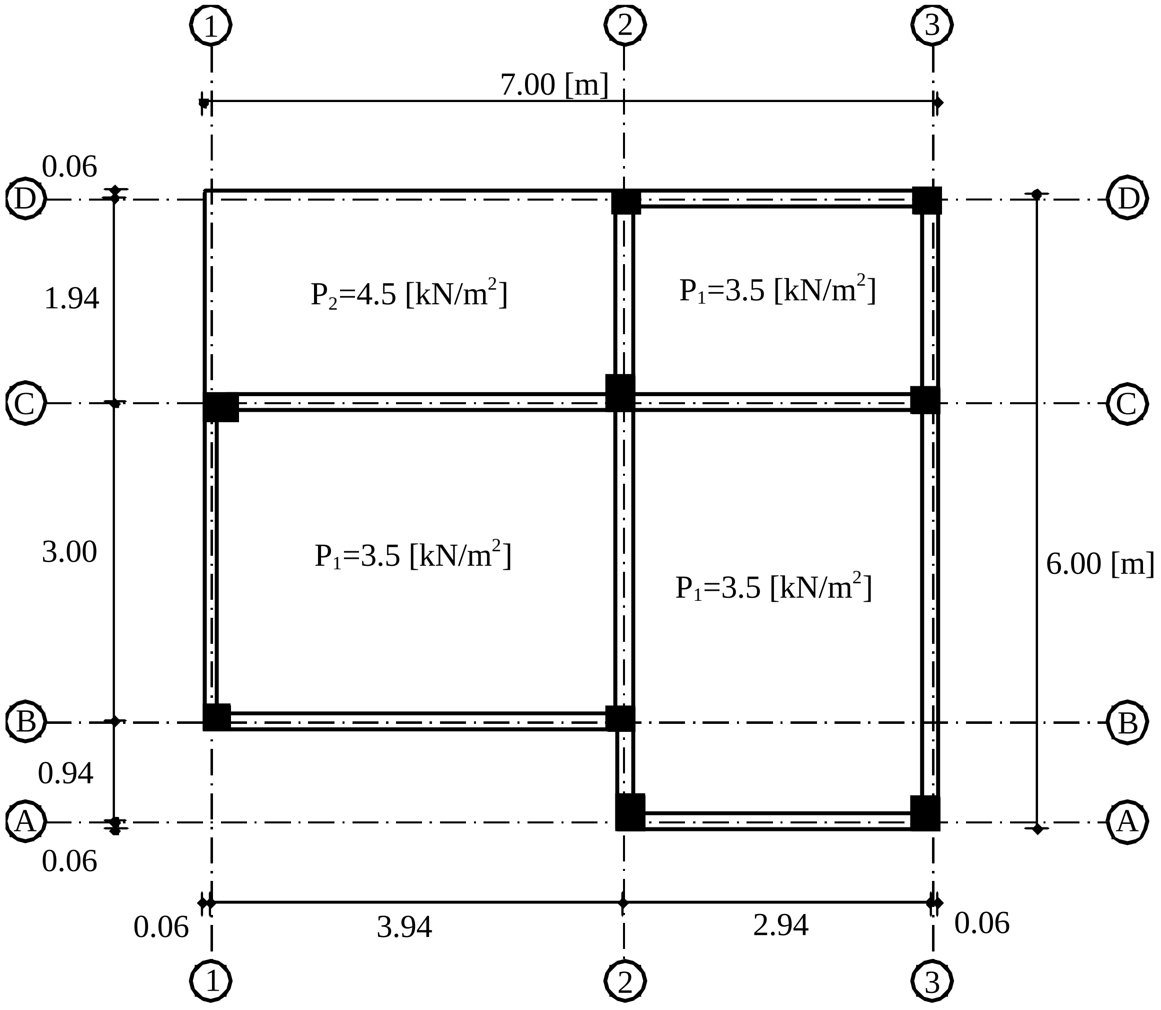

Figure 2.1 Dimensions of the slab with loads

1.2 Slab material

Material of the slab is concrete (C 30/37) that has the following parameters:

Young's modulus of the concrete	E_b	$= 3.2 \times 10^7$	[kN/m²]
Poisson's ratio of the concrete	ν_b	= 0.20	[-]
Unit weight of the concrete	γ_b	= 25	[kN/m³]
Shear modulus of the concrete	$G_b = 0.5\, E_b\, (1+ \nu_b)$	$= 1.3 \times 10^7$	[kN/m²]

1.3 Analysis and concrete design

The concrete sections are designed according to EC2 code for the following parameters:

Concrete grade C 30/37		
Steel grade BSt 500		
Characteristic compressive cylinder strength of concrete f_{ck}	= 30	[MN/m²]
Characteristic tensile yield strength of reinforcement f_{yk}	= 500	[MN/m²]
Partial safety factor for concrete strength γ_c	= 1.5	[-]
Design concrete compressive strength $f_{cd} = f_{ck}/\ \gamma_c$	= 30/1.5 = 20	[MN/m²]
Partial safety factor for steel strength γ_s	= 1.15	[-]
Design tensile yield strength of reinforcing steel $f_{yd} = f_{yk}/\ \gamma_s$	= 500/1.15 = 435	[MN/m²]

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 numerical calculation methods is 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 slab floor. The example 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

To create the project, start the *ELPLA* and choose "New Project" command from "File" menu. The "Calculation Method" wizard appears, Figure 2.2. As shown in this Figure, the first form of the wizard is the "Analysis Type" form.

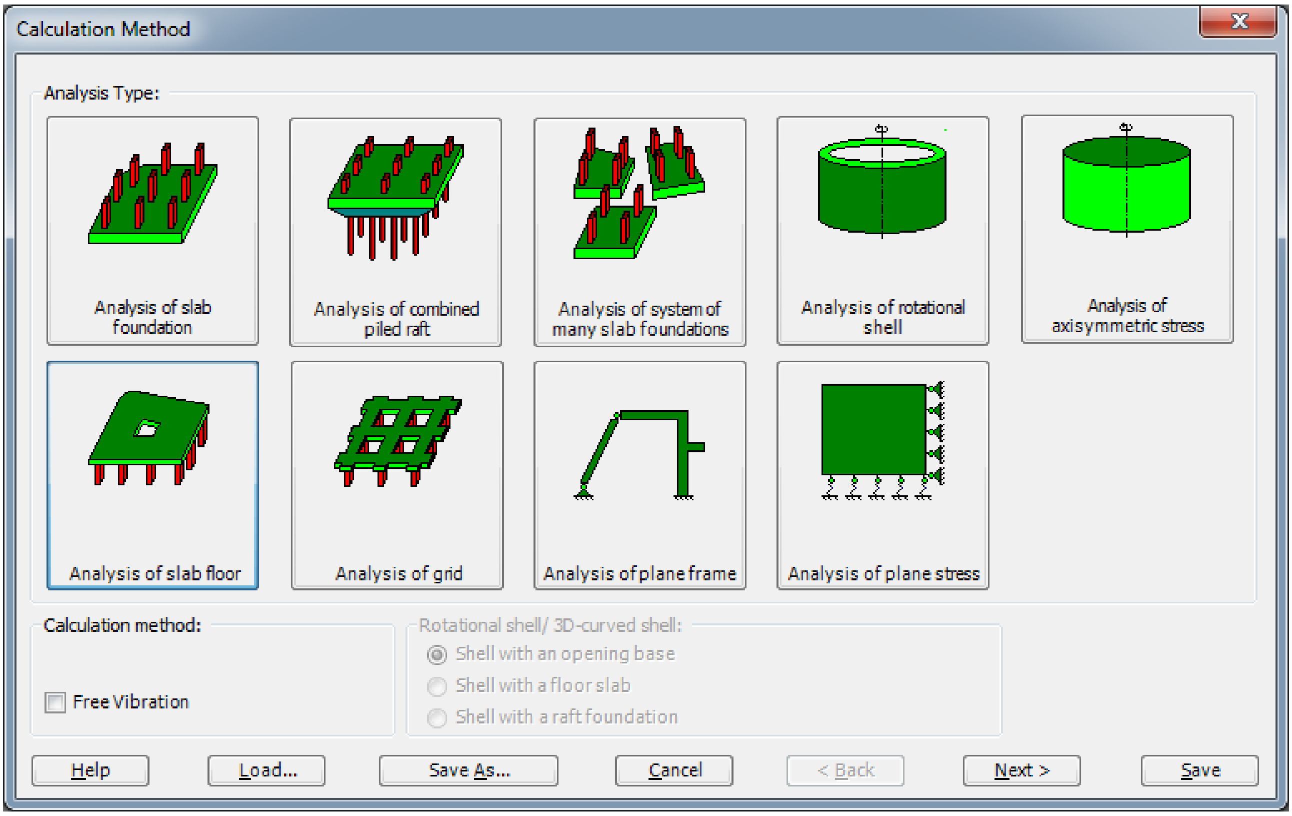

Figure 2.2 "Calculation Method" wizard with "Analysis type" form

In the "Analysis Type" form in Figure 2.2, define the analysis type of the problem. As the analysis type is a slab floor problem, select "Analysis of slab floor". Then, click "Next" button to go to the next form. The next form is the "System Symmetry", Figure 2.3.

In this form

- Choose "Unsymmetrical System"
- Click "Next" button

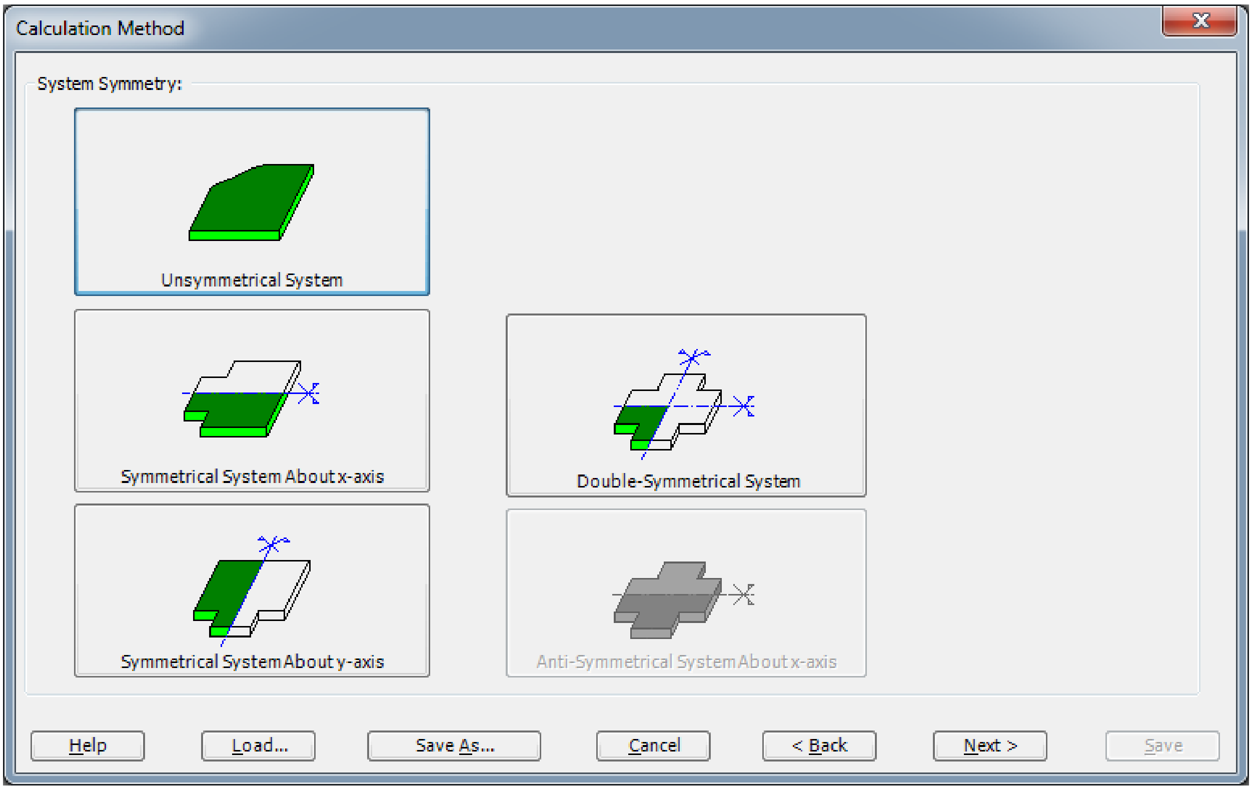

Figure 2.3 "System Symmetry" form

The last form of the wizard assistant contains the "Options" list, Figure 2.4. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other.

In this list

- Check "Supports/ Boundary Conditions" check box
- Check "Slab With Girders" check box
- Check "Concrete Design" check box
- Click "Save" button

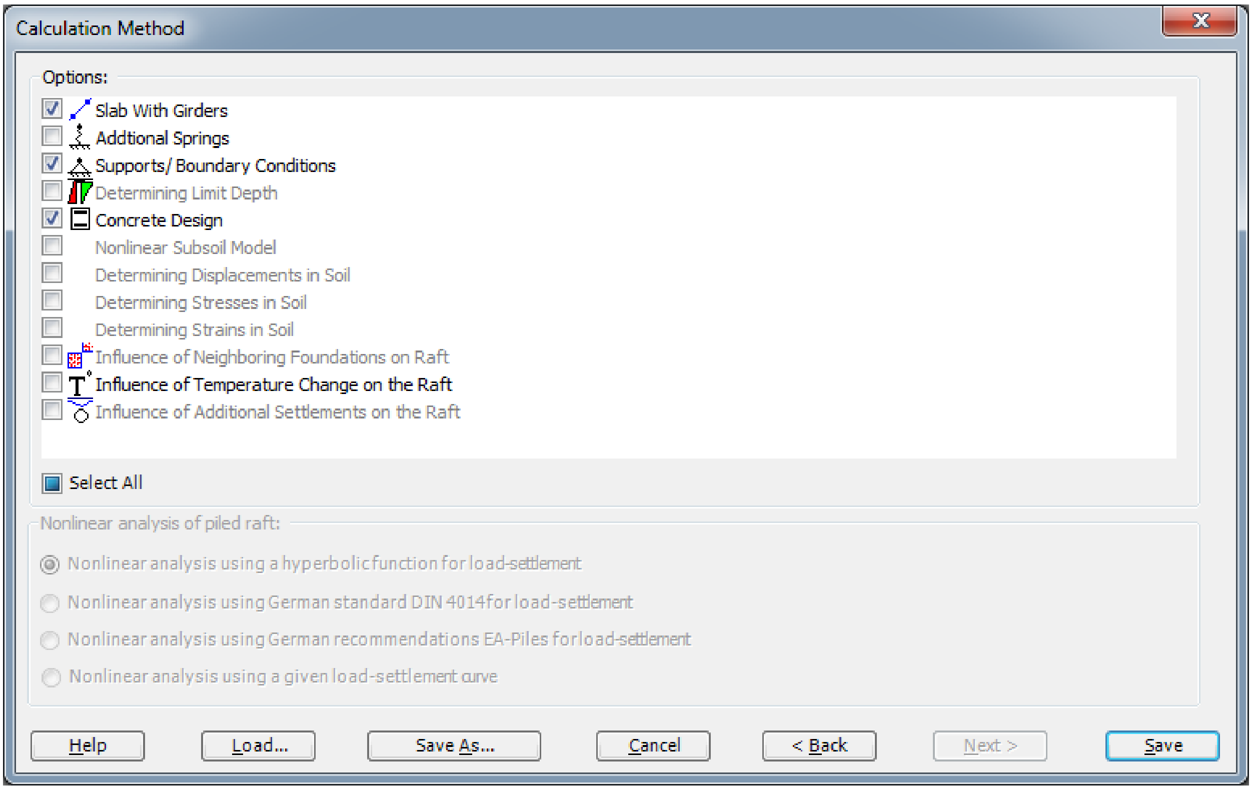

Figure 2.4 "Options" list

After clicking "Save" button, the "Save as" dialog box in Figure 2.5 appears. In this dialog box

