

Example 2

Analysis of a slab floor

Contents		Page
1	Description of the problem	3
	1.1 Loads and dimensions	3
	1.2 Slab material	3
	1.3 Analysis and concrete design	4
2	Creating the project	4
	2.1 Calculation method	4
	2.2 Project identification	7
	2.3 FE-Net data	8
	2.4 Girders	12
	2.5 Supports/ Boundary conditions	16
	2.6 Slab properties	20
	2.7 Reinforcement data	21
	2.8 Loads	23
3	Carrying out the calculations	25
	3.1 Starting <i>ELPLA-Solver</i>	25
	3.2 Carrying out all computations	26
4	Viewing data and result graphics	27
5	Index	30

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 may be taken as 1.875 [kN/m].

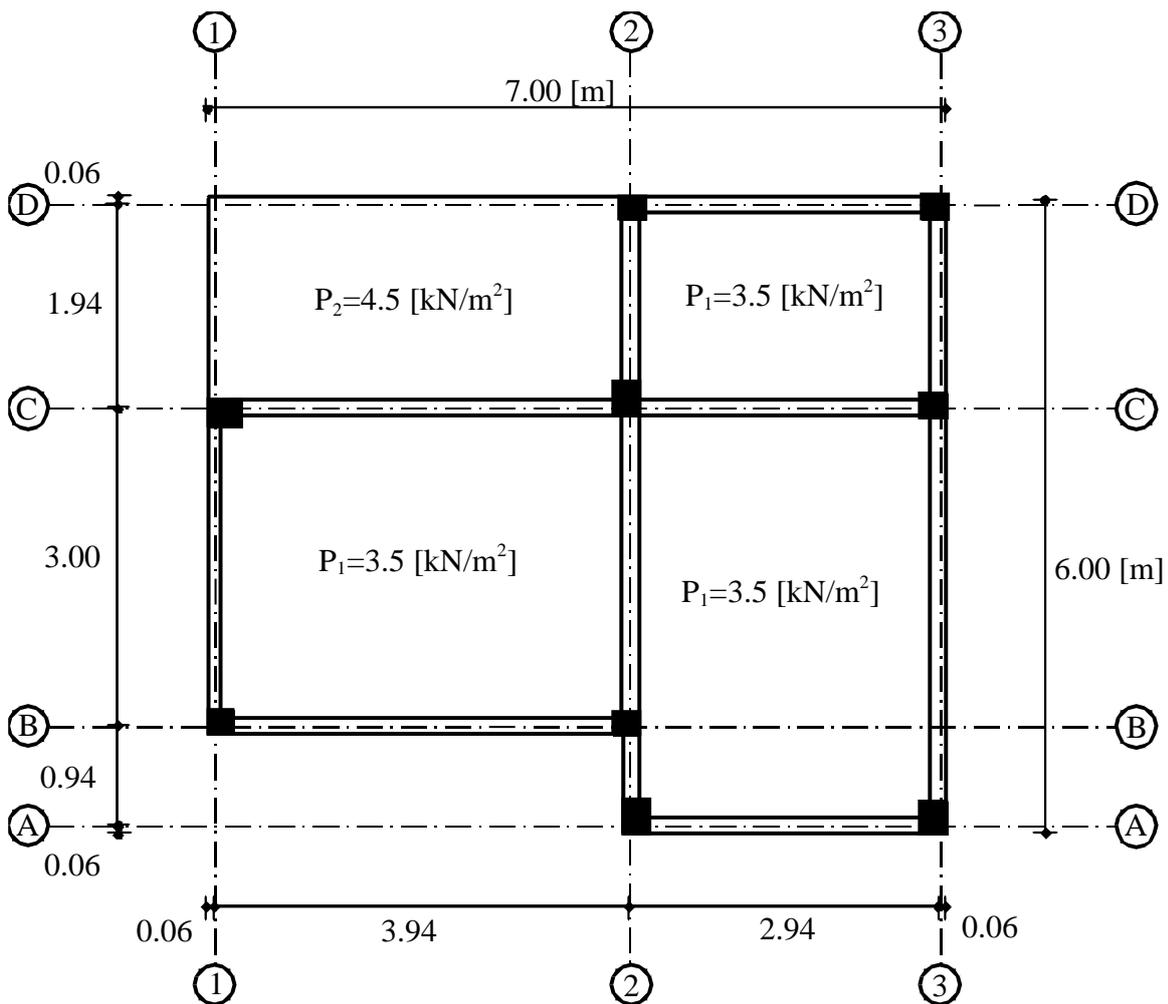


Figure 2-1 Dimensions of the slab with loads

1.2 Slab material

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

Young's modulus of concrete	E_b	= $3.2 * 10^7$	[kN/m ²]
Poisson's ratio of concrete	ν_b	= 0.20	[-]
Unit weight of concrete	γ_b	= 25	[kN/m ³]
Shear modulus of concrete	$G_b = 0.5 E_b (1 + \nu_b)$	= $1.3 * 10^7$	[kN/m ²]

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/m ²]
Characteristic tensile yield strength of reinforcement f_{yk}	= 500		[MN/m ²]
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/m ²]
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/m 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 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 step by step to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

2.1 Calculation method

To create the project start the sub program *ELPLA-Data* 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.

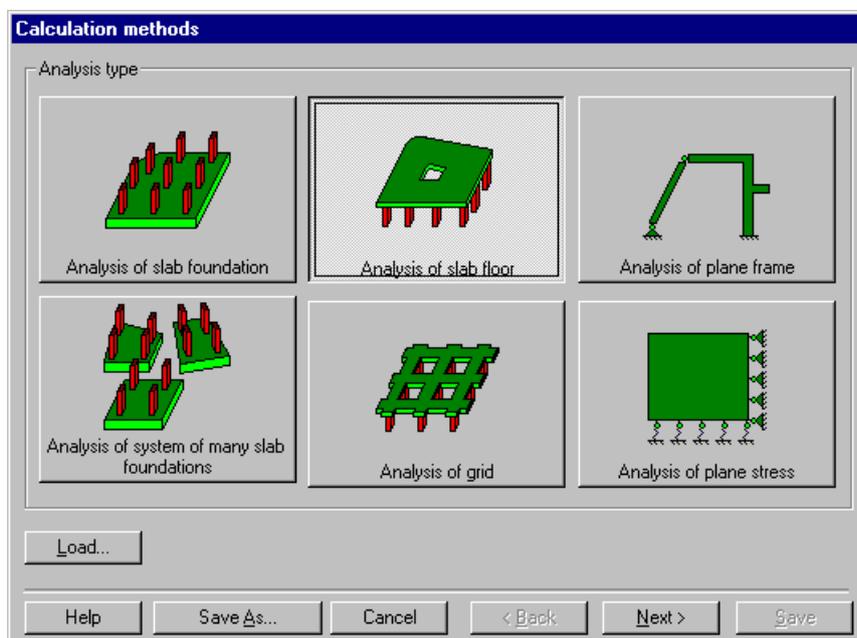


Figure 2-2 "Calculation method" wizard with "Analysis type" form

In the "Analysis type" form in Figure 2-2, define the analysis type of the problem where *ELPLA* can deal with different structural systems. 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

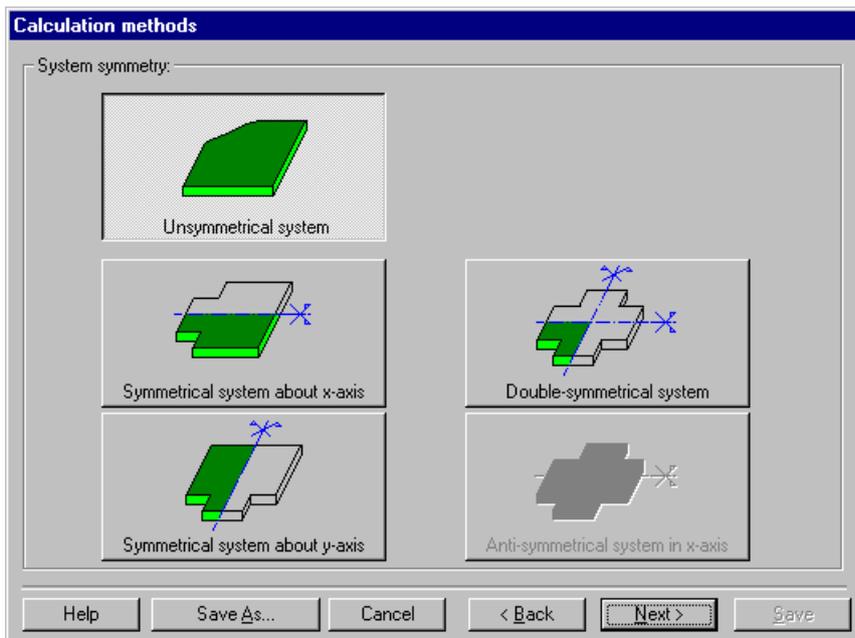


Figure 2-3 "System symmetry" form

The last form of the wizard assistant contains the "Option" 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

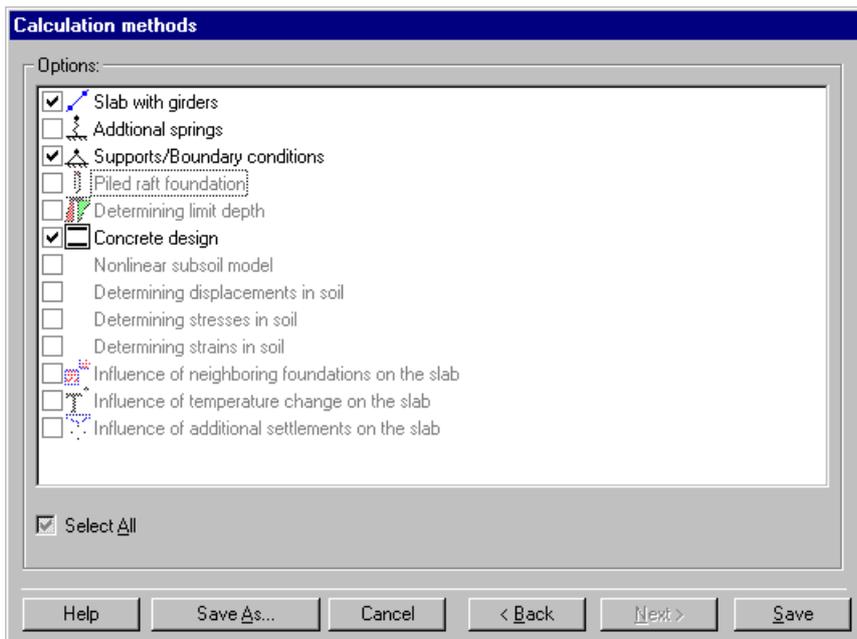


Figure 2-4 "Options" list

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

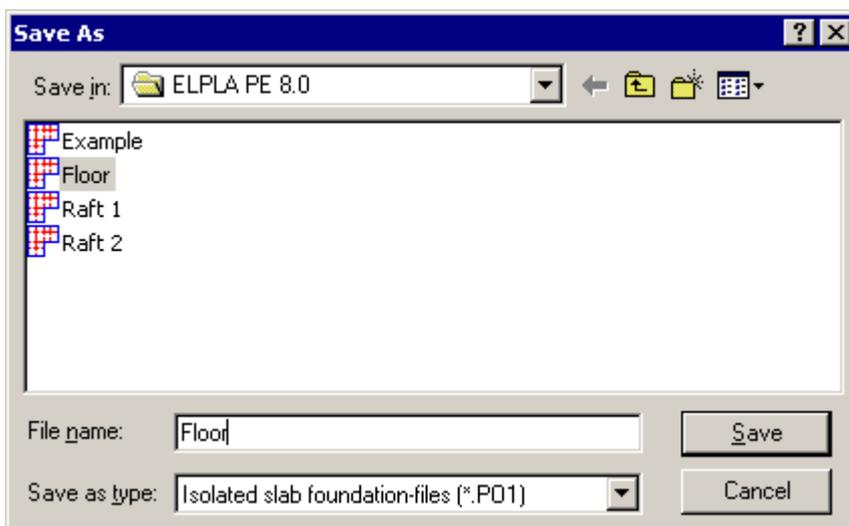


Figure 2-5 "Save as" dialog box

ELPLA will activate the "Data" menu. Also the file name of the current project [Floor] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 2-6.

In the "Data" menu, the user can enter the remaining data of the project using the same sequence of commands in this menu. The first command in the menu is "Calculation methods", which has been already entered. Therefore, *ELPLA* has put the sign "√" beside this command, Figure 2-6.

ELPLA puts this sign beside the commands those the user has entered so that the user can know which data were defined.

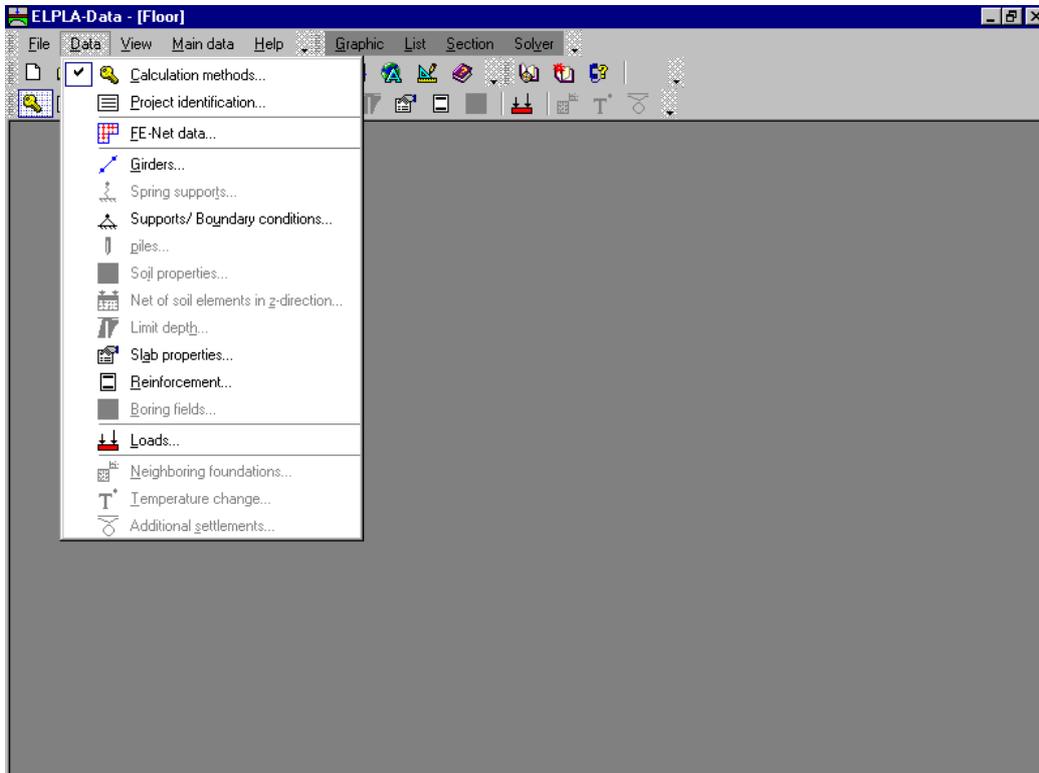


Figure 2-6 ELPLA-Data after defining the calculation method

2.2 Project identification

To identify the project choose "Project identification" command from "Data" menu of ELPLA-Data. The dialog box in Figure 2-7 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

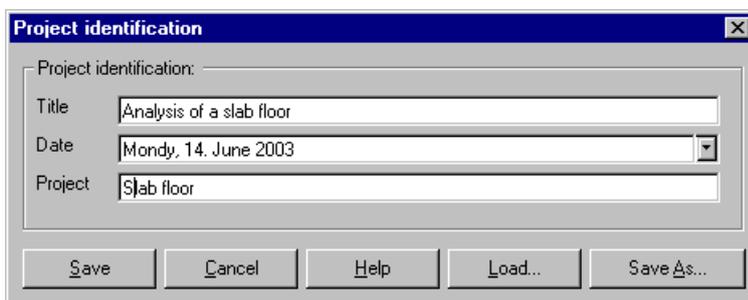


Figure 2-7 "Project identification" dialog box

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" menu. The "FE-Net generation" wizard appears as shown in Figure 2-8. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 2-8, 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

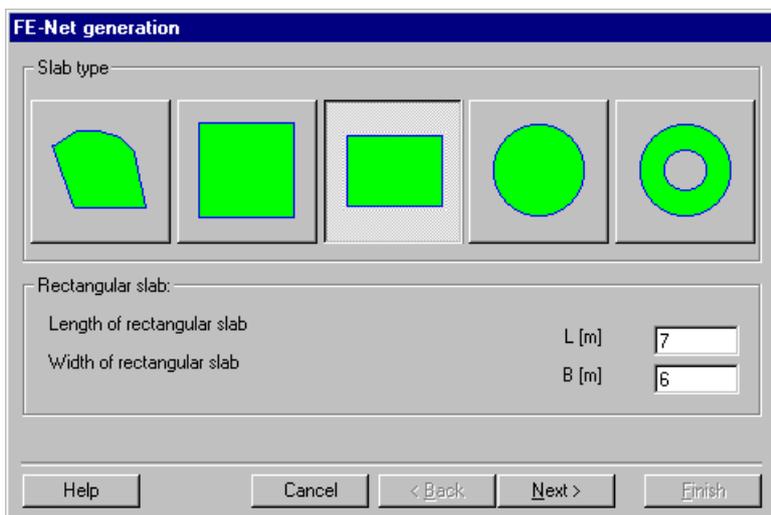


Figure 2-8 "FE-Net generation" wizard with "Slab type" form

After clicking "Next" in the "Slab type" form, the following "Generation type" form appears, Figure 2-9. *ELPLA* can deal with various types of generation with triangle and/ or rectangular elements.

In the "Generation type" form

- Choose rectangular elements
- Click "Next"

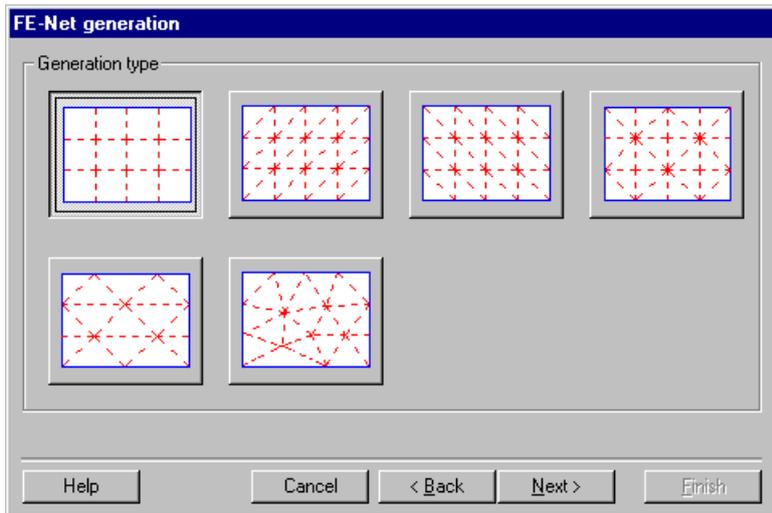


Figure 2-9 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "Grid definition" dialog box in Figure 2-10 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 spaces" edit box
- In "Grid in *y*-direction" frame, type 6 in the "No. of grid spaces" 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 embedded program in Figure 2-11 appears with the generated net.