- Type a file name for the current project in the file name edit box. For example, type "Floor". *ELPLA* will use automatically this file name in all reading and writing processes
- Click "Save" button to complete the definition of the calculation method and the file name of the project

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

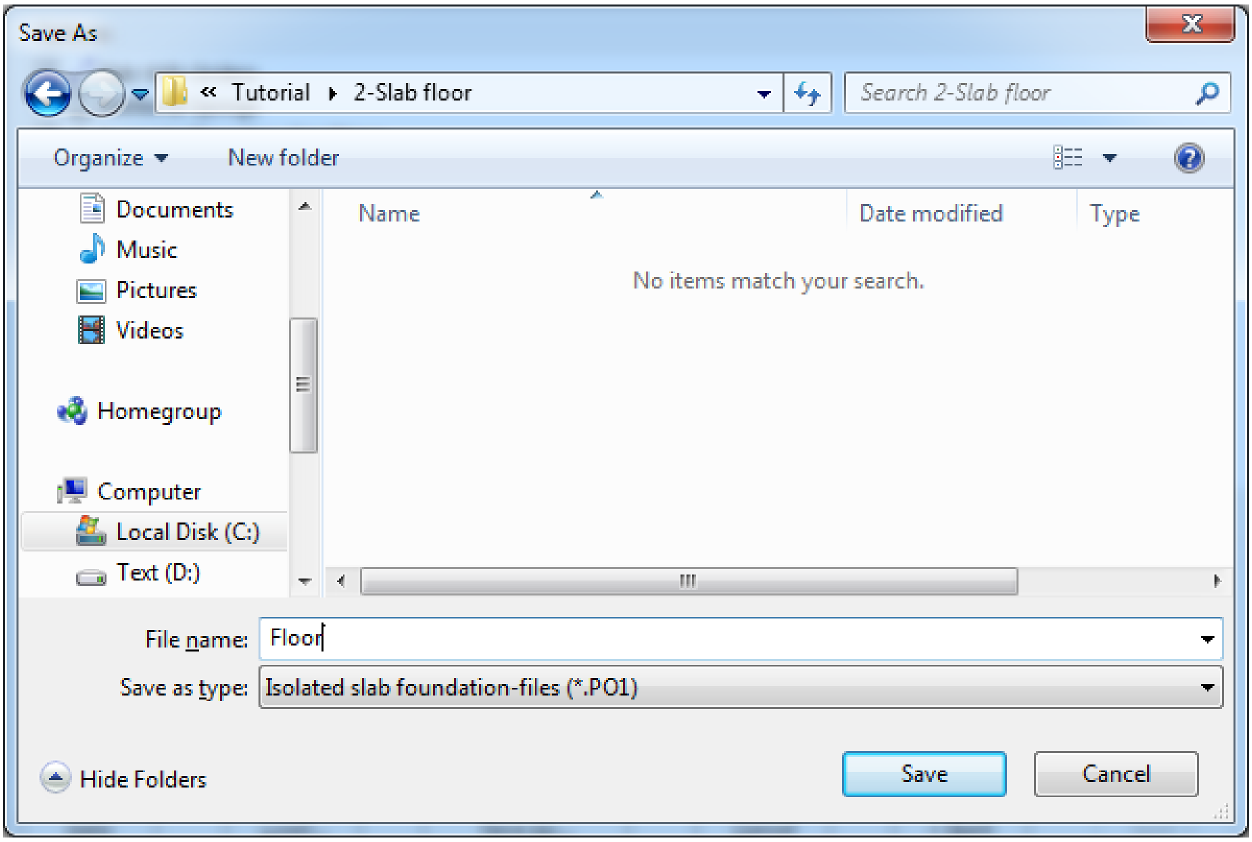

Figure 2.5 "Save as" dialog box

2.2 Project Identification

To identify the project choose "Project Identification" command from “Data” Tab. The dialog box in Figure 2.6 appears.

In this dialog box

- To describe the problem, type the following line in the "Title" edit box: "Analysis of a slab floor"
- Type the date of the project in the "Date" edit box
- Type "slab floor" in the "Project" edit box
- Click "Save" button

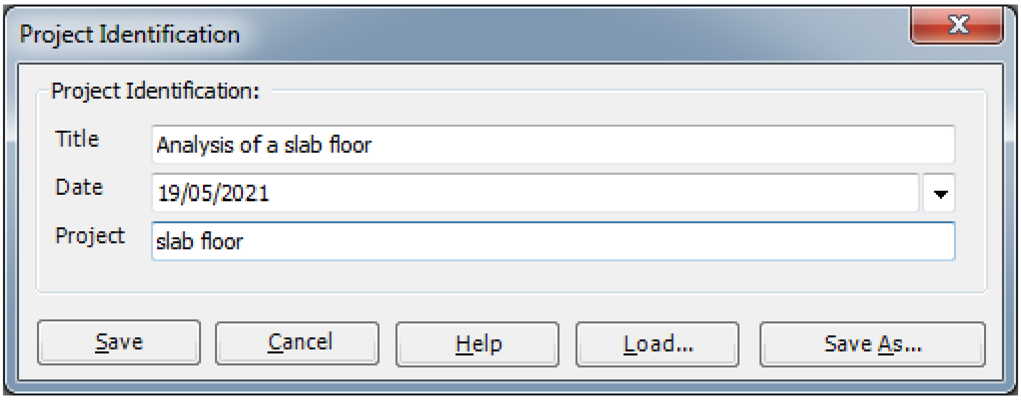

Figure 2.6 "Project Identification" dialog box

2.3 FE-Net Data

For the given problem, the slab has irregular shape and is divided into 7 × 6 elements. Element size in both *x*- and *y*-directions is 1.0 [m] as shown in Figure 2.1. *ELPLA* has different procedures for defining the same problem. The easy procedure to define the FE-Net of this slab is generating a mesh for the entire area first and then removing the unnecessary nodes to get the slab shape.

To define the FE-Net for this slab, choose "FE-Net Data" command from “Data” Tab. The "FE-Net Generation" wizard appears as shown in Figure 2.7. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 2.7, 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 constant size in both *x*- and *y*-directions.

To generate the FE-Net

- Choose the rectangular slab option in the "Slab Type" options
- Type 7 in the "Length of Rectangular Slab" edit box
- Type 6 in the "Width of Rectangular Slab" edit box
- Click "Next" button to go to the next form

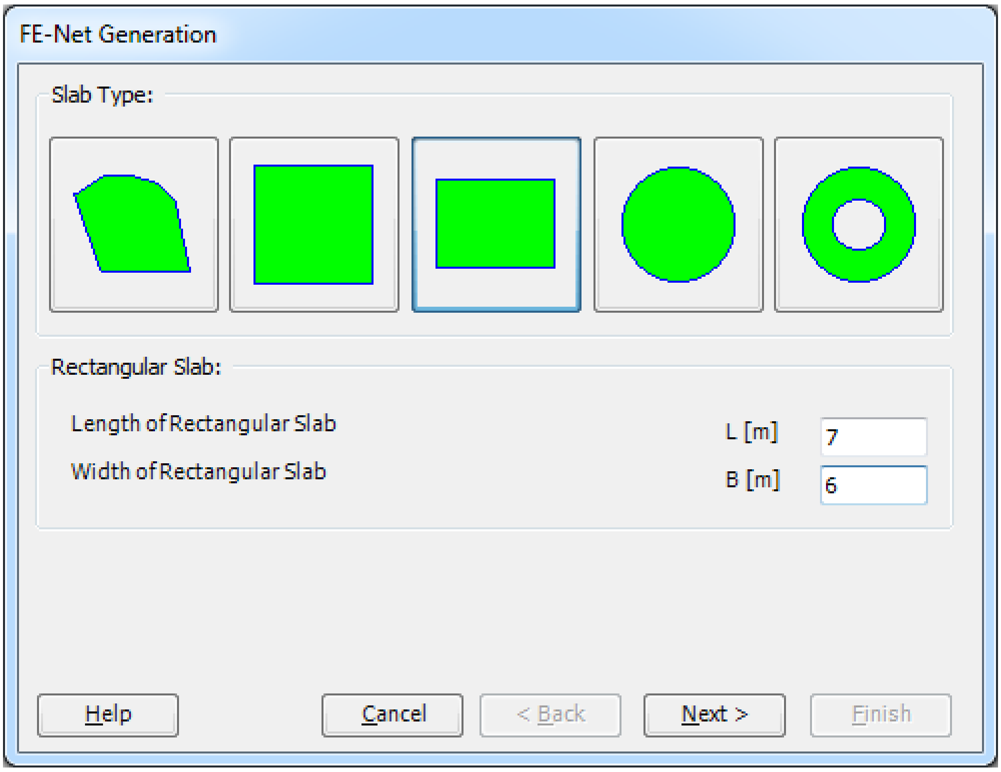

Figure 2.7 "FE-Net Generation" wizard with "Slab Type" form

After clicking "Next" in the "FE-Net Generation" wizard, the following "Generation Type" form appears, Figure 2.8. *ELPLA* can deal with various types of generation with triangle and / or rectangular elements.

In the "Generation Type" form

- Choose rectangular elements
- Click "Next"

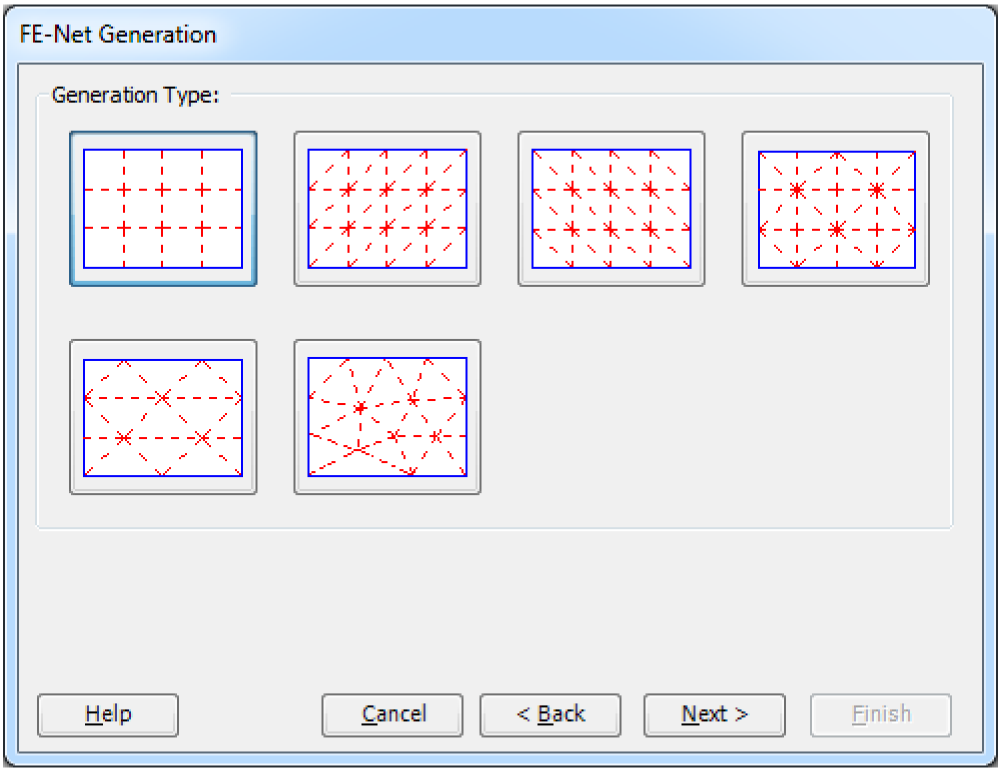

Figure 2.8 "Generation Type" form

After clicking "Next" button in "Generation Type" form, the following "Grid Definition" dialog box in Figure 2.9 appears with default values of constant element size.

In this dialog box

- In "Grid in *x*-direction" frame, type 7 in the "No. of grid intervals" edit box
- In "Grid in *y*-direction" frame, type 6 in the "No. of grid intervals" edit box
- Click "Finish" button

ELPLA will generate a FE-Net for a rectangular slab of 7 [m] length and 6 [m] width with square elements of 1.0 [m] each side. The following window Figure 2.10 appears with the generated net.