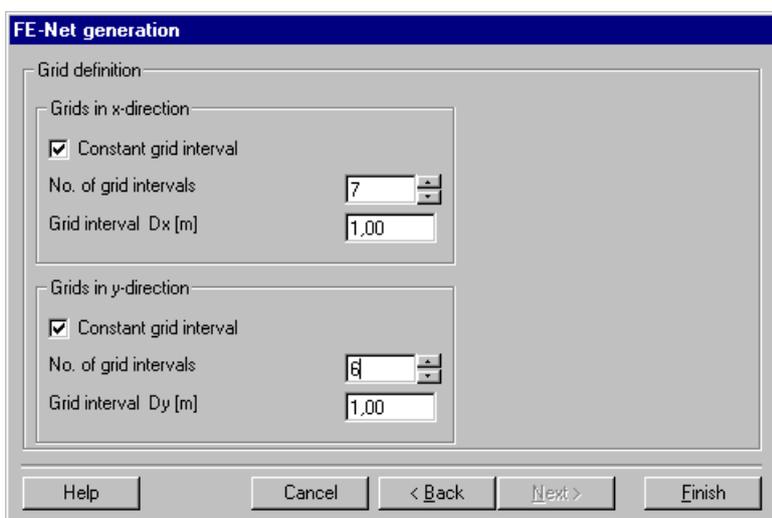


Figure 2-10 "FE-Net generation" dialog box

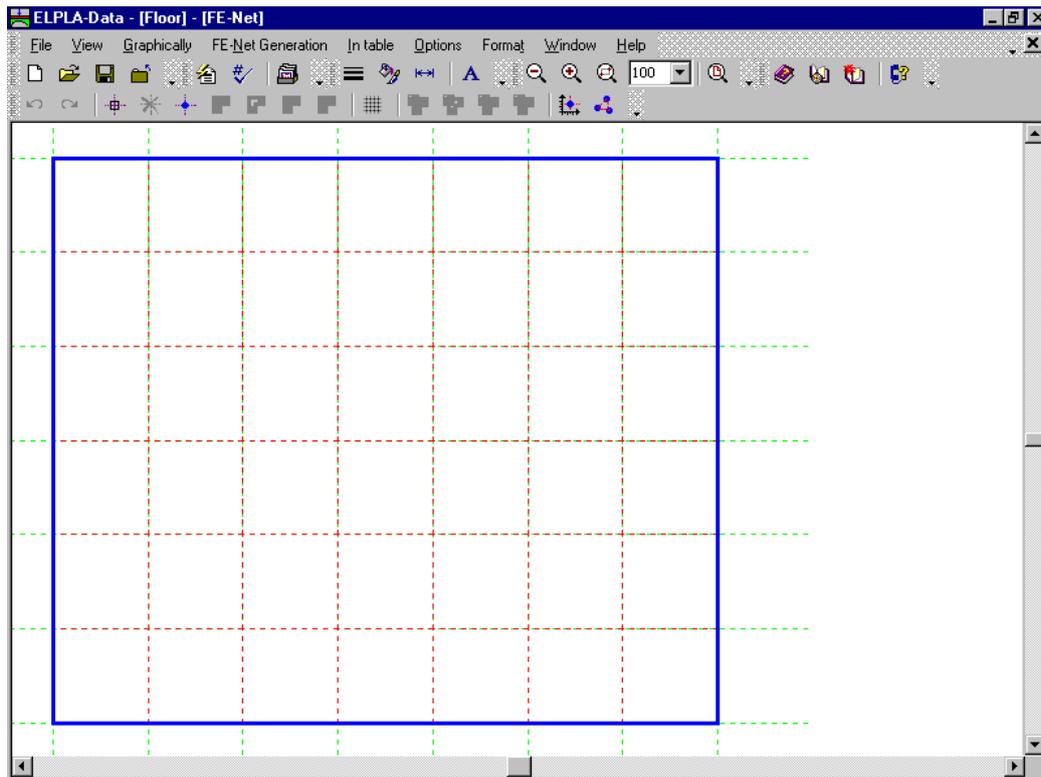


Figure 2-11 FE-Net of a rectangular slab on the screen

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 "Graphically" menu in Figure 2-12. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair. The command "Remove nodes" in the menu "Graphically" 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 2-12. 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.

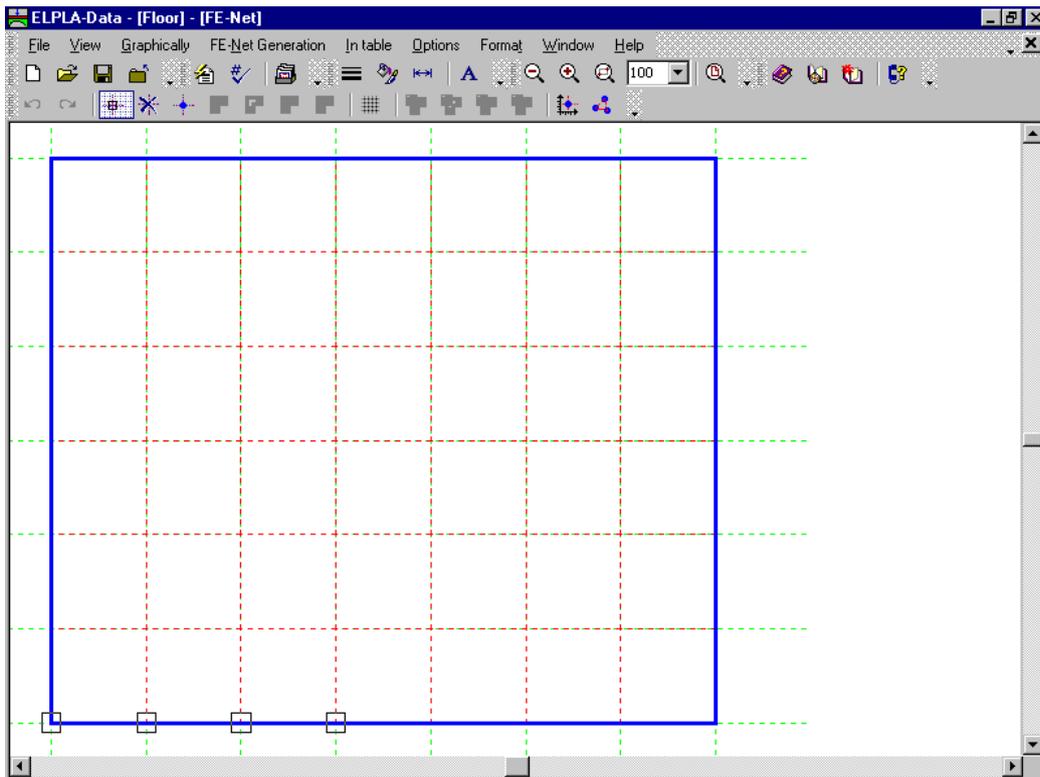


Figure 2-12 Generated FE-Net after selecting the unnecessary nodes

To remove the selected nodes, choose "Remove nodes" command from the "Graphically" menu. The action of this command is indicated in Figure 2-13. To leave the graphic mode, press "Esc" key.

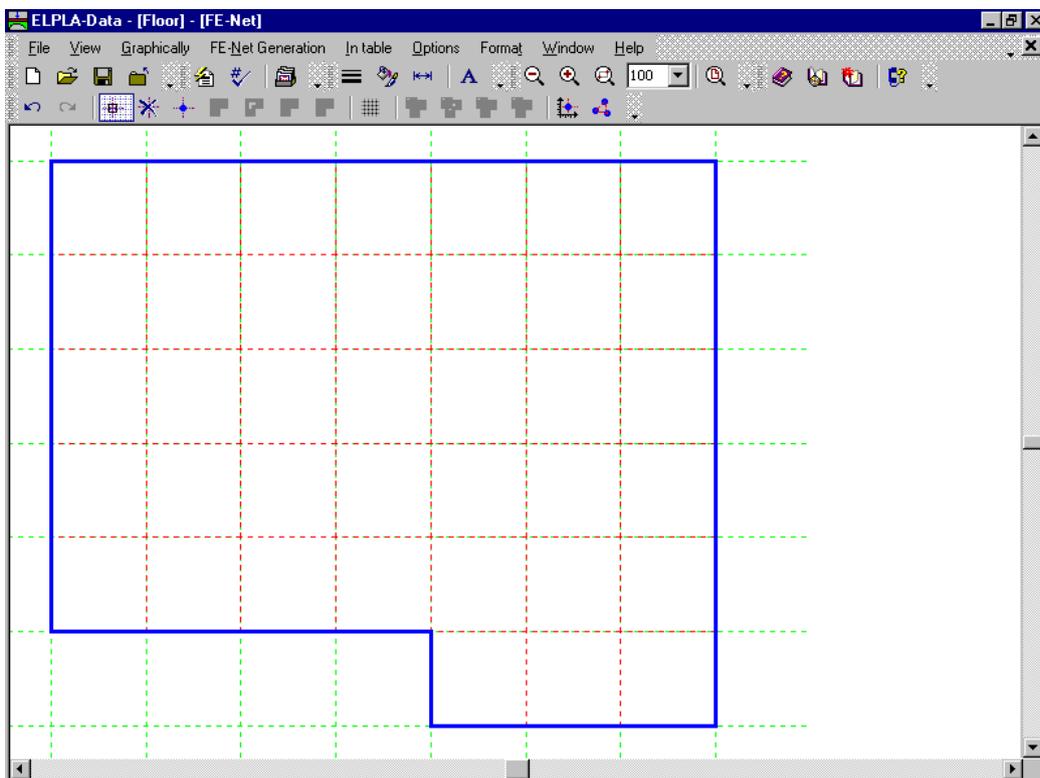


Figure 2-13 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save FE-Net" command from "File" menu in Figure 2-13 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 2-13 to close "FE-Net" embedded program and to return to the main window of *ELPLA-Data*

2.4 Girders

To define the girders choose "Girders" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 2-14 appears.

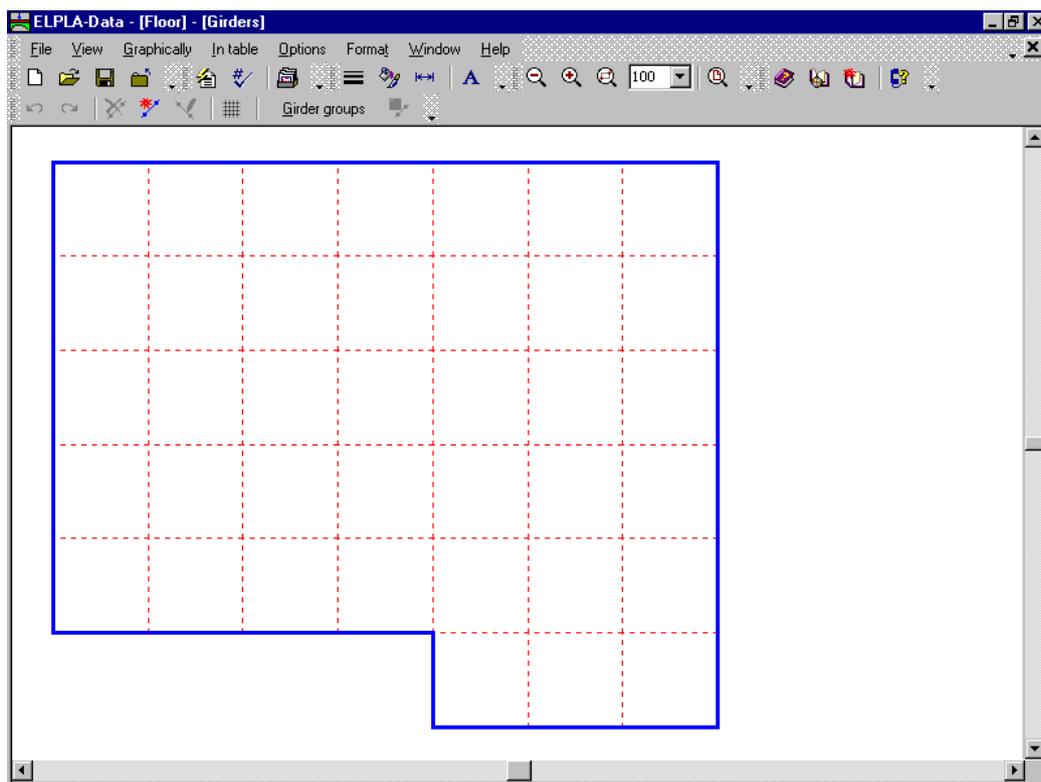


Figure 2-14 "Girders" embedded program

To enter the cross section of the girders

- Choose "Girder groups" command from "In Table" menu in Figure 2-14. The following option box in Figure 2-15 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

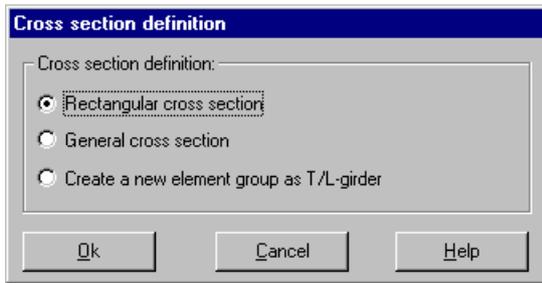


Figure 2-15 "Cross section definition" option box

After clicking "OK" button in the "Cross section definition" option box, the following list box in Figure 2-16 appears.

In this list box

- Enter the material properties of the girder, cross section dimensions and the girder weight as indicated in Figure 2-16. This is done by entering the value in the corresponding cell and press "Enter" button
- Click "OK" button

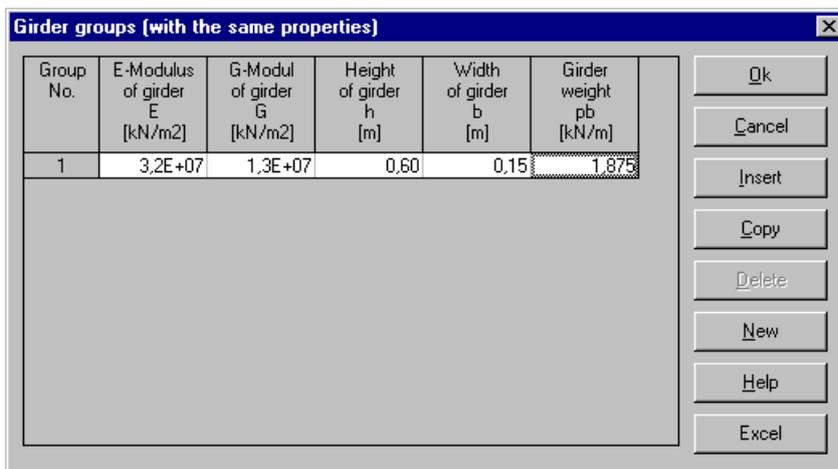


Figure 2-16 "Defining girder groups" list box

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 2-14. 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 2-17) and click on the end node. The "Girder elements" dialog box in Figure 2-18 appears

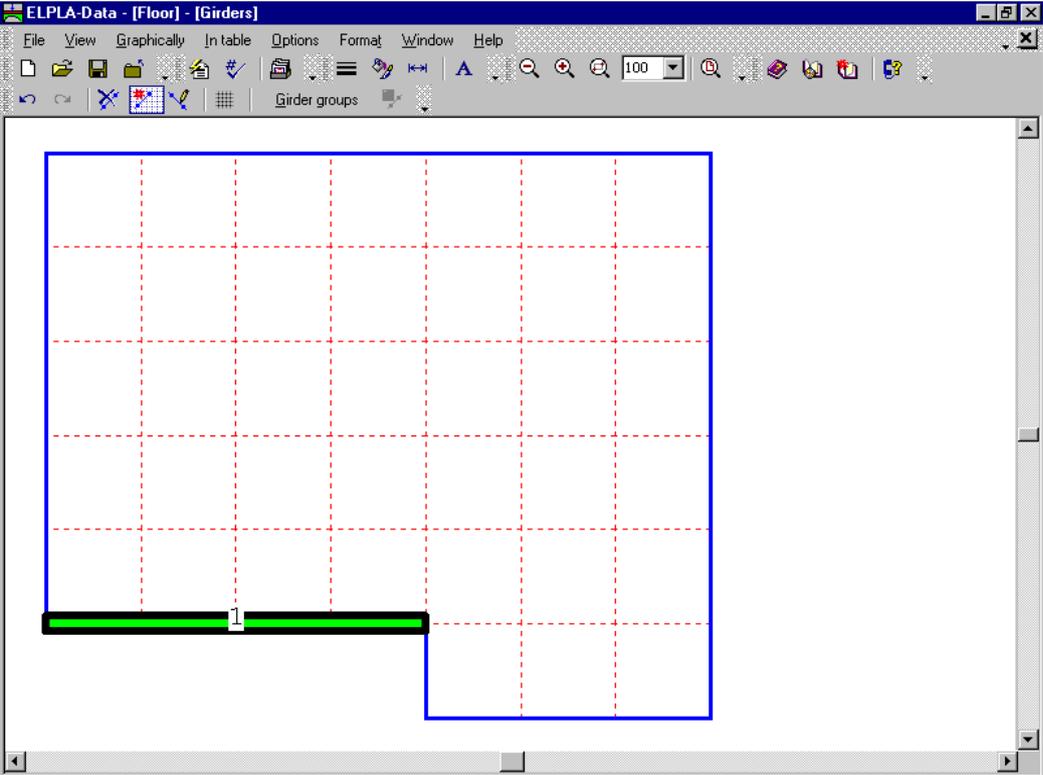


Figure 2-19 First girder on the screen

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 2-20.

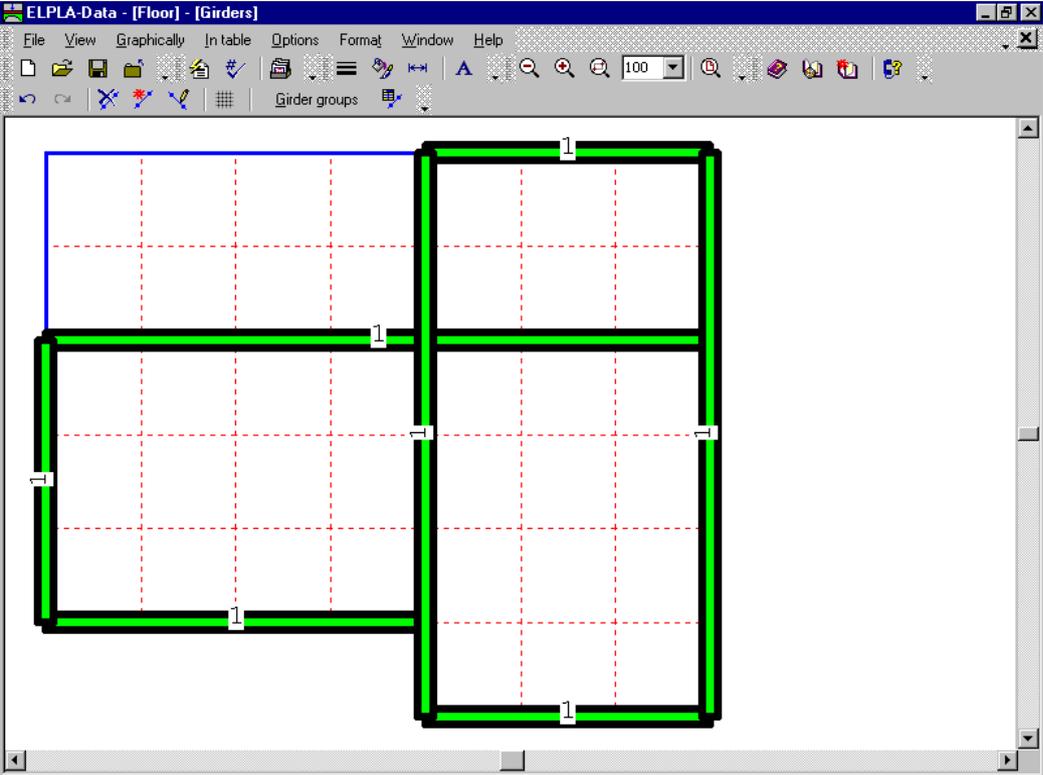


Figure 2-20 Girders on the screen

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

- Choose "Save girders" command from "File" menu in Figure 2-20 to save the data of girders
- Choose "Close girders" command from "File" menu in Figure 2-20 to close the "Girders" embedded program and to return to the main window of *ELPLA-Data*

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" menu of *ELPLA-Data*. The following embedded program in Figure 2-21 appears.