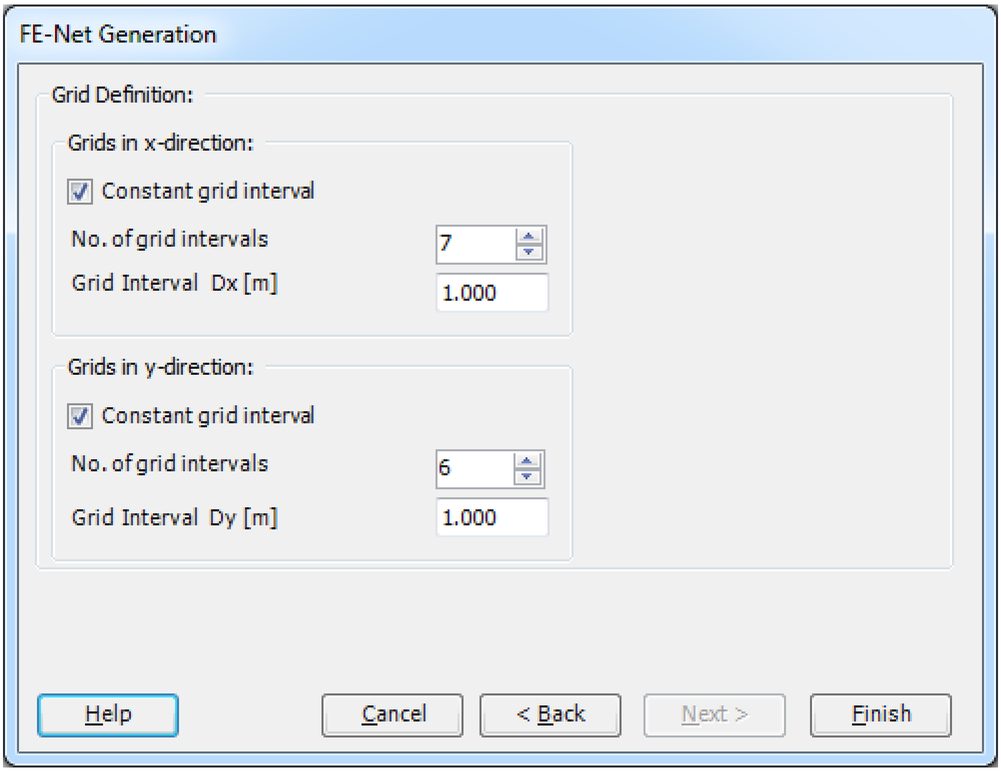

Figure 2.9 "FE-Net Generation" dialog box

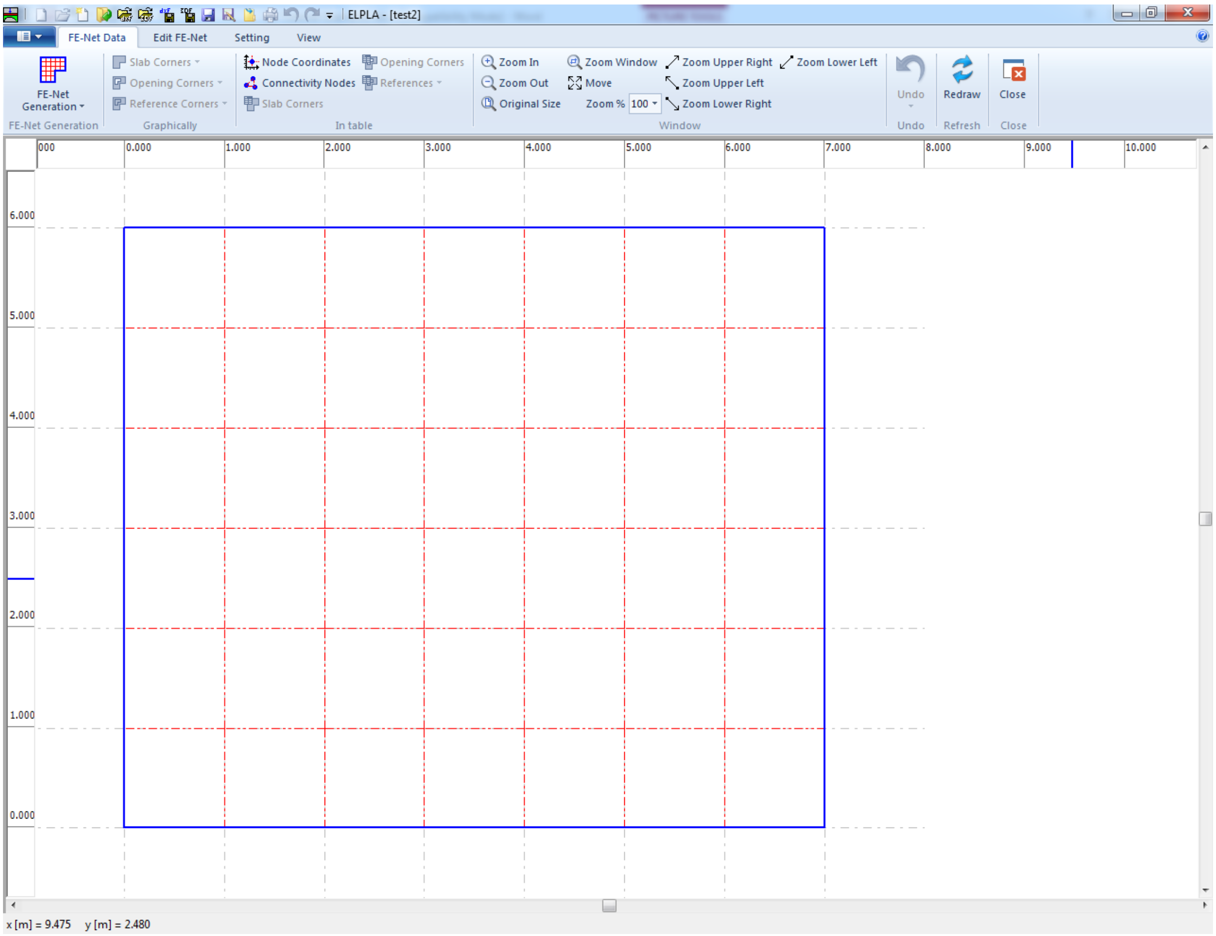

Figure 2.10 Imaginary net of a rectangular area

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 the "Edit FE-Net" menu (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 the "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 2.11. A group of nodes can be selected by holding the left mouse button down at the corner of the region. Then, drag 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.

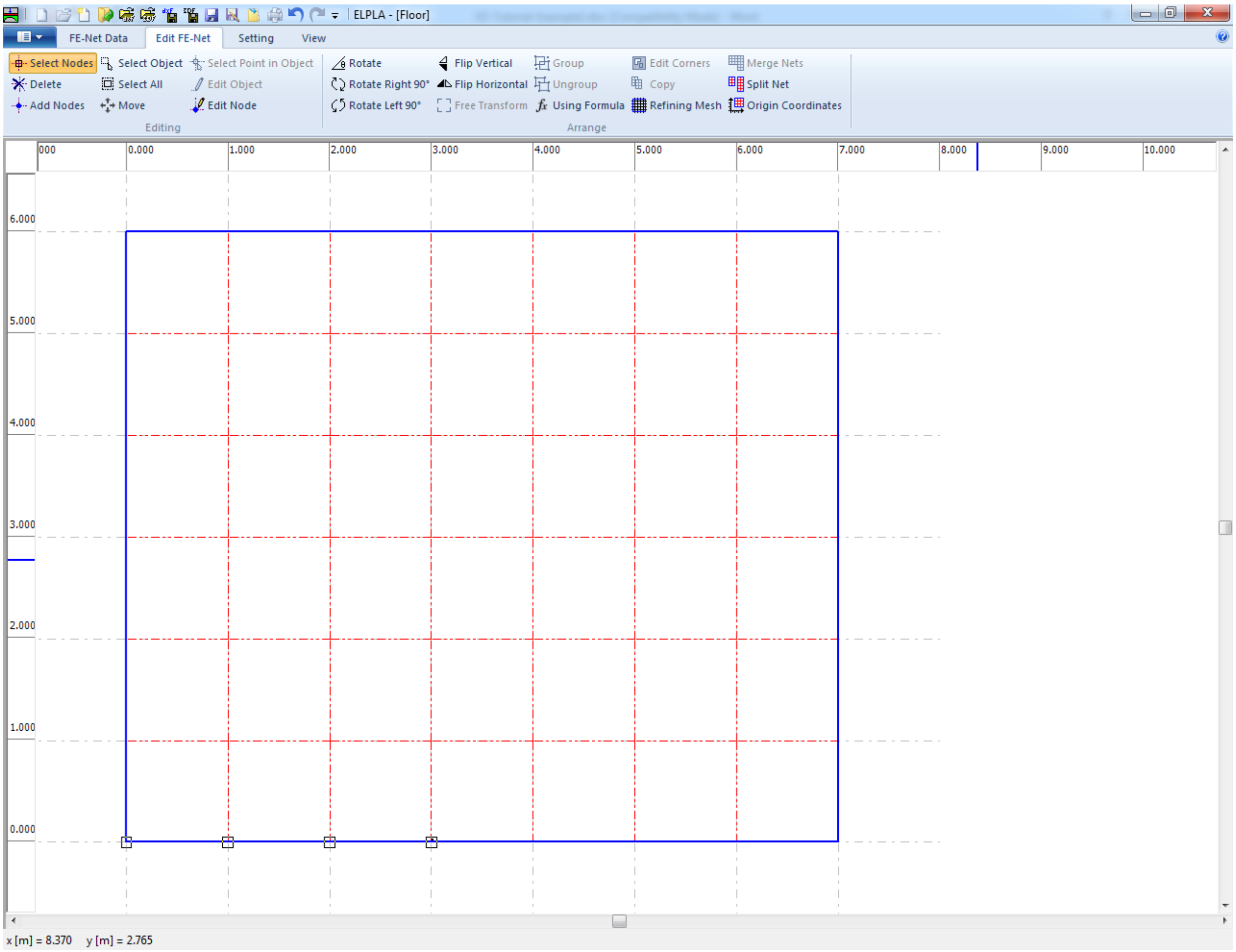

Figure 2.11 Generated FE-Net after selecting the unnecessary nodes

To remove the selected nodes, choose "Delete" command from "Edit FE-Net" menu. The action of this command is indicated in Figure 2.12. To leave the graphic mode, press "Esc" key.

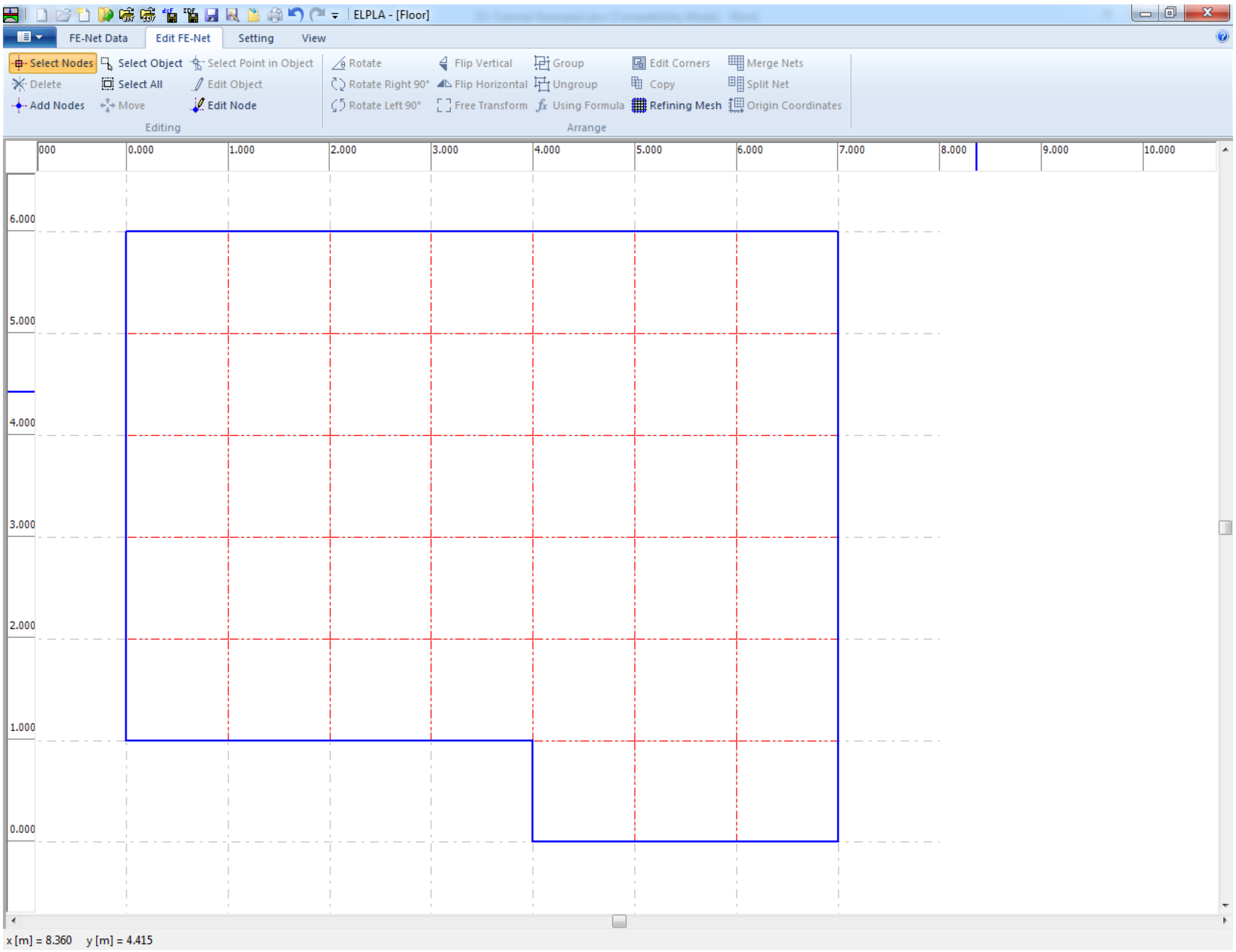

Figure 2.12 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save" command from "File" menu in Figure 2.12 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 2.12 to close "FE-Net" Window and return to ELPLA main window.

2.4 Girders

To define the girders choose "Girders" command from “Data” Tab. The following Window in Figure 2.13 appears.

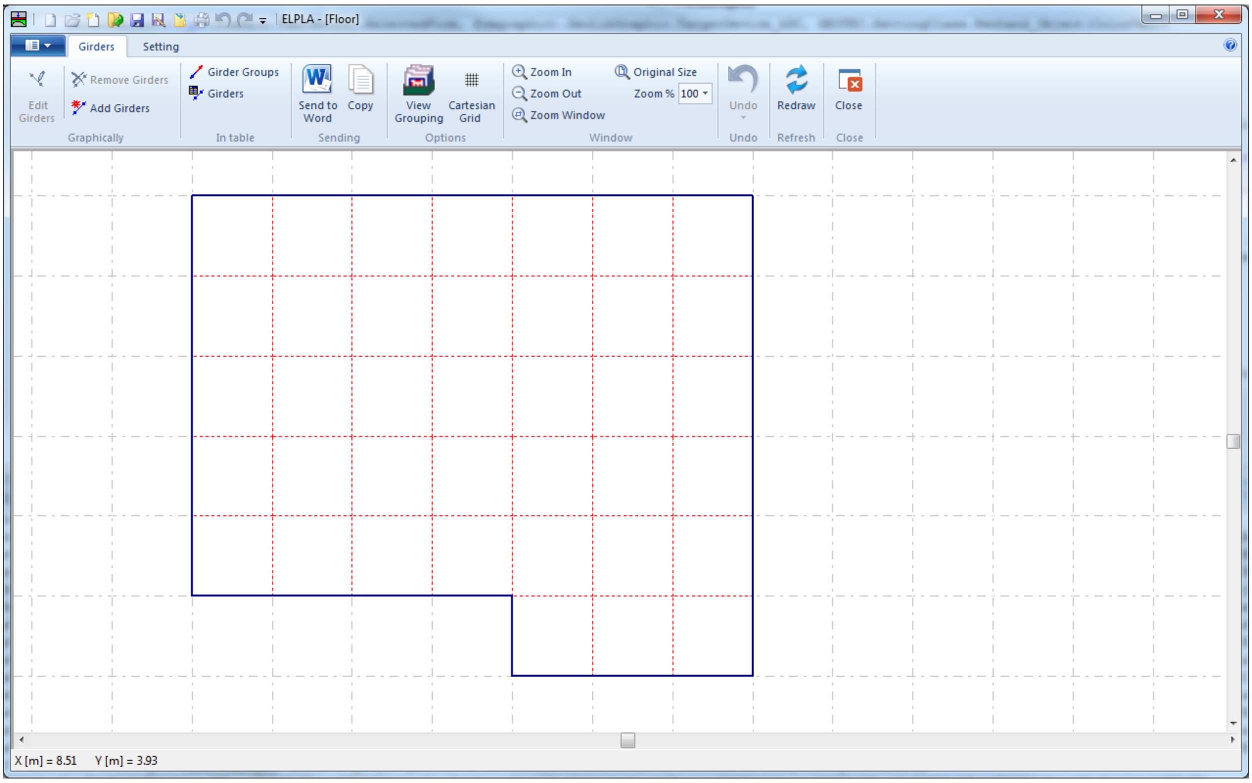

Figure 2.13 "Girders" Window

To enter the cross section of the girders

- Choose "Girder Groups" command from "In table" menu in Figure 2.13. The following option box in Figure 2.14 appears
- In this option box, select the option of cross section definition. Although the cross section of the girder must be defined whether it is T or L girder type, but for simplicity a rectangular cross section is chosen in this example to define the girder cross section
- Click "OK" button

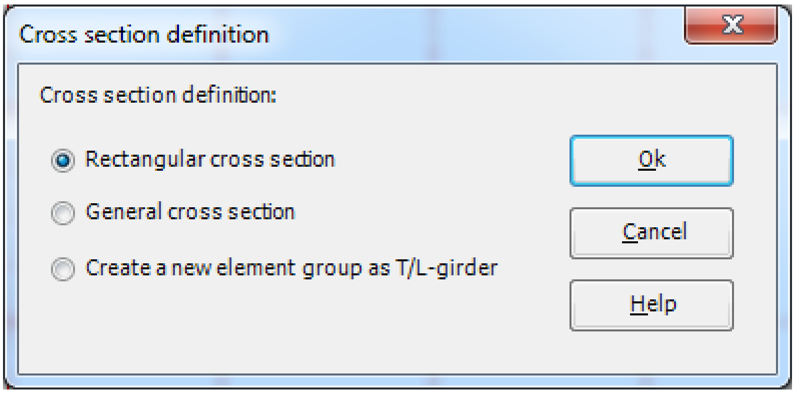

Figure 2.14 "Cross section definition" option box

After clicking "OK" button in the "Cross section definition" option box, the following list box in Figure 2.15 appears.