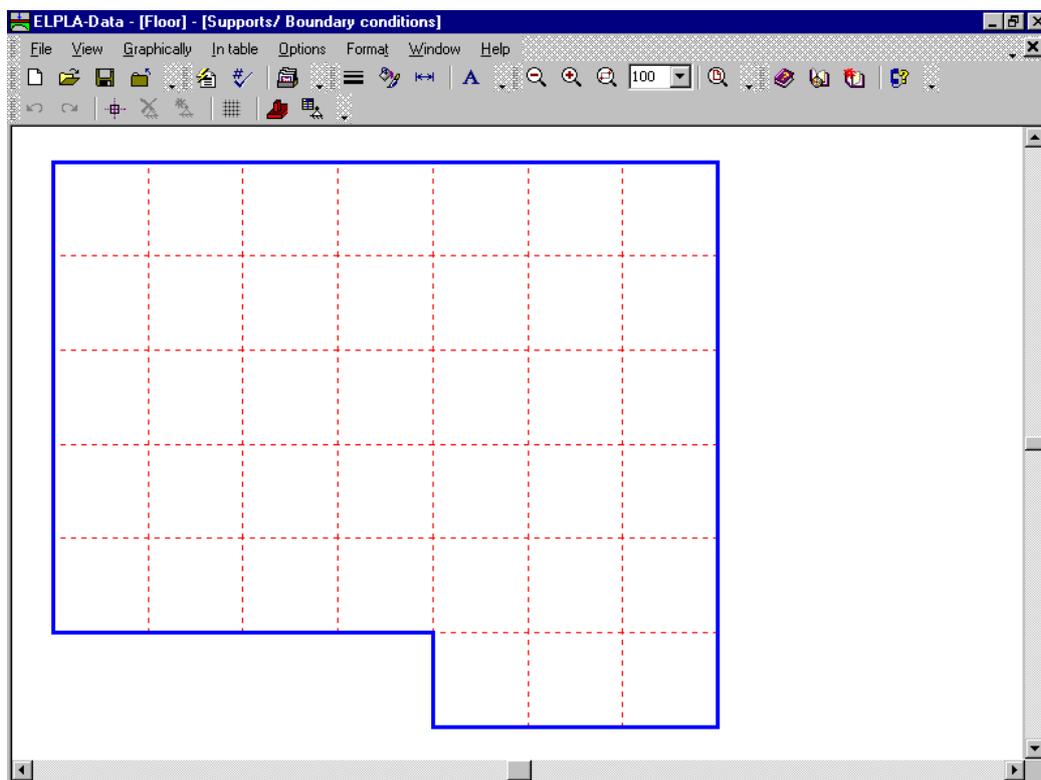


Figure 2-21 "Supports/ Boundary conditions" embedded program

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 from "Option" Menu in Figure 2-21. The "View grouping" check group box in Figure 2-22 appears
- In this check group box, check "Girders" check box
- Click "OK" button

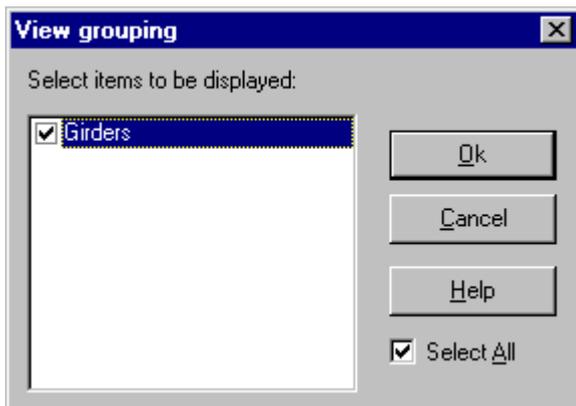


Figure 2-22 "View grouping" check group box

After clicking "OK" in the "View grouping" check group box, the screen should look like the following Figure 2-23.

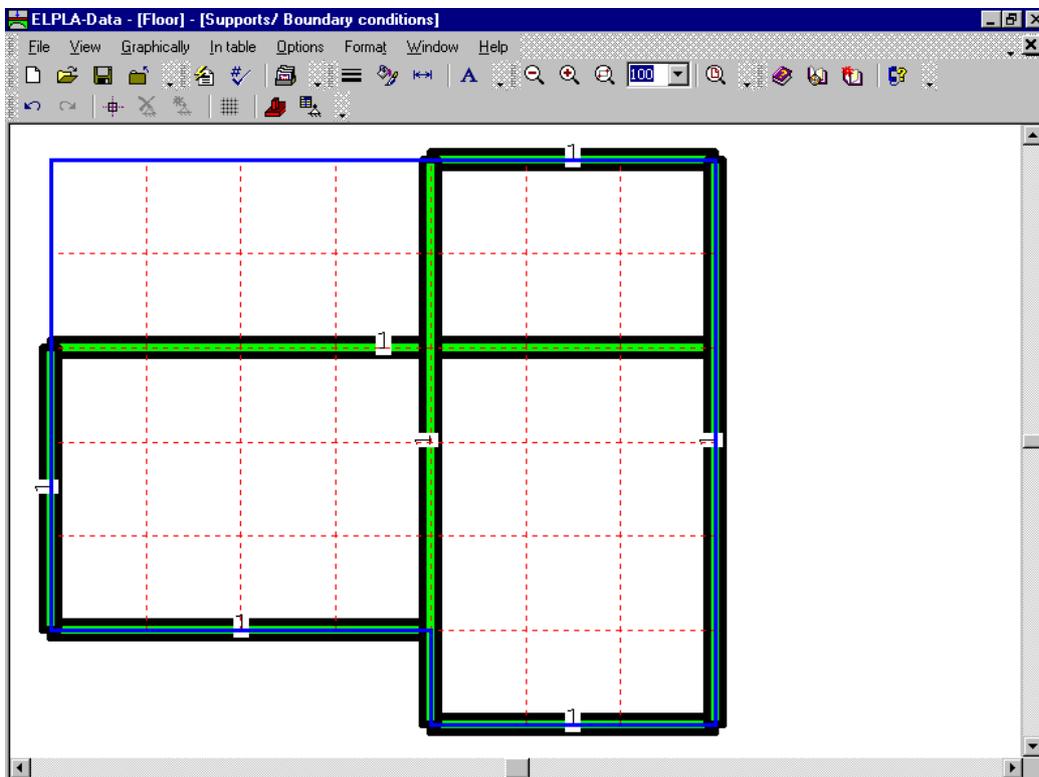


Figure 2-23 Girders in the window of the "Supports/ Boundary conditions" embedded program

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 the "Graphically" menu in Figure 2-23. 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 2-23
- After selecting nodes of supports, choose "Add boundaries" command from "Graphically" menu (Figure 2-23). The "Supports/ Boundary conditions" dialog box in Figure 2-25 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.

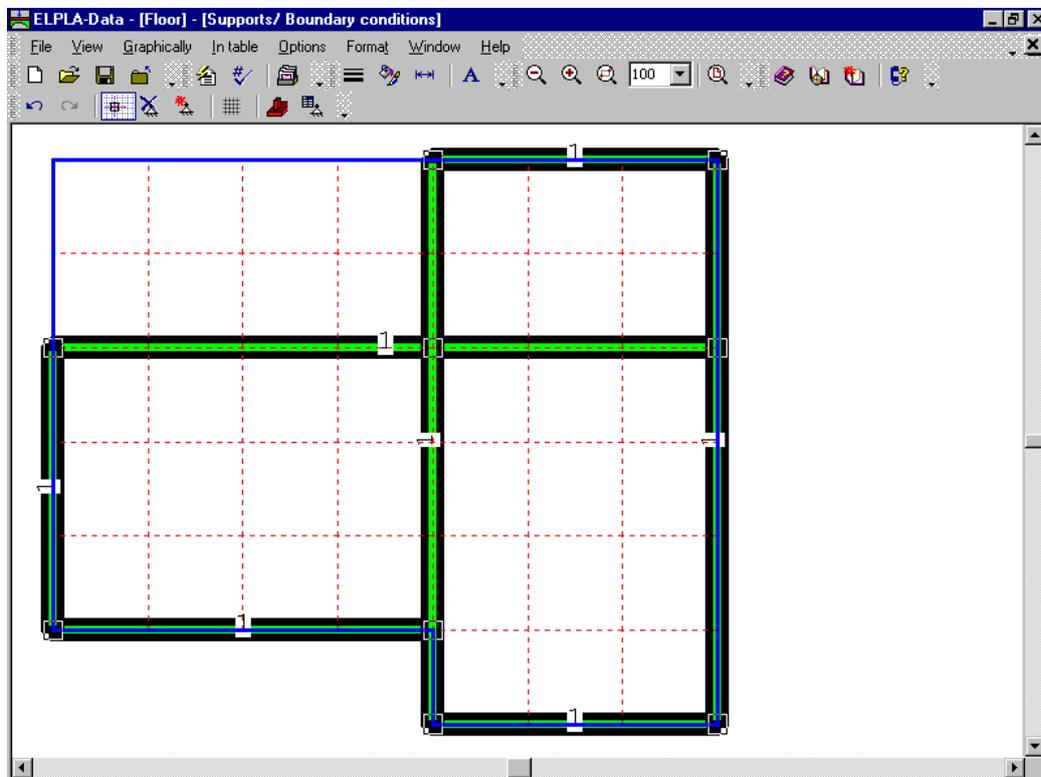


Figure 2-24 Selection of nodes that have supports

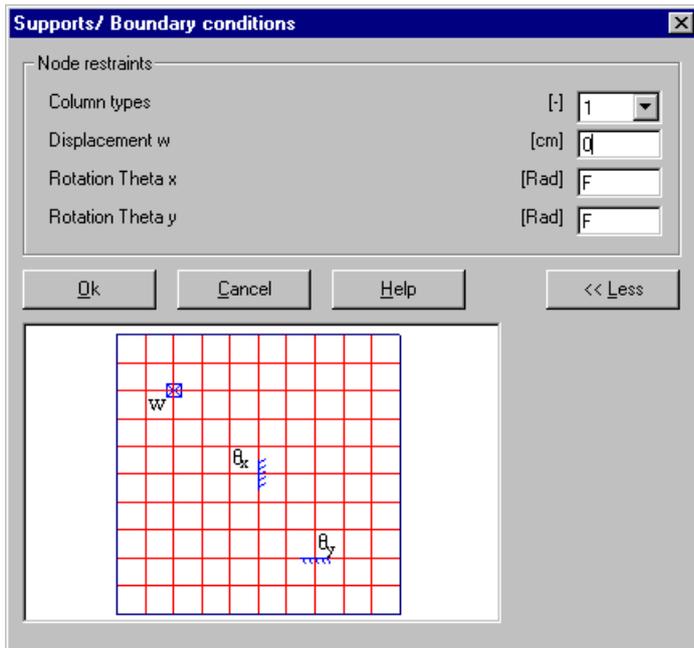


Figure 2-25 "Supports/ Boundary conditions" dialog box

After you have completed the definition of the supports, the screen should look like the following Figure 2-26.

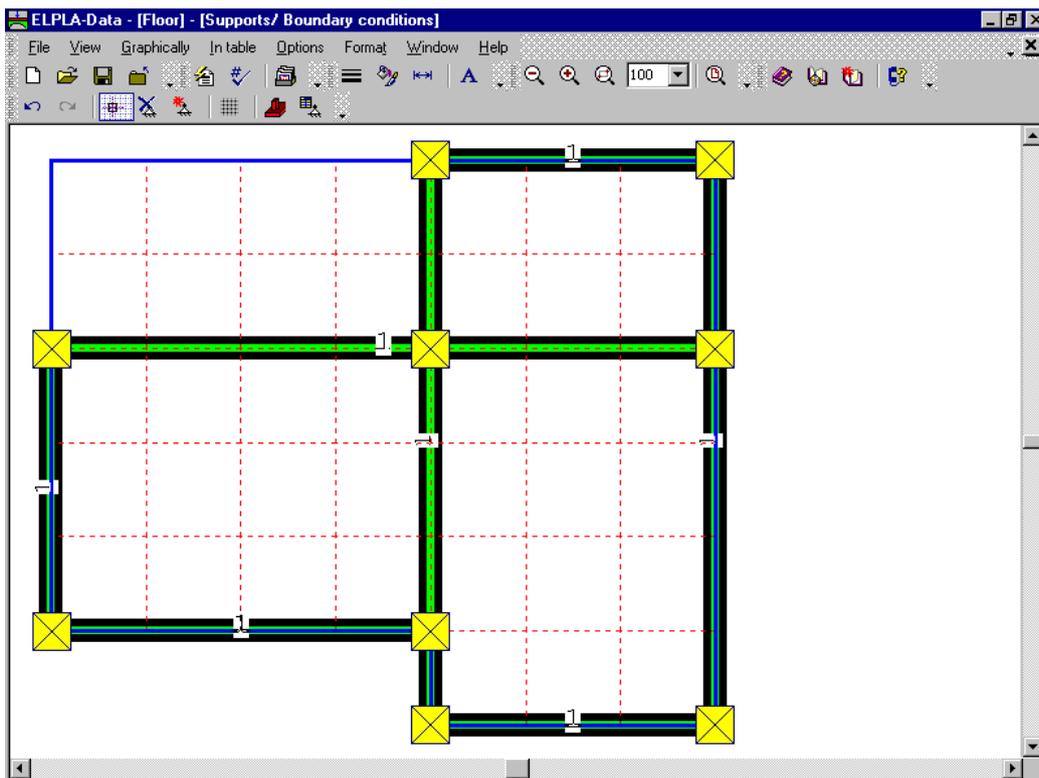


Figure 2-26 Supports on the screen

After entering supports, do the following two steps:

- Choose "Save supports/ Boundary conditions" command from "File" menu in Figure 2-26 to save the data of supports
- Choose "Close supports/ Boundary conditions" command from "File" menu in Figure 2-26 to close the "Supports/ Boundary conditions" embedded program and to return to the main window of *ELPLA-Data*

2.6 Slab properties

To define the slab properties choose "Slab properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 2-27 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.

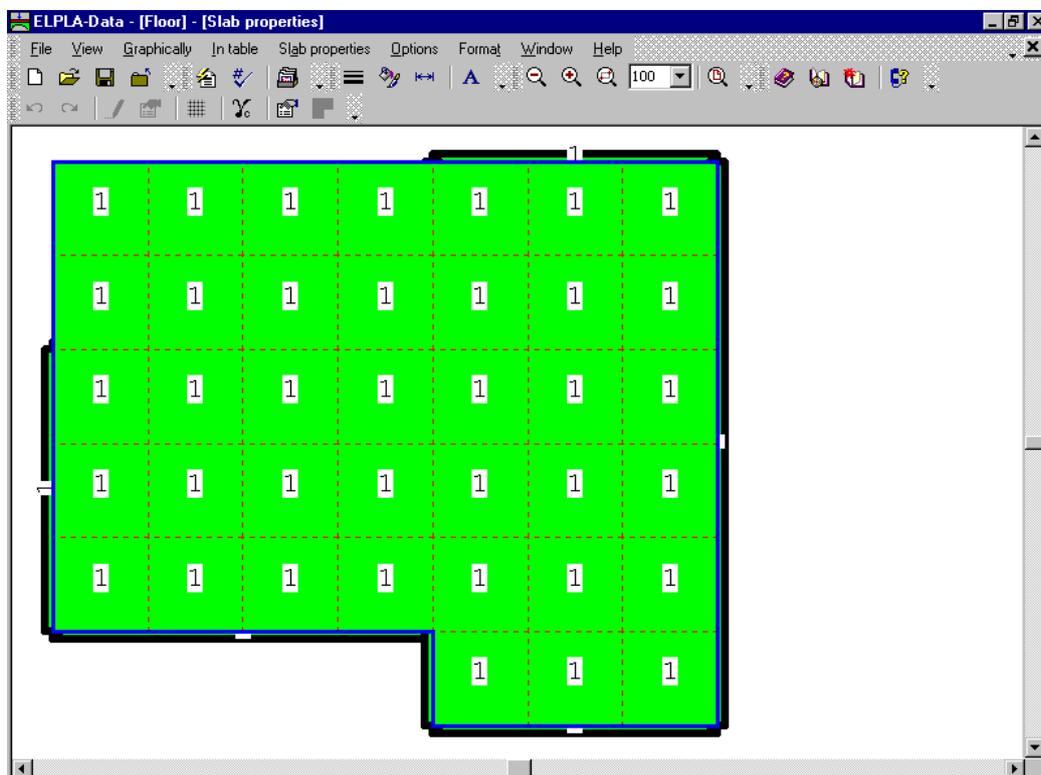


Figure 2-27 "Slab properties" embedded program

To enter the slab material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 2-27. The following list box in Figure 2-28 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 2-28, enter E-Modulus of the slab, *Poisson's* ratio of the slab and slab thickness
- Click "OK" button

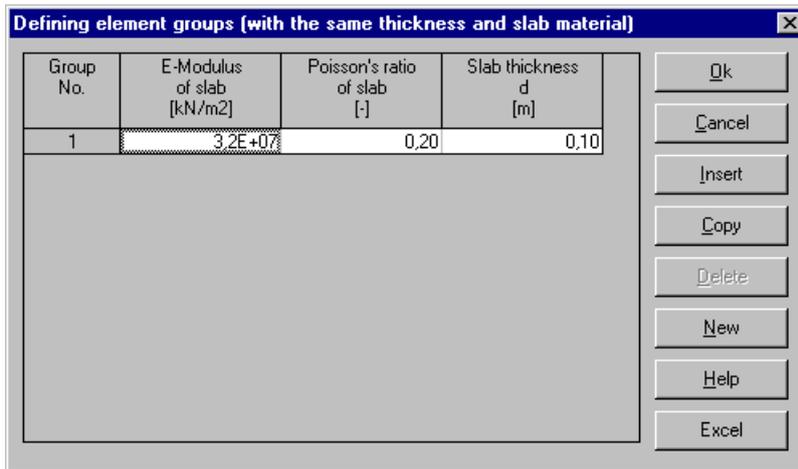


Figure 2-28 "Defining element groups" list box

To enter the unit weight of the slab

- Choose "Unit weight of the slab" command from "Slab properties" menu in the window of Figure 2-27. The following dialog box in Figure 2-29 with a default unit weight of 25 [kN/m³] appears
- Click "OK" button

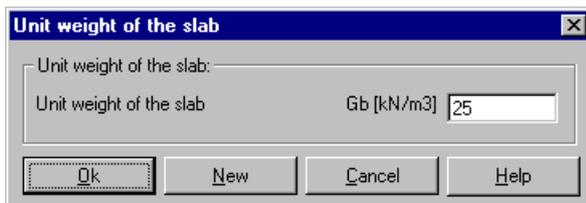


Figure 2-29 "Unit weight of the slab" dialog box

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

- Choose "Save slab properties" command from "File" menu in Figure 2-27 to save the slab properties
- Choose "Close slab properties" command from "File" menu in Figure 2-27 to close the "Slab properties" embedded program and to return to the main window of *ELPLA-Data*

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 2-30):

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]

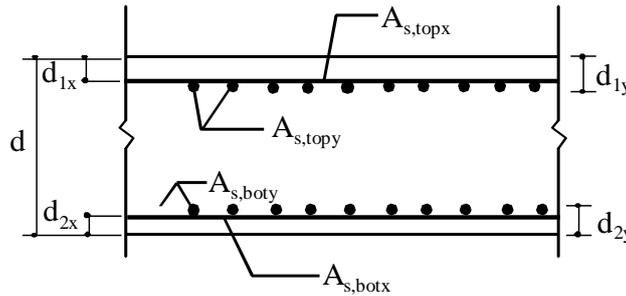


Figure 2-30 Section geometry and reinforcement parallel to x -direction

It is important to say that 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 "Main data" menu in *ELPLA-Data*, while reinforcement data such as design code, concrete grade, steel grade and concrete covers are defined by choosing the "Reinforcement" command from "Data" menu in *ELPLA-Data*. 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" menu of *ELPLA-Data*. The dialog box in Figure 2-31 appears with default reinforcement data.

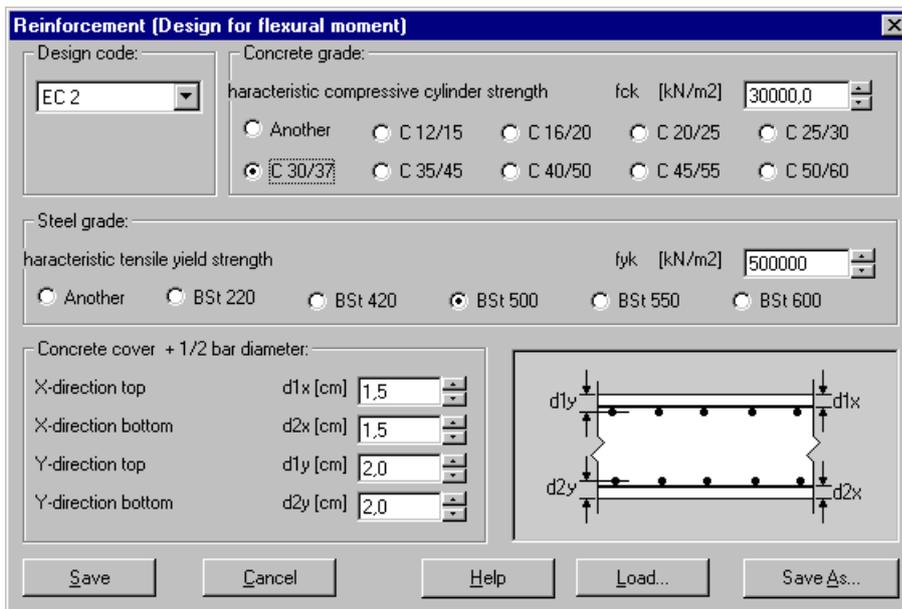


Figure 2-31 "Reinforcement" dialog box

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 indicated in the "Concrete cover" dialog group box
- Click "Save" button

2.8 Loads

To define the loads choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 2-32 appears with girders on the net.