In this list box

- Enter the material properties of the girder, cross section dimensions and the girder weight as indicated in Figure 2.15. This is done by entering the value in the corresponding cell and press "Enter" button
- Click "OK" button

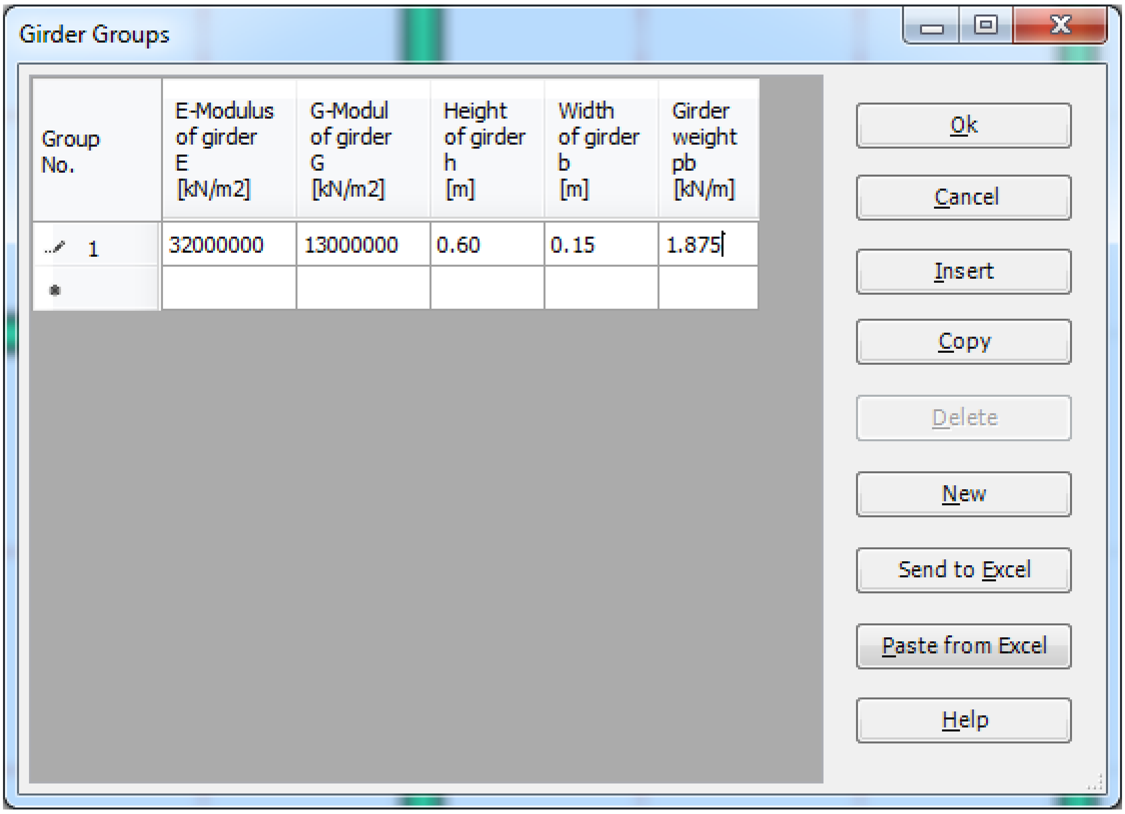

Figure 2.15 "Defining girder groups" list box

Defining the girder locations on the net

Defining girder locations on the net may be carried out either graphically or numerically (in a table). In the current example, the user will learn how to define girder locations on the net graphically.

To define the girder locations on the net graphically

- Choose "Add Girders" command from the "Graphically" menu in Figure 2.13. When "Add Girders" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the start node of the first girder. Then drag the mouse until the end node of that girder (Figure 2.16) and click on the end node. The "Girder elements" dialog box in Figure 2.17 appears

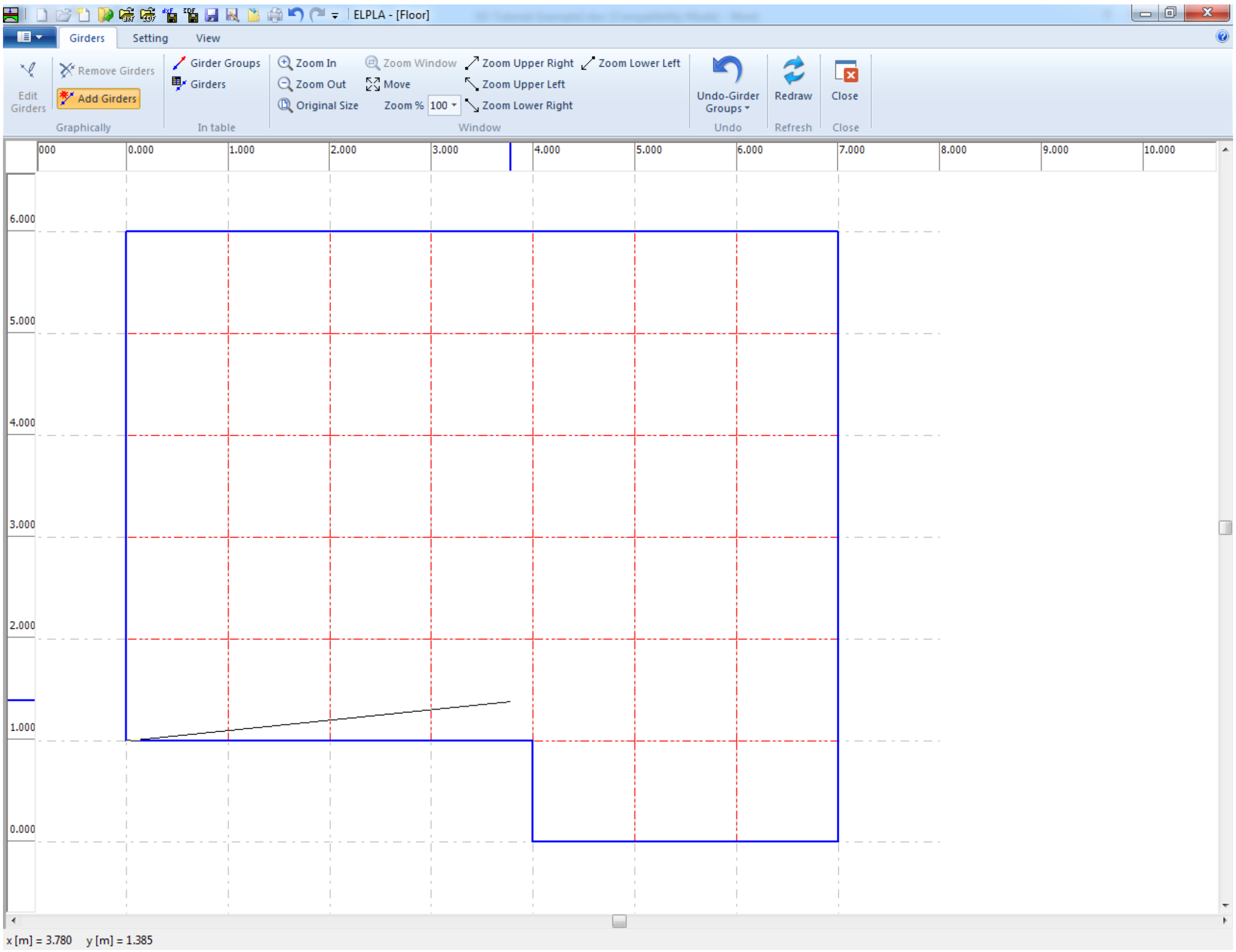

Figure 2.16 Add girder by mouse

In this dialog box, click "OK" button.

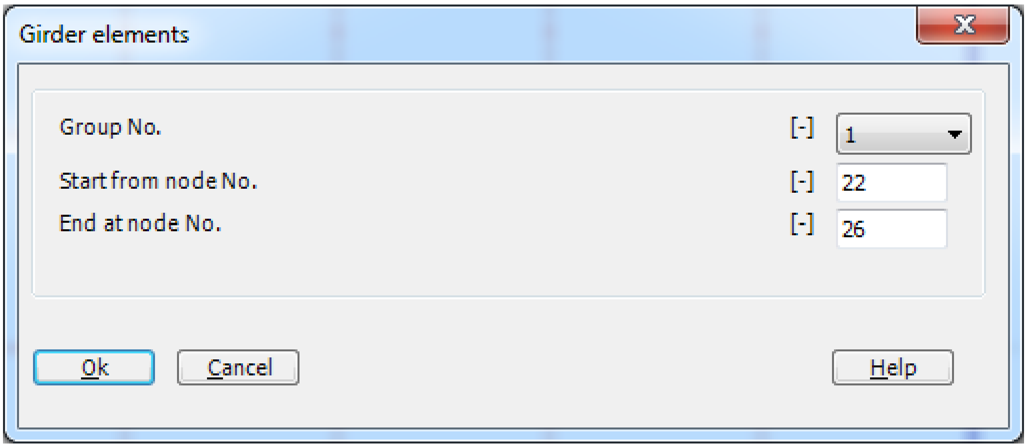

Figure 2.17 "Girder elements" dialog box

Now, the first girder is defined as shown in Figure 2.18. Note that *ELPLA* has typed automatically the girder type on it indicating the No. of girder group.

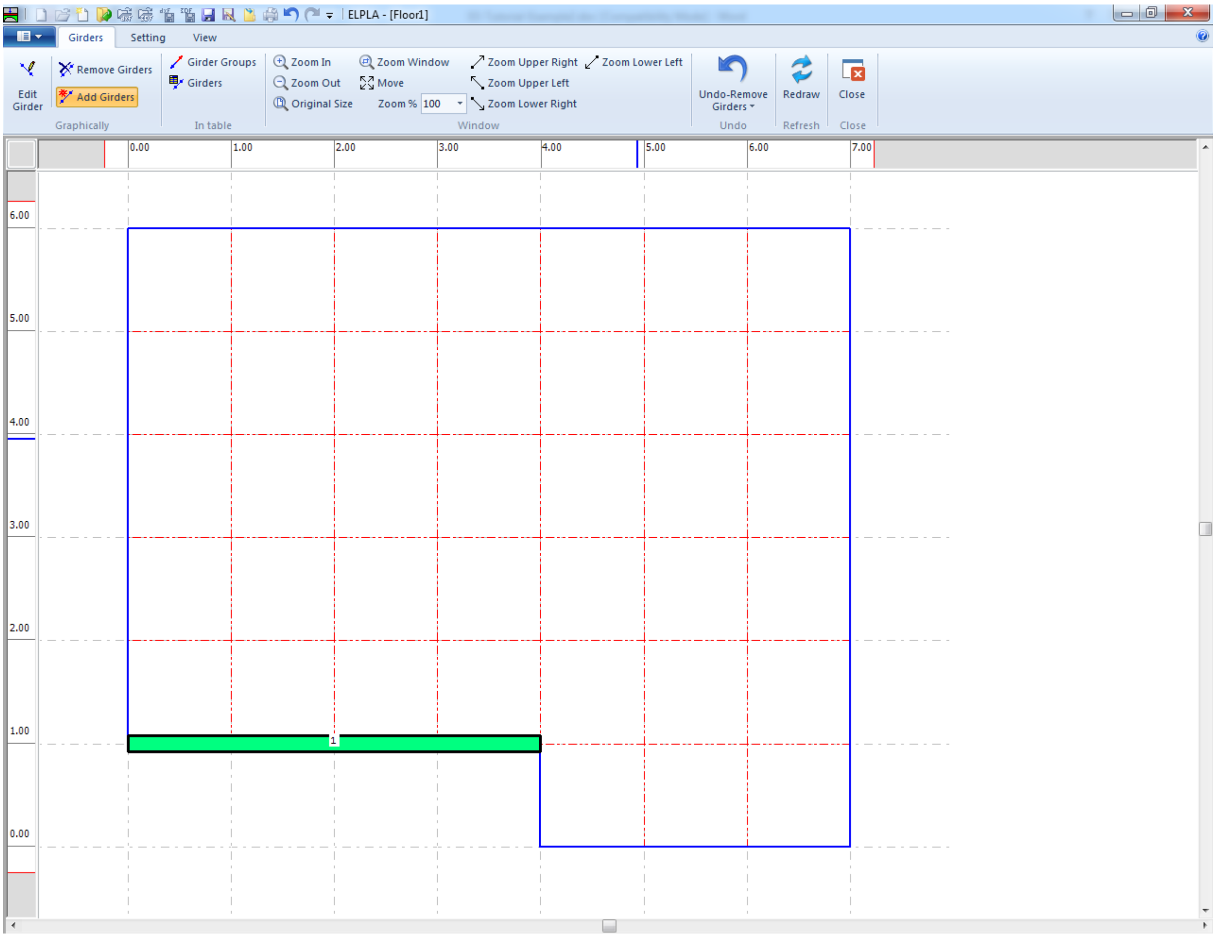

Figure 2.18 First girder

Repeat the previous steps to add the remaining girders on the net. After you have completed the definition of all girders, the screen should look like the following Figure 2.19.

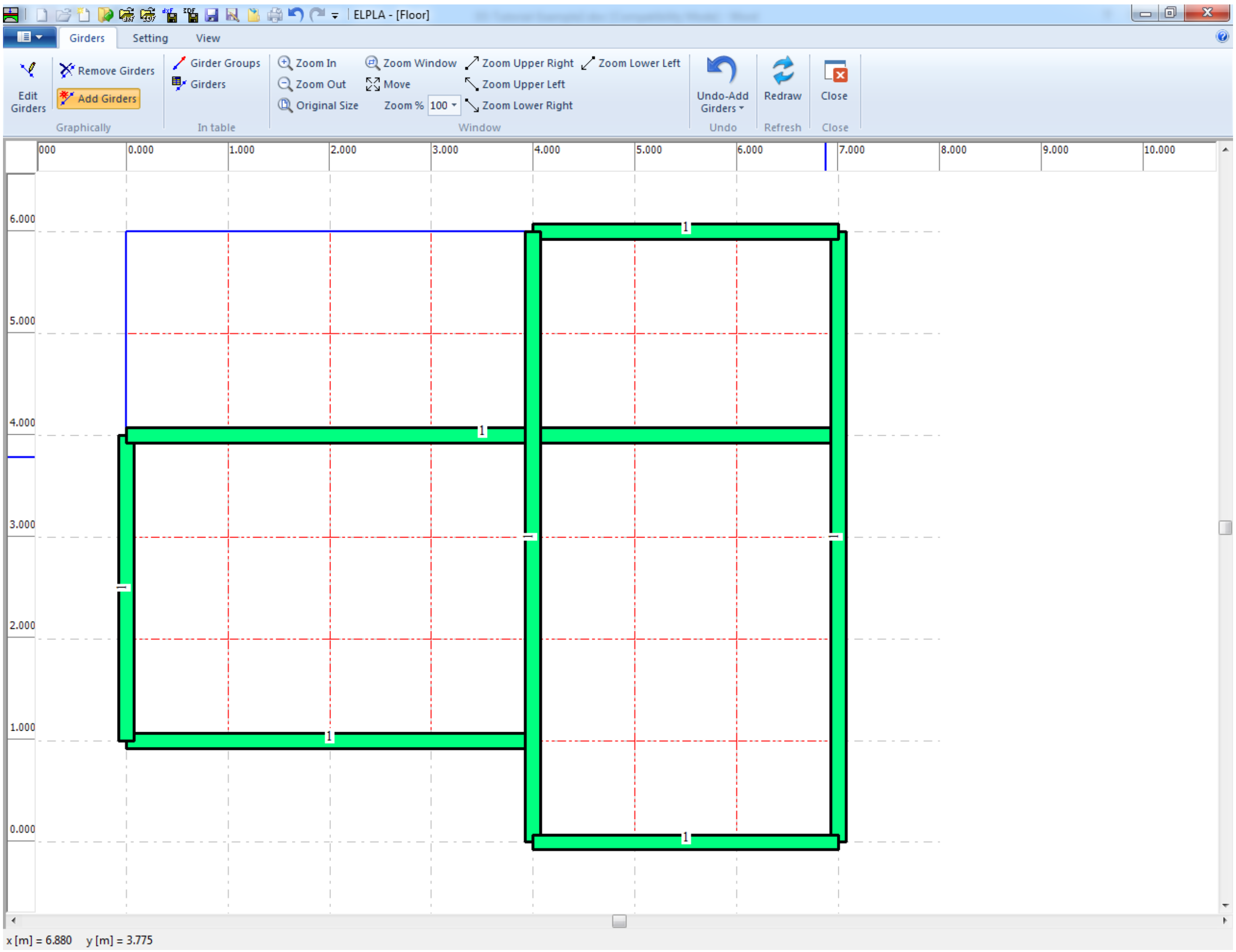

Figure 2.19 Girders

After entering all data and parameters of girders, do the following two steps

- Choose "Save" command from "File" menu in Figure 2.19 to save the data of girders
- Choose "Close" command from "File" menu in Figure 2.19 to close the "Girders" Window and return to ELPLA main window.