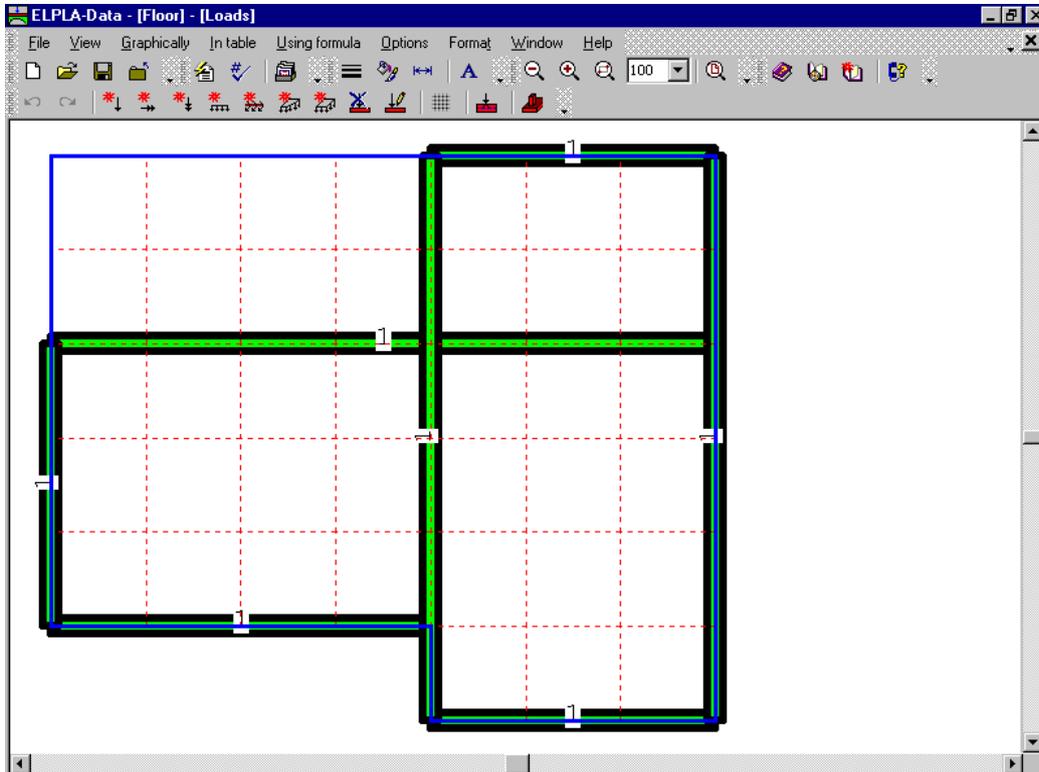


Figure 2-32 "Loads" embedded program

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" command from "Graphically" menu in the window of Figure 2-32. When "Distributed loads" 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 2-33 appears with the load value and coordinates

In this dialog box

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

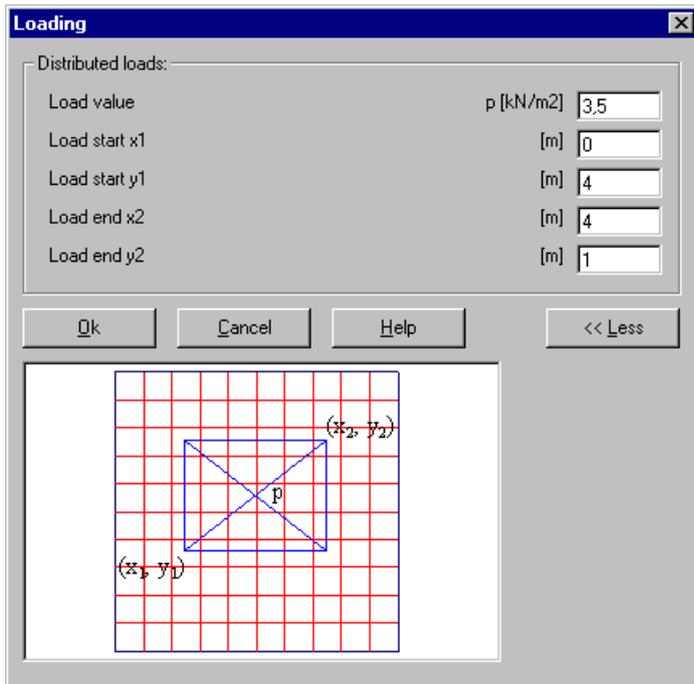


Figure 2-33 "Loading" dialog box

After you have completed the definition of the first distributed load, the screen should look like the following Figure 2-34.

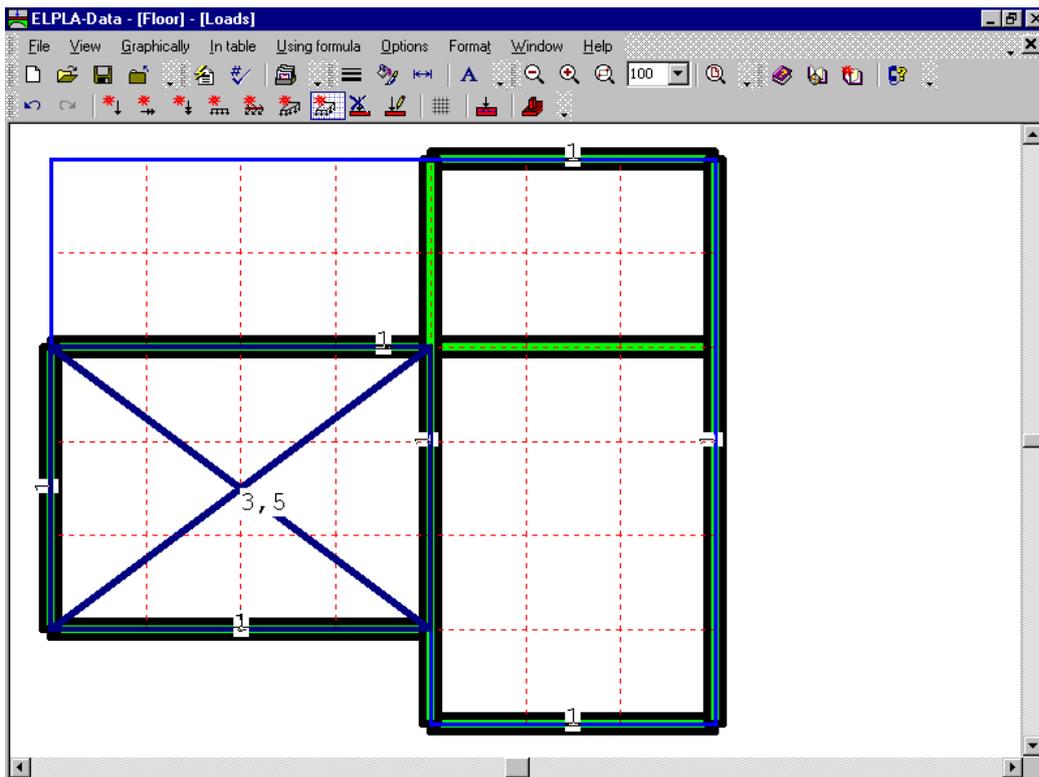


Figure 2-34 First distributed load on the screen

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 2-35.

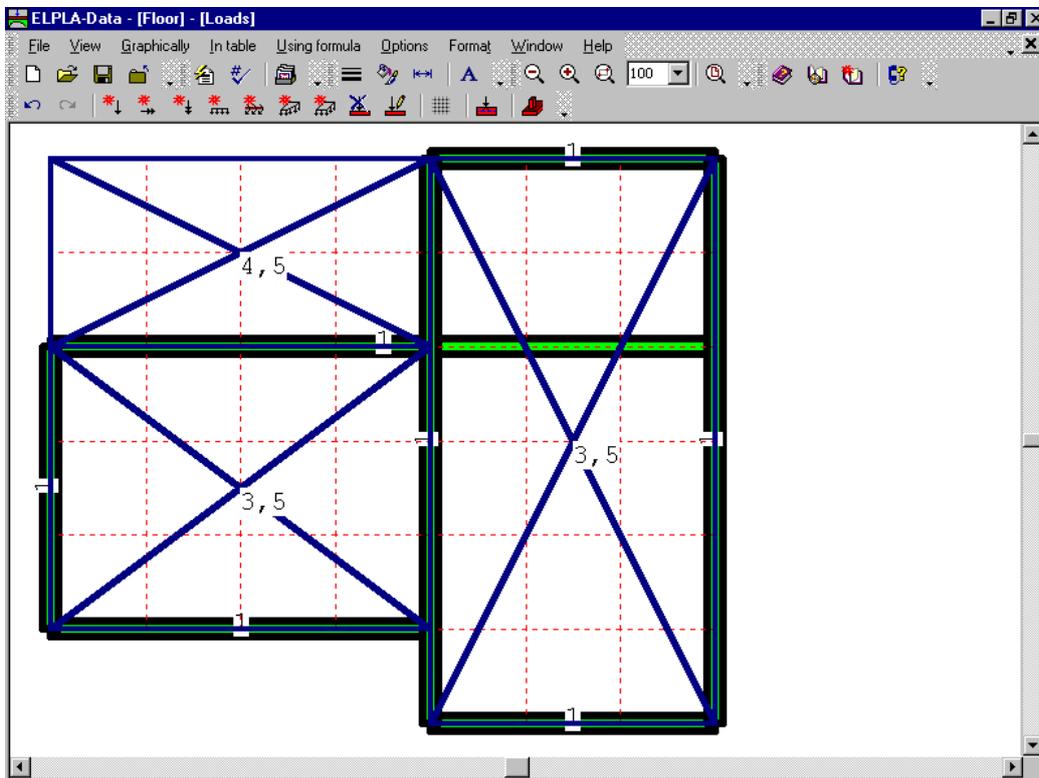


Figure 2-35 Loads on the FE-Net

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

- Choose "Save loads" command from "File" menu in Figure 2-35 to save the load data
- Choose "Close loads" command from "File" menu in Figure 2-35 to close the "Loads" embedded program and to return to the main window of *ELPLA-Data*

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

3.1 Starting *ELPLA-Solver*

To analyze the problem, switch to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window in Figure 2-36 appears.