2.5 Supports/ Boundary Conditions

In general, columns under the slab are considered as rigid supports. These supports are defined by the "Supports/ Boundary Conditions" command. To define supports choose "Supports/ Boundary Conditions" command from “Data” Tab. The following Tab in Figure 2.20 appears.

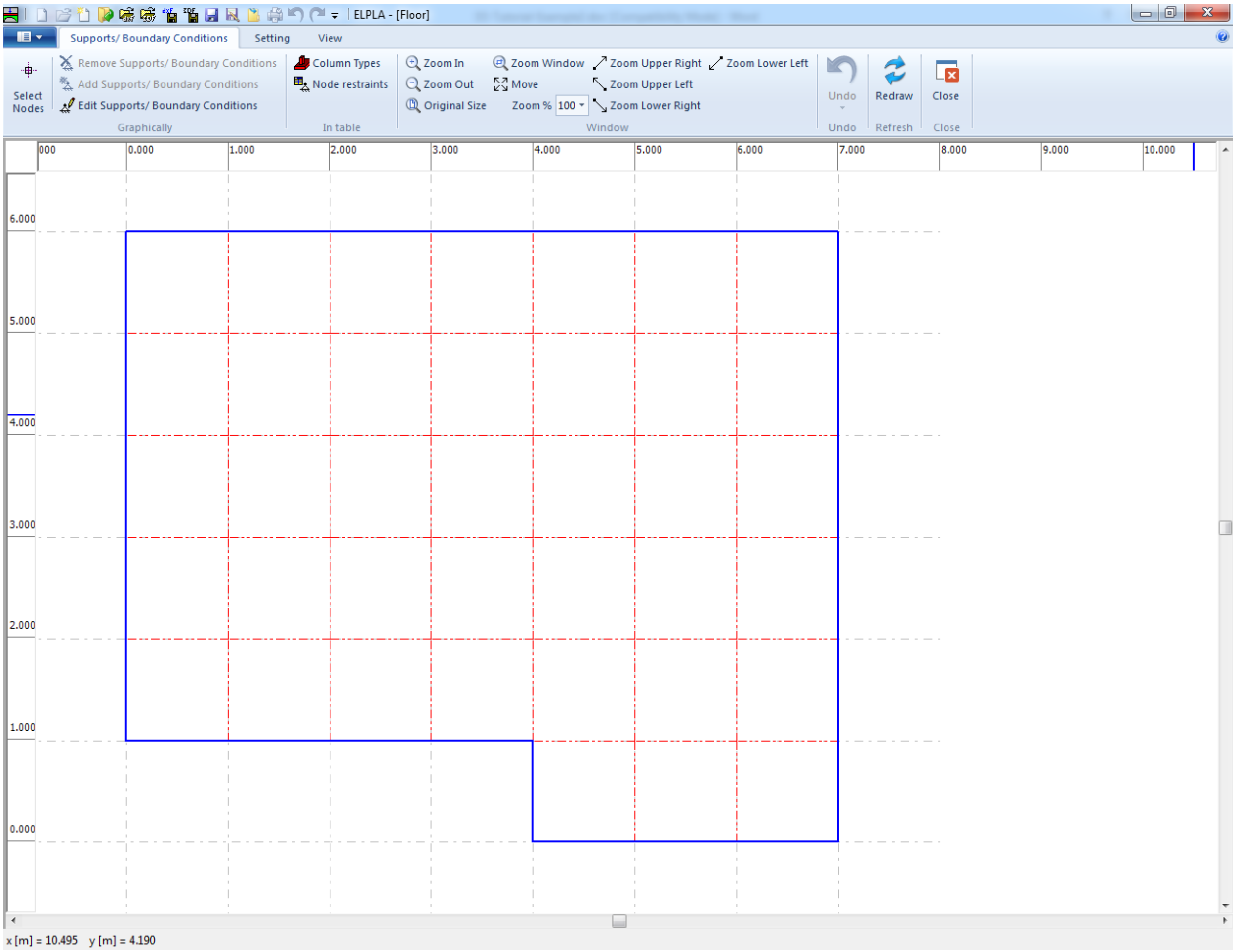

Figure 2.20 "Supports/ Boundary Conditions" Window

ELPLA can display girders, supports, loads, etc. in one view together. The advantage of this option is that the user can control easily locations of supports or loads on the net when entering the rest of the data.

To view the girder on the FE-Net when defining the other data

- Choose "View Grouping" command in “View” menu in Figure 2.20. The "View Grouping" check group box in Figure 2.21 appears
- In this check group box, check "Girder system" check box
- Click "OK" button

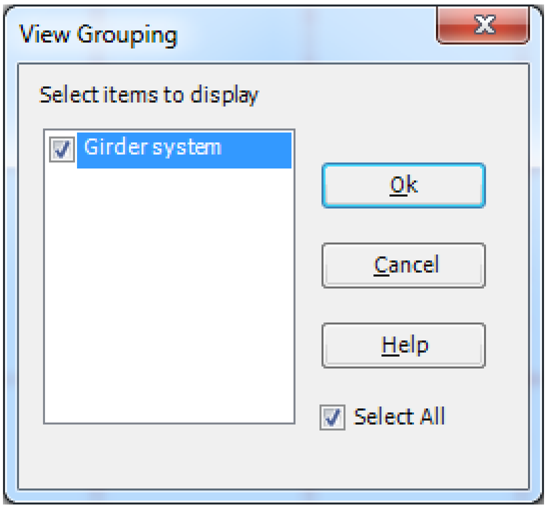

Figure 2.21 "View Grouping" check group box

After clicking "OK" in the "View Grouping" check group box, the screen should look like the following Figure 2.22.

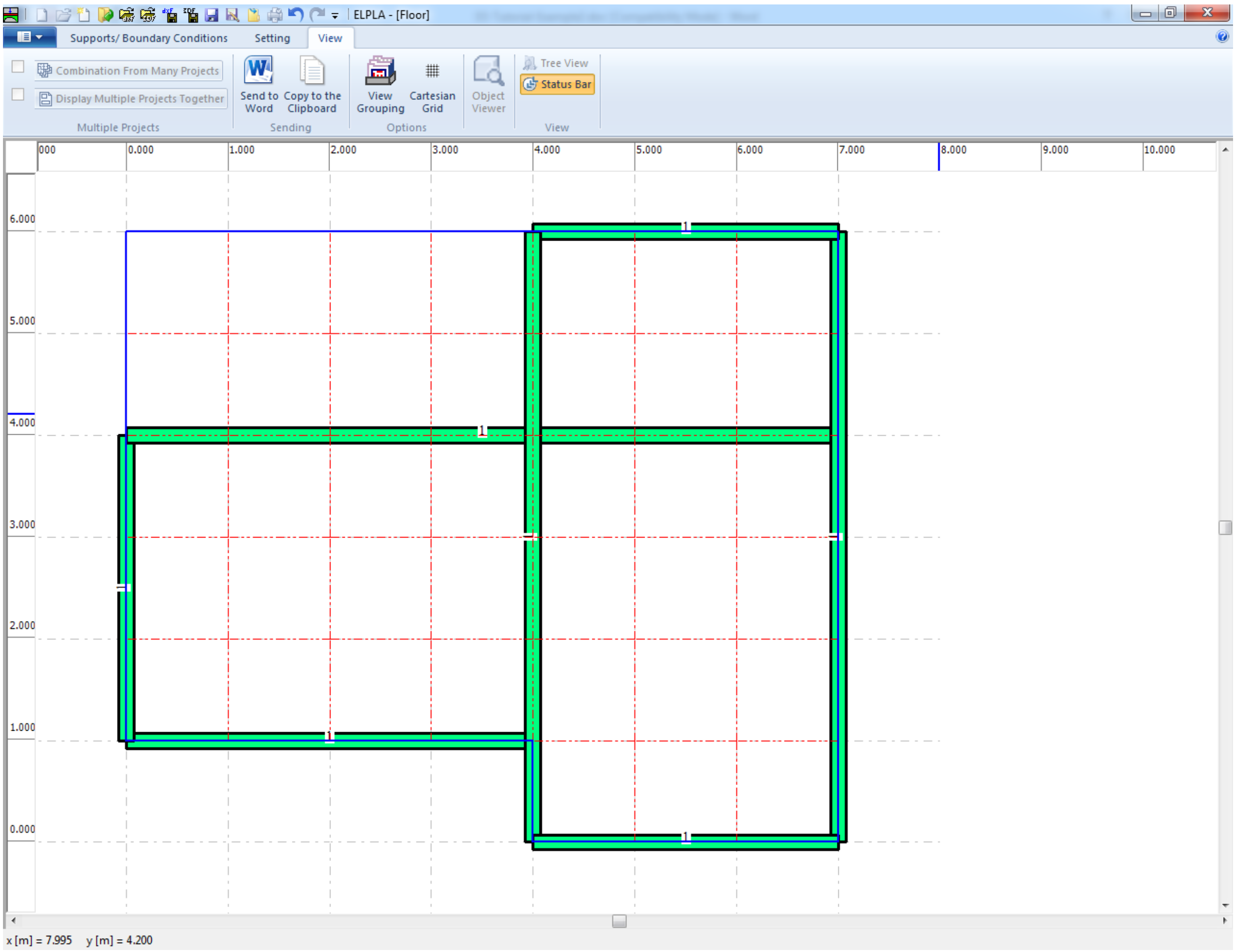

Figure 2.22 Girders in the window of the "Supports/ Boundary Conditions" Window

Defining supports on the net

Defining supports or boundary conditions on the net may be carried out either graphically or numerically (in a table). In the current example, the user will learn how to define supports on the net graphically.

To define supports on the net

- Choose "Select Nodes" command from "Supports/ Boundary Conditions" Tab in Figure 2.22. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nodes that have supports as shown in Figure 2.22
- After selecting nodes of supports, choose "Add Supports/ Boundary Conditions" command from "Graphically" menu (Figure 2.22). The "Supports/ Boundary Conditions" dialog box in Figure 2.24 appears

In this dialog box

- Type 0 in the "Displacement w" edit box to define a rigid support
- Click "OK" button

ELPLA can calculate the punching stresses due to reactions of column supports. 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 supports have column type 1.

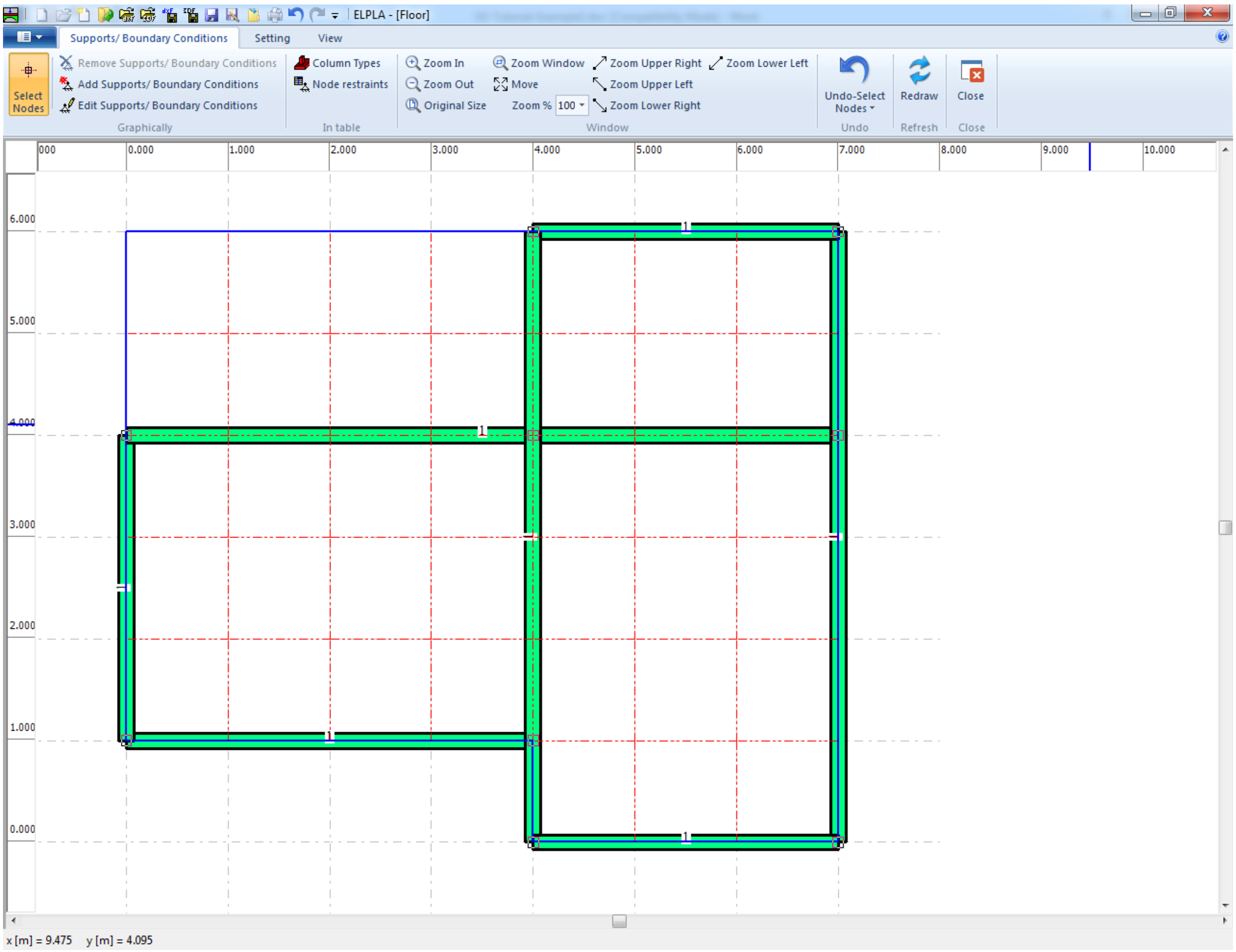

Figure 2.23 Selection of nodes that have supports

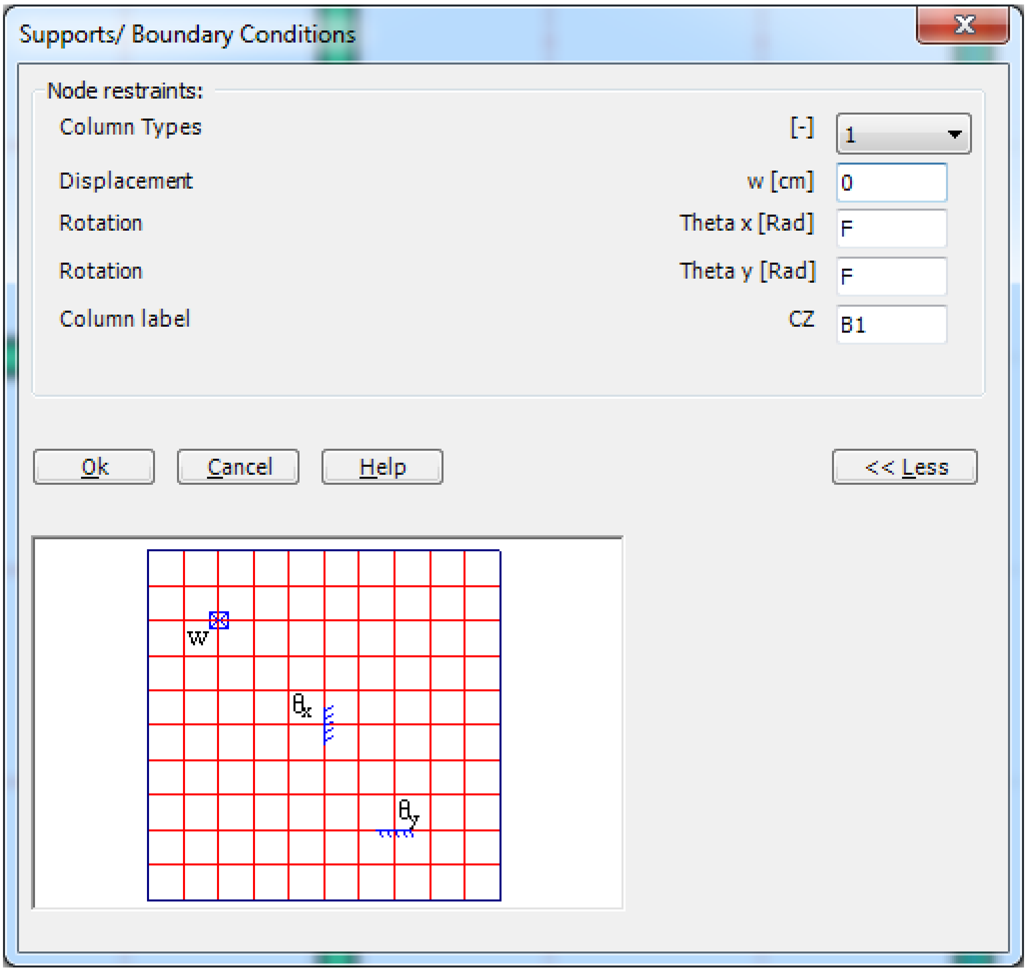

Figure 2.24 "Supports/ Boundary Conditions" dialog box

After you have completed the definition of the supports, the screen should look like the following Figure 2.25.

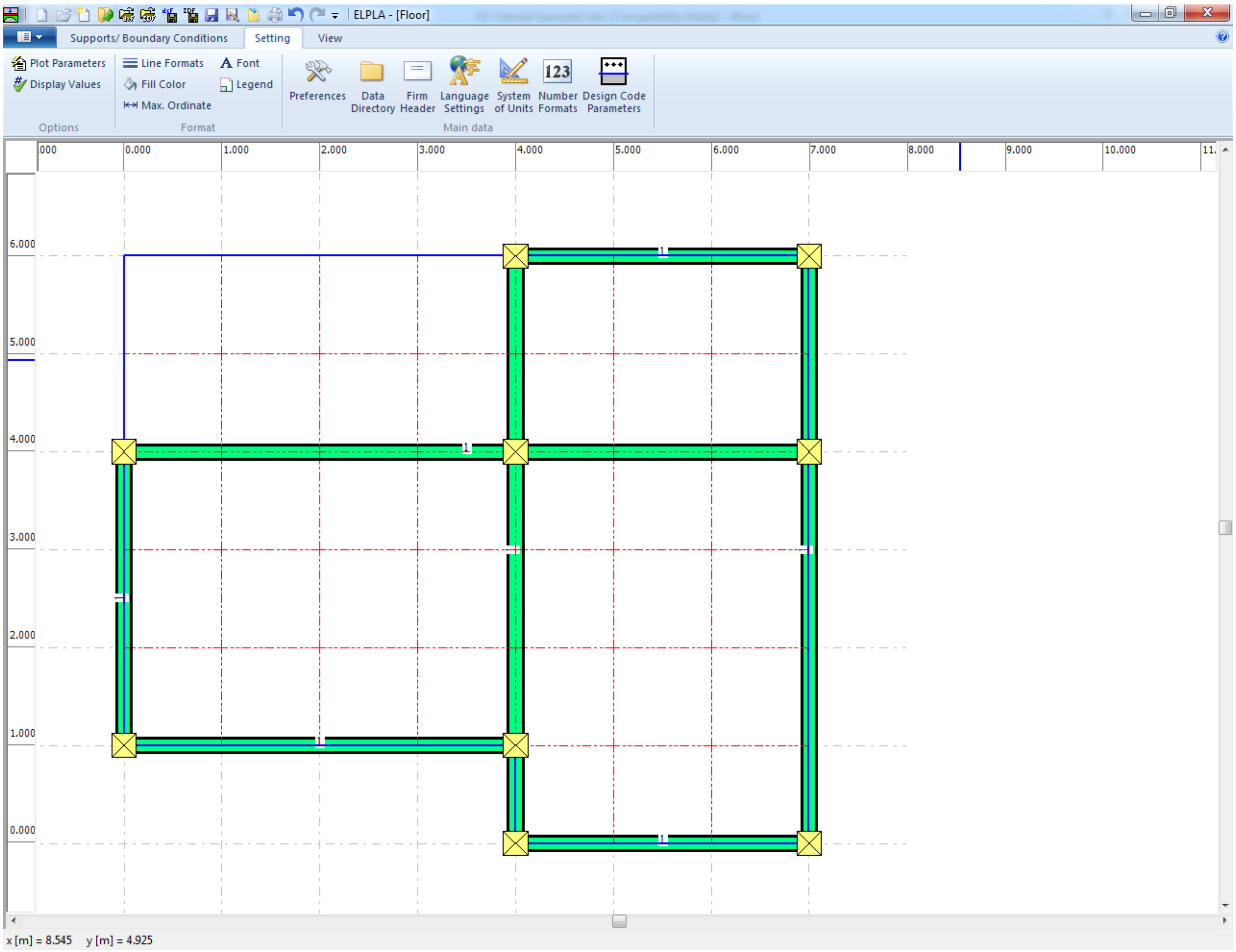

Figure 2.25   9 Supports on the screen

After entering supports, do the following two steps:

- Choose "Save" command from "File" menu in Figure 2.23 to save the data of supports
- Choose "Close" command from "File" menu in Figure 2.23 to close the "Supports/ Boundary Conditions" Window and return to ELPLA main window.

2.6 Slab Properties

To define the slab properties choose "Slab Properties" command from “Data” Tab. The following Window in Figure 2.26 appears with default slab properties. The data of slab properties for the current example, which are required to define, are raft material and slab thickness.

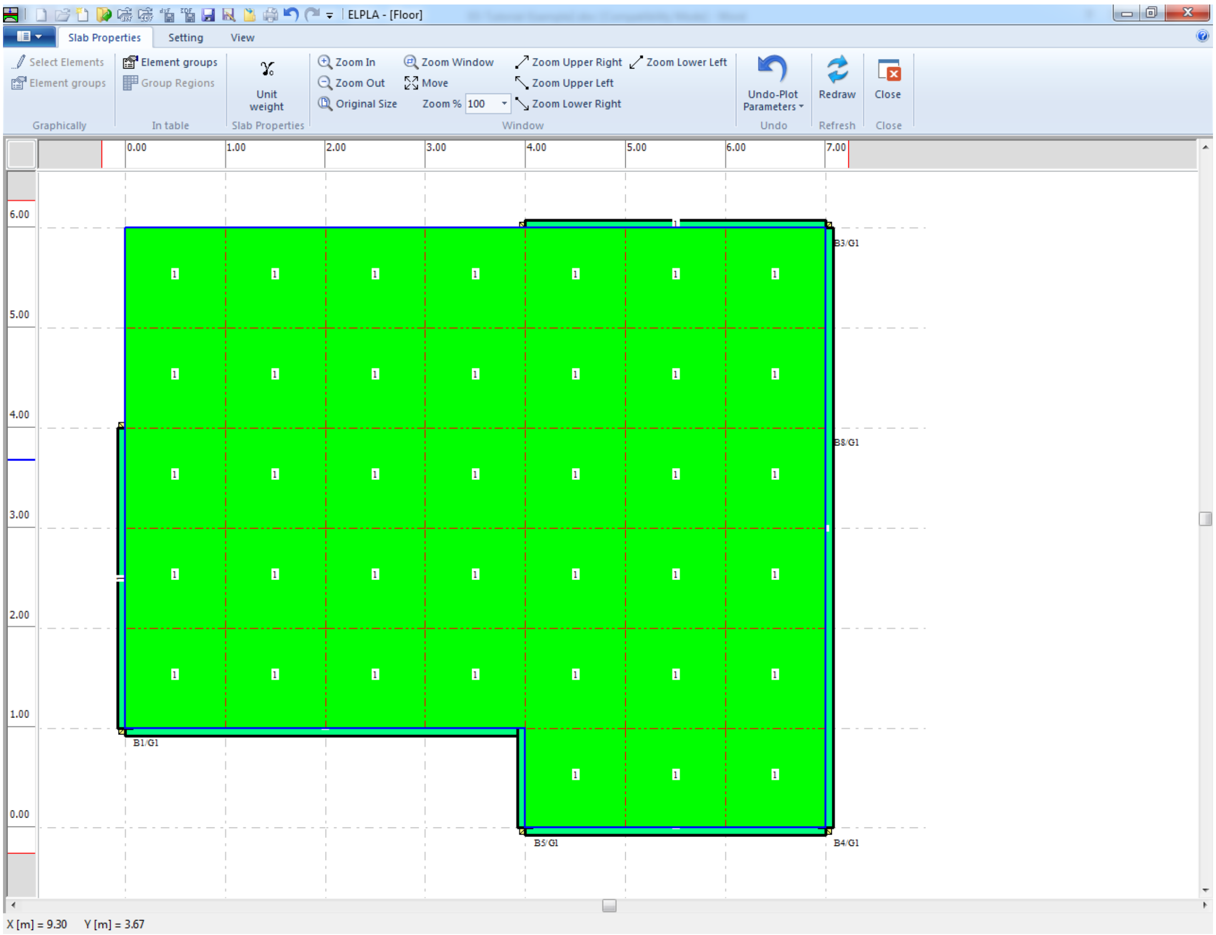

Figure 2.26 "Slab Properties" Window

To enter the slab material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 2.26.
  The following list box in Figure 2.27 with default data appears. To enter or modify a value in this list box, type that value in the corresponding cell and then press "Enter" key. In the list box of Figure 2.27, enter E-Modulus of the slab, *Poisson's* ratio of the slab and the slab thickness
- Click "OK" button

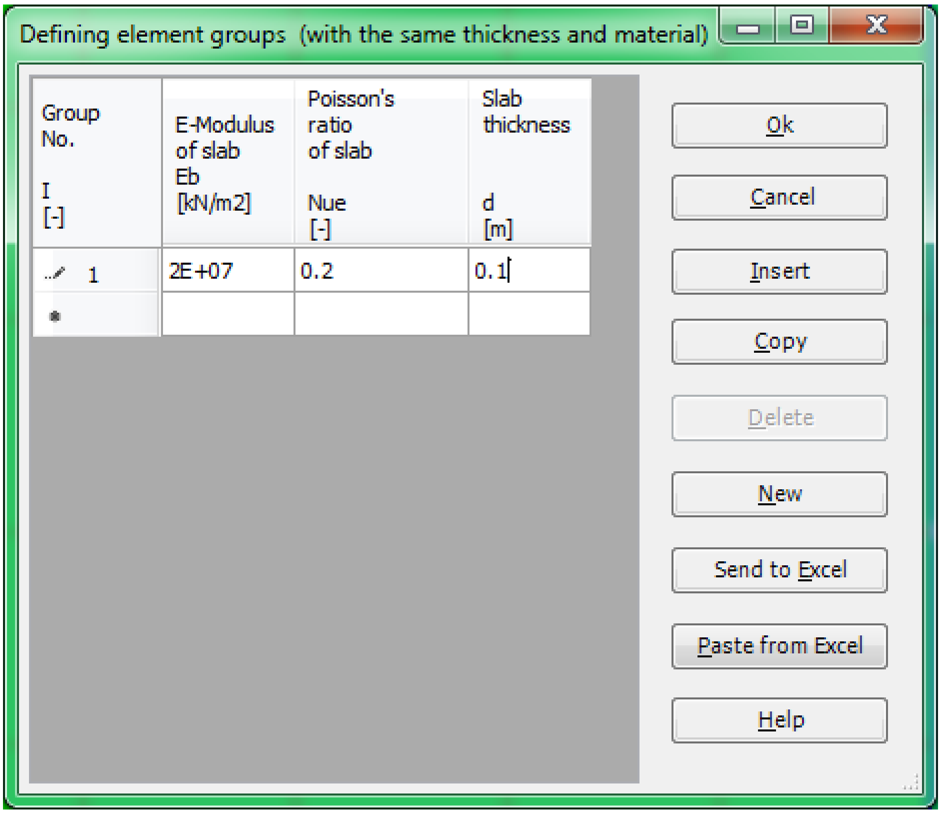

Figure 2.27 "Defining element groups" list box

To enter the unit weight of the slab

- Choose "Unit weight" command from "Slab Properties" menu (Figure 2.26). The following dialog box in Figure 2.28 with a default unit weight of 25 [kN/m^3] appears
- Click "OK" button

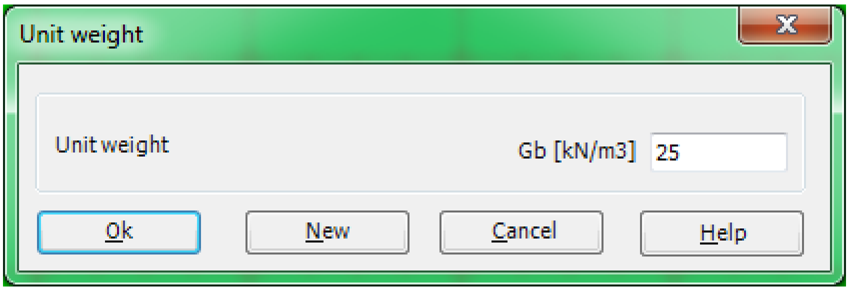

Figure 2.28 "Unit weight" dialog box

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

- Choose "Save" command from "File" menu in Figure 2.26 to save the slab properties
- Choose "Close" command from "File" menu in Figure 2.26 to close the "Slab Properties" Window and return to ELPLA main window.

2.7 Reinforcement Data

The reinforcement of the slab can be carried out according to the design codes EC 2, DIN 1045, ACI and ECP (working stress and limit state design methods). In the current example, the concrete sections of the slab are designed according to EC 2 for Concrete Grade C 30/37 and Steel grade BSt 500. The concrete cover for the slab may be taken as (Figure 2.29):

Top concrete cover +1/2 bar diameter in *x*-direction	d_{1x} = 1.5	[cm]
Bottom concrete cover +1/2 bar diameter in *x*-direction	d_{2x} = 1.5	[cm]
Top concrete cover +1/2 bar diameter in *y*-direction	d_{1y} = 2	[cm]
Bottom concrete cover +1/2 bar diameter in *y*-direction	d_{2y} = 2	[cm]

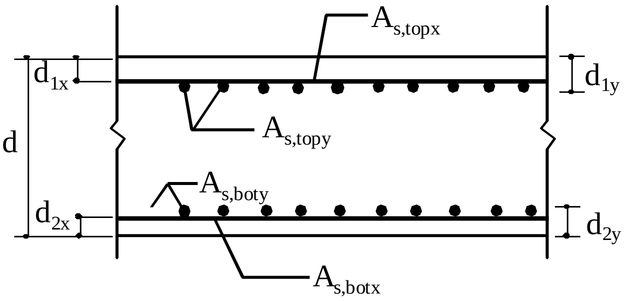

Figure 2.29 Section geometry and reinforcement parallel to *x*-direction

The design code parameters such as partial safety factors for concrete strength, steel strength and internal forces are defined by choosing the "Design Code Parameters" command from "Setting" Tab in *ELPLA*, while reinforcement data such as design code, concrete grade, steel grade and concrete covers are defined by choosing the "Reinforcement" command from “Data” Tab in *ELPLA*. Design code parameters are standard data for all projects while reinforcement data may be varied from one project to another.

To define the reinforcement data, choose "Reinforcement" command from “Data” Tab. The dialog box in Figure 2.30 appears with default reinforcement data.

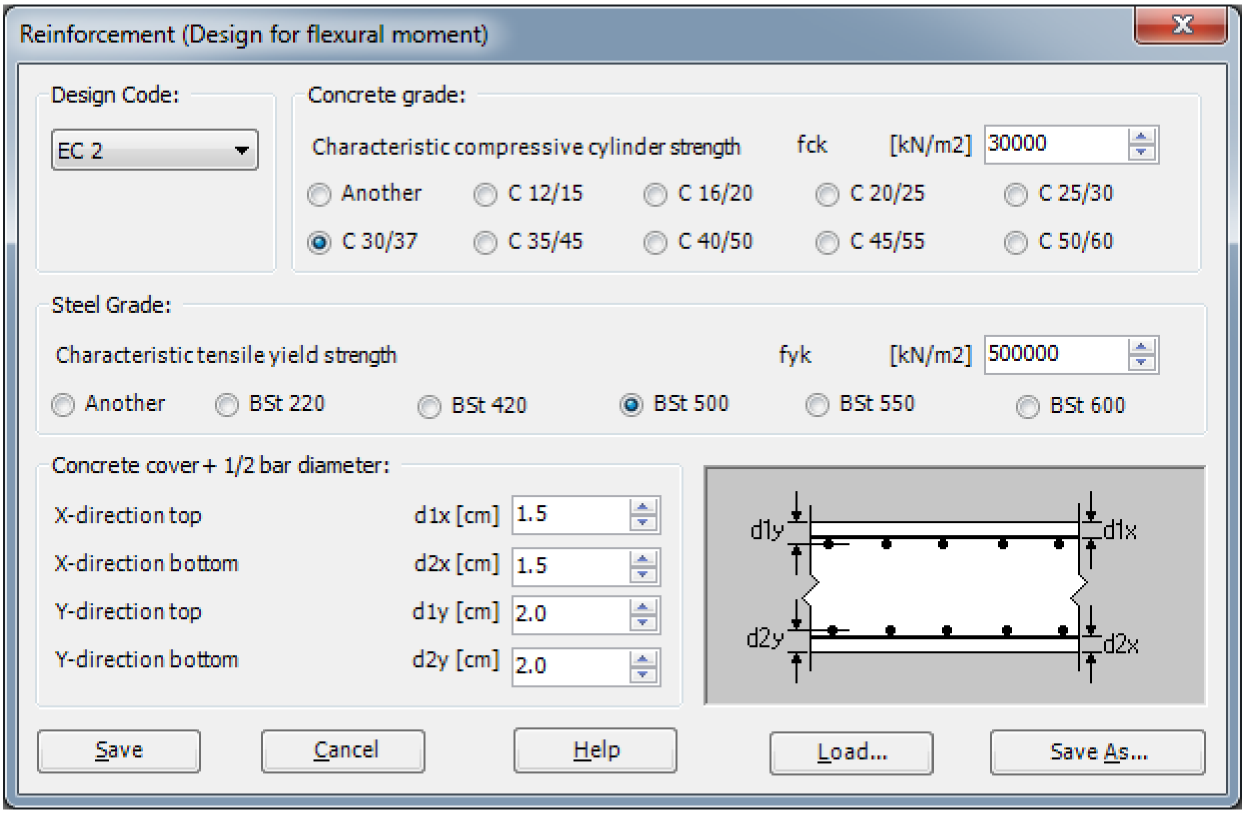

Figure 2.30 "Reinforcement" dialog box

In this dialog box

- Select design code "EC 2" in the "Design code" combo box
- Select steel grade "BSt 500" in the "Steel grade" option box
- Select concrete grade "C 30/37" in the "Concrete grade" option box
- Select the default concrete covers as indicted in the "Concrete cover" dialog group box
- Click "Save" button

2.8 Loads

To define the loads choose "Loads" command from “Data” Tab. The following Window in Figure 2.31 appears with girders on the net.

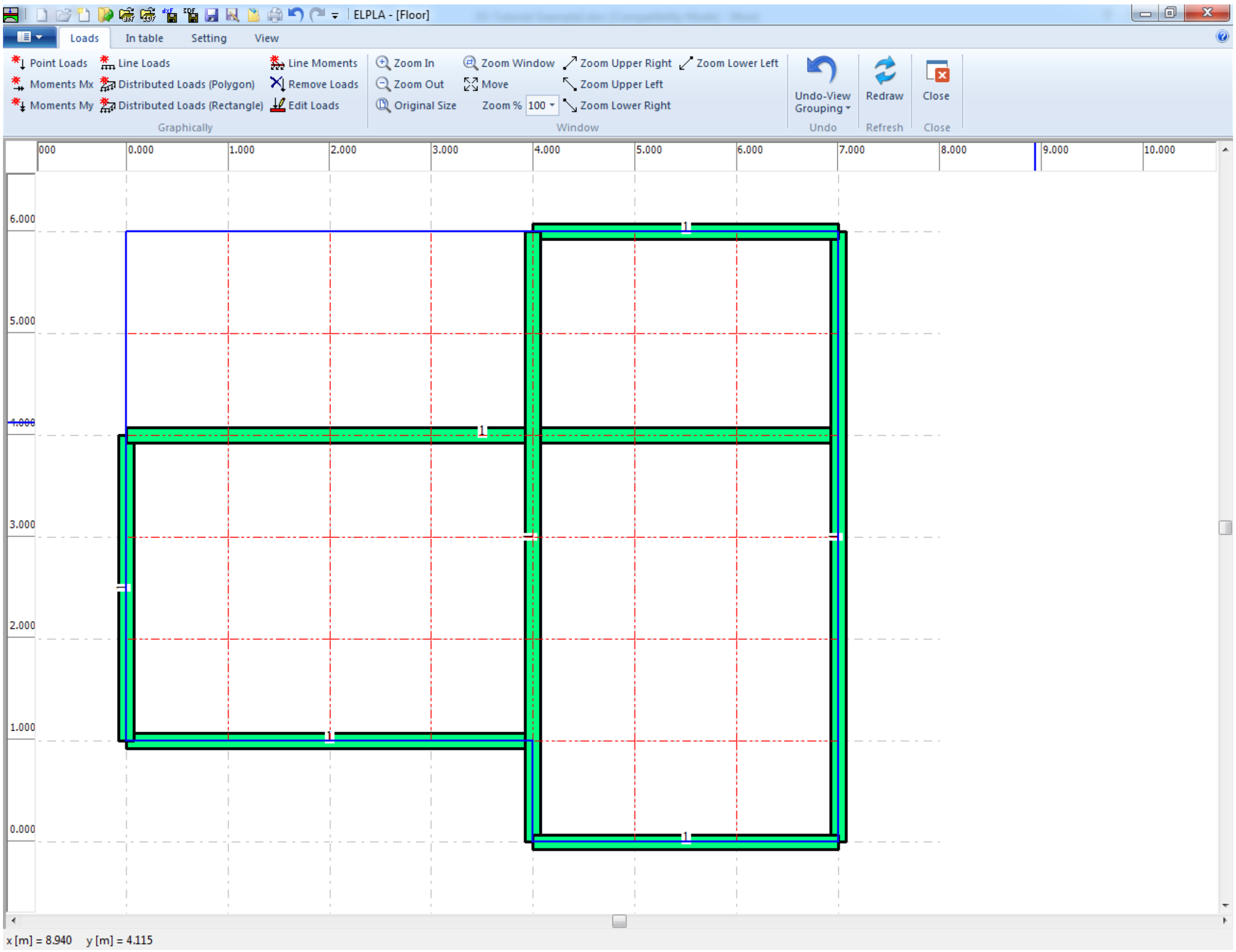

Figure 2.31 "Loads" Window

Defining loads on the net may be carried out either graphically or numerically (in a table). In the current example, the user will learn how to define loads on the net graphically.

To enter the first distributed loads

- Choose "Distributed Loads (Rectangle)" command from "Graphically" menu in Figure 2.31. When "Distributed Loads (Rectangle)" command is chosen, the cursor is changed from an arrow to a cross hair. Then the load can be defined by holding the left mouse button down at the starting point of the distributed load. As the mouse is dragged, a box appears, indicating a distributed load is being defined. When the left mouse button is released, the following dialog box in Figure 2.32 appears with the load value and coordinates

In this dialog box

- Type 3.5 in the "Load value" edit box
- Click "OK" button

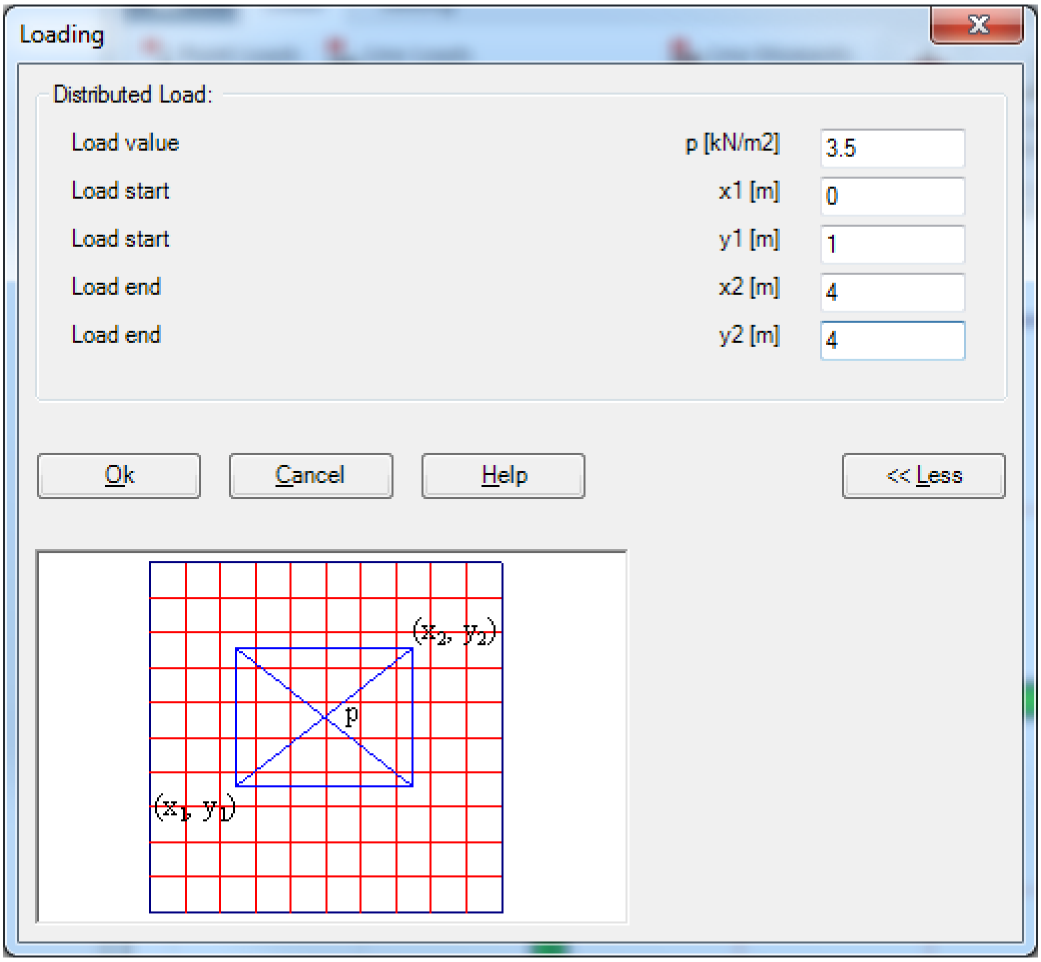

Figure 2.32 "Loading" dialog box

After you have completed the definition of the first distributed load, the screen should look like the following Figure 2.33.

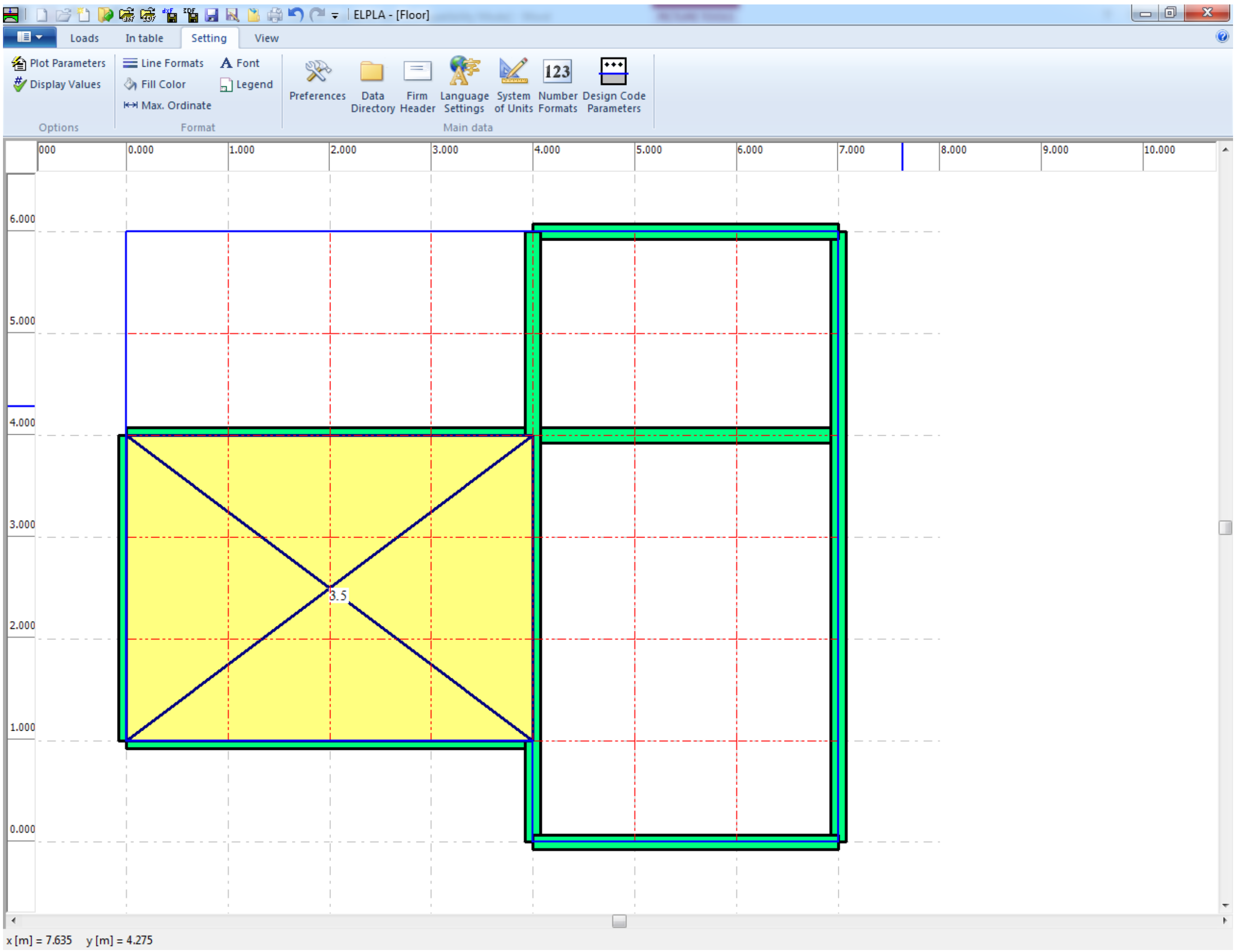

Figure 2.33 First distributed load

Repeat the previous steps to enter the remaining distributed loads on the net. After you have completed the definition of all loads on the net, the screen should look like the following Figure 2.34.

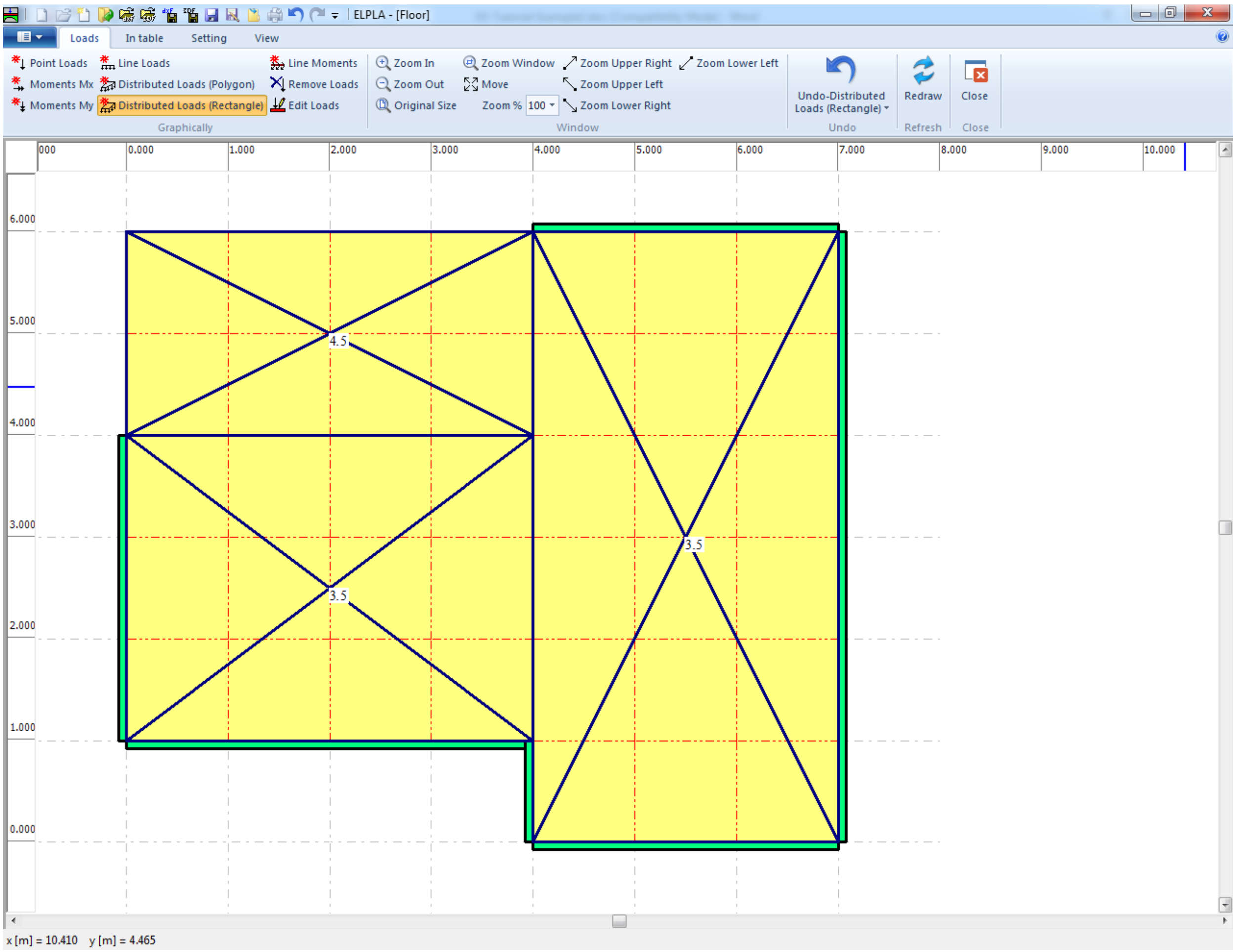

Figure 2.34 Loads on the FE-Net

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

- Choose "Save" command from "File" menu in Figure 2.34 to save the load data
- Choose "Close" command from "File" menu in Figure 2.34 to close the "Loads" Window and return to ELPLA main window.

Creating the project of the slab floor 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 2.36.

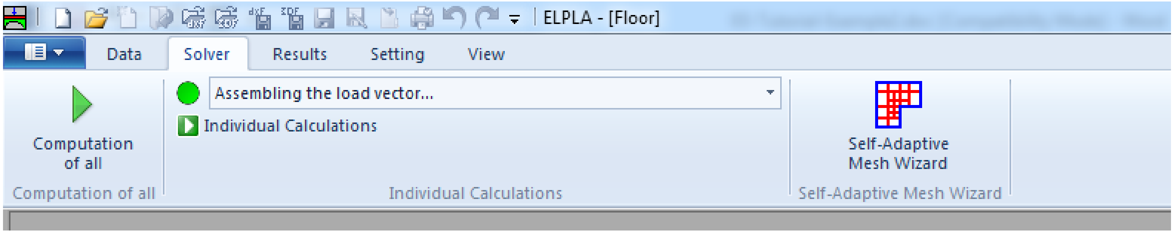

Figure 2.35 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of the calculation depend on the analysis type. For the current example, the items, which are required to be calculated, are:

- Assembling the load vector
- Assembling the girder stiffness matrix
- Assembling the slab stiffness matrix
- Solving system of linear equations (band matrix)
- Determining deformation, internal forces
- Design of the slab

Carrying out all computations

To carry out all computations in one time, choose "Computation of all" command from "Calculation" menu in "Solver" Tab. The progress of all computations according to the defined analysis will be carried out automatically with displaying information through menus and messages.

Analysis progress

Analysis progress menu in Figure 2.36 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of "Solver" Tab window displays information about the progress of calculation.

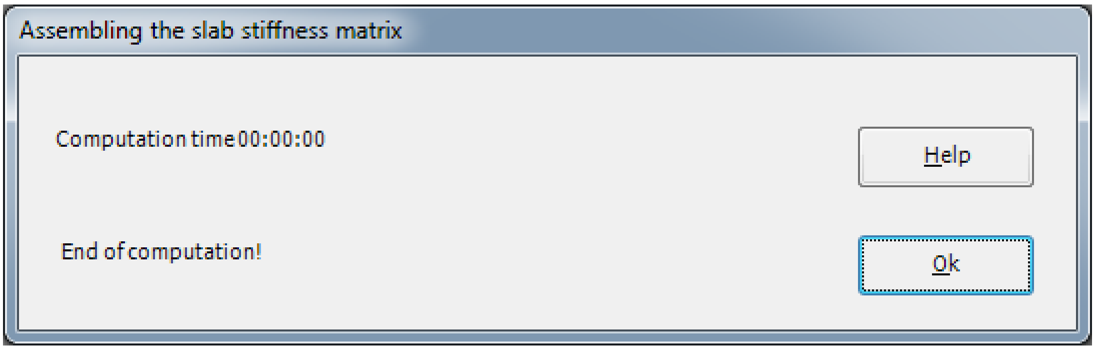

Figure 2.36 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution in Figure 2.37 appears. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

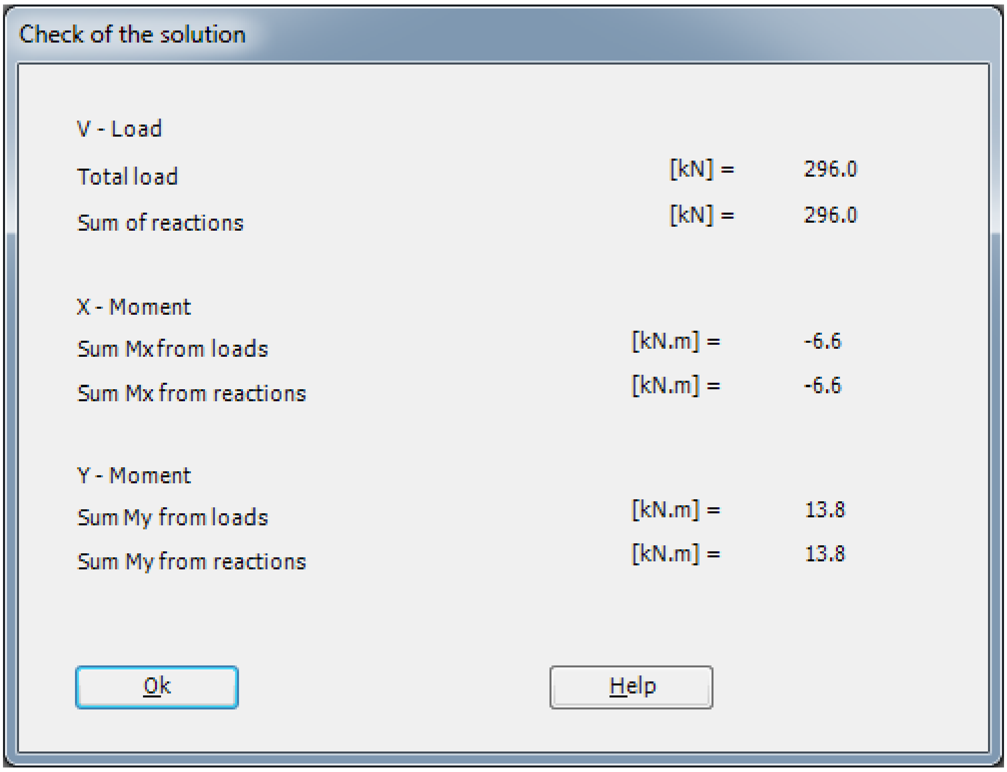

Figure 2.37 Menu "Check of the solution"

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

4 Viewing data and result

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

To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab, Figure 2.38.

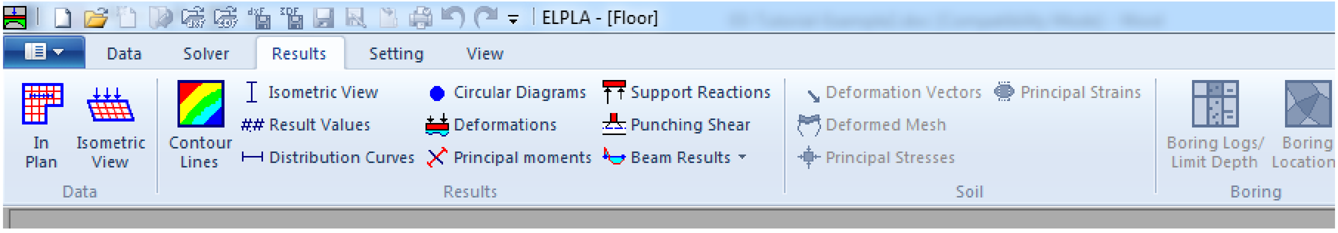

Figure 2.38 "Results" Tab

To view the results of girders, choose "Beam Results" command and then the "Isometric View" command from "Results" group command. The following option box in Figure 2.39 appears.

In this option box

- Select "Beam-bending moments *Mb*" as a sample for the results to be displayed
- Click "OK" button

The moments are now displayed for the girders as shown in Figure 2.40.

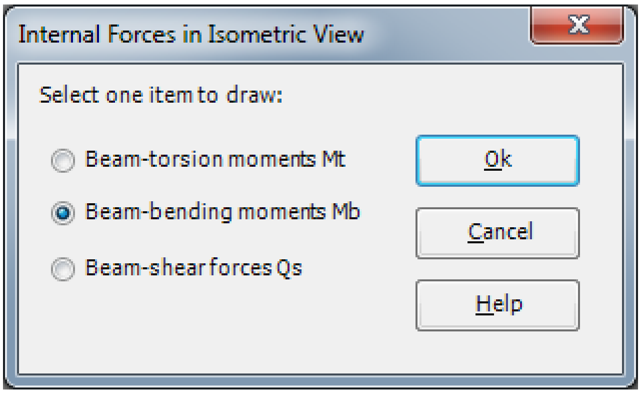

Figure 2.39 "Distribution of internal forces in isometric view" option box

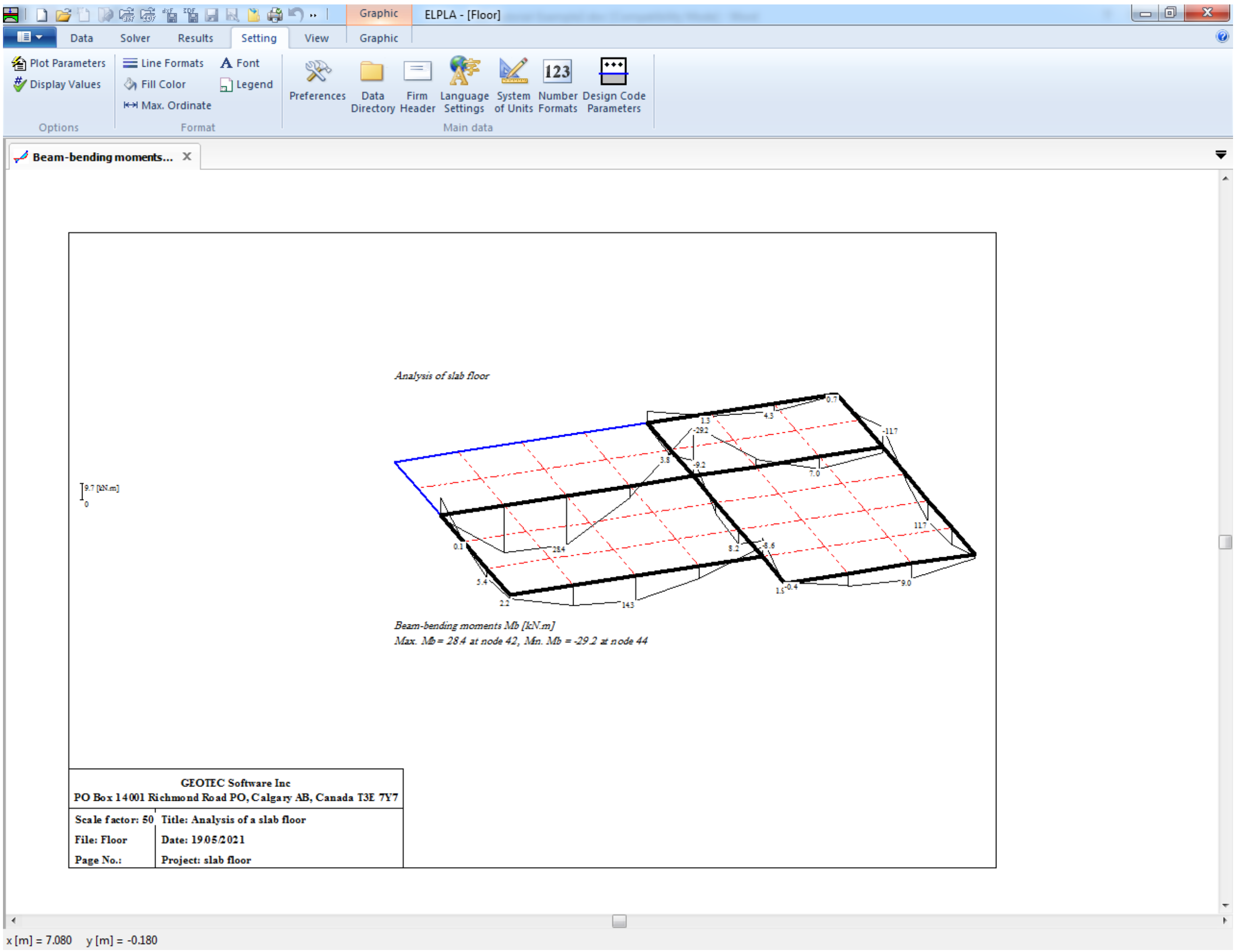

Figure 2.40 Beam-bending moments *Mb*

5 Index