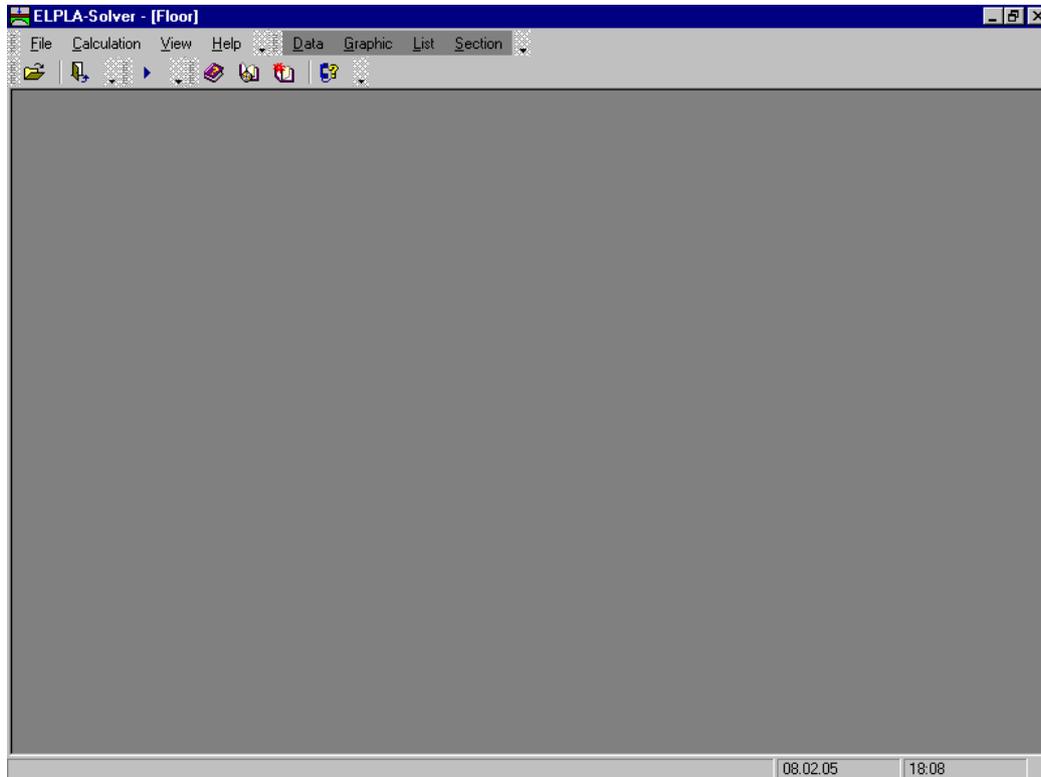


Figure 2-36 Opening screen of the sub program *ELPLA-Solver*

ELPLA-Solver automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. Also, *ELPLA* will activate the "Calculation" menu. This menu 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

These calculation items can be carried out individually or in one time.

3.2 Carrying out all computations

To carry out all computations in one time choose "Computation of all" command from "Calculation" menu in *ELPLA-Solver* window. The 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 2-37 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

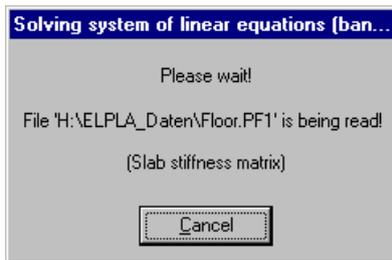


Figure 2-37 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution in Figure 2-38 appears. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

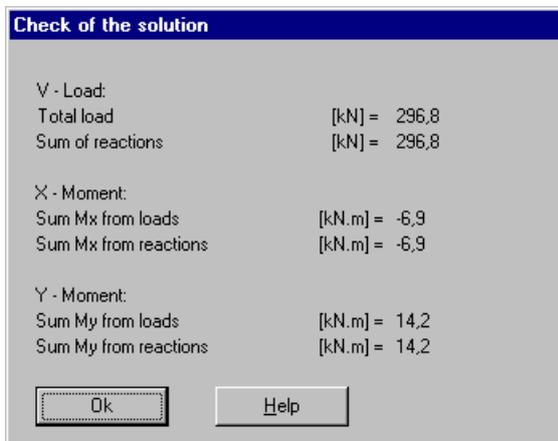


Figure 2-38 Menu "Check of the solution"

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

4 Viewing data and result graphics

ELPLA can view the data and results for the slab and girders either separately or together. Individual data or results for the slab can be viewed in a manner similar to that in the previous example. Here, the user will learn to view the results of the girders through the following sample.

To view the data and results of a problem that has already been defined and analyzed graphically, switch to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window.

ELPLA-Graphic window in Figure 2-39 appears. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

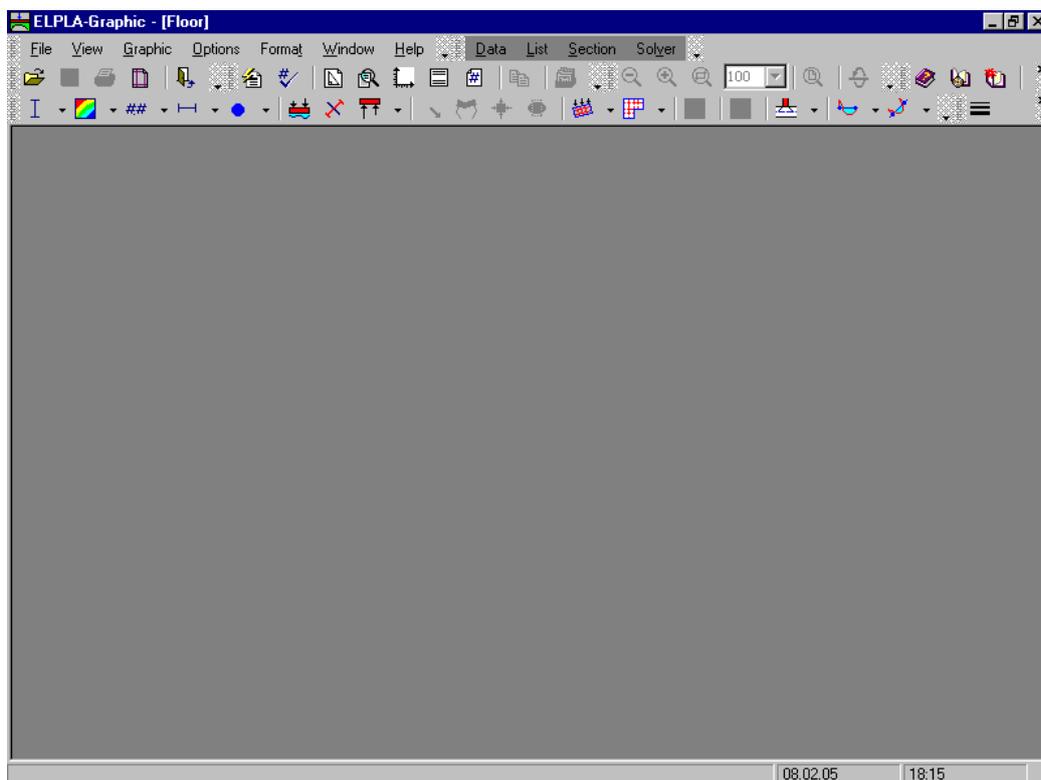


Figure 2-39 Opening screen of the sub program *ELPLA-Graphic*

To view the results of girders choose "Girders" command and then the "Distribution of internal forces in the plan" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 2-40 appears.

In this option box

- Select "Beam-Bending moments M_b " as a sample for the results to be displayed
- Click "OK" button

The moments are now displayed for the girders as shown in Figure 2-41.

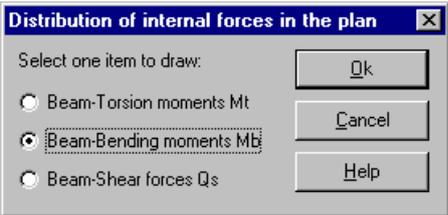


Figure 2-41 "Distribution of internal forces in the plan" option box

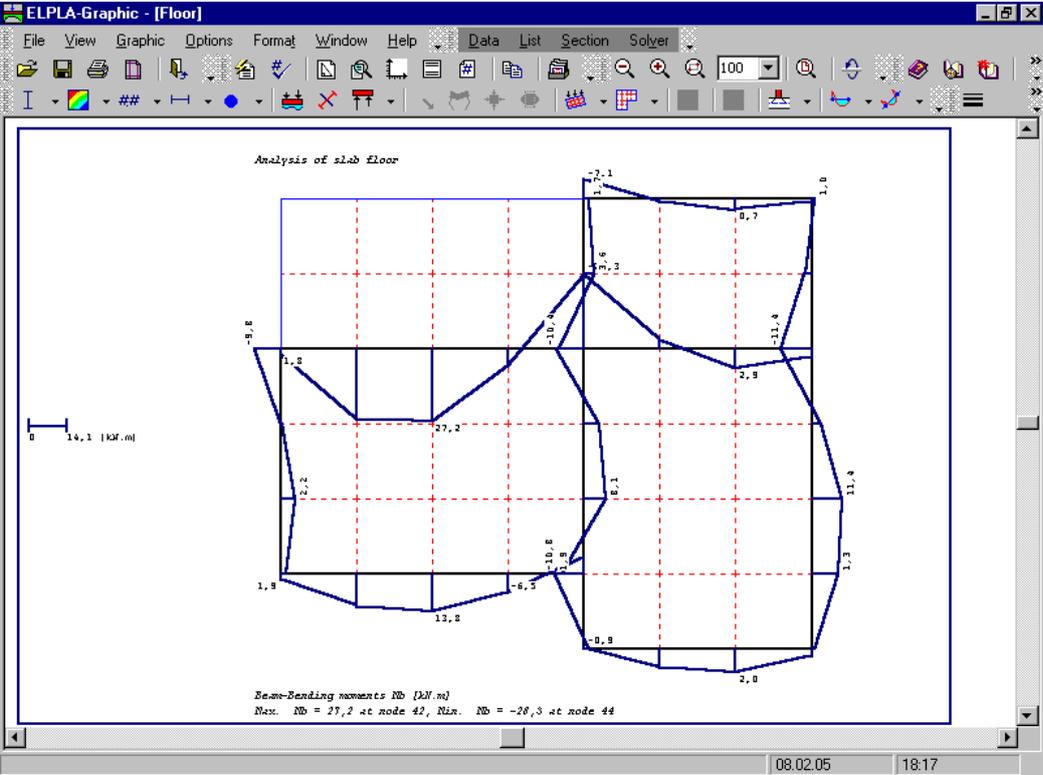


Figure 2-41 Beam-Bending moments M_b

5 Index

A

Analysis type 4, 5

B

Boundary conditions 16, 18

C

Calculation method 4
 Check of the solution 27
 Computation of all..... 27
 Concrete design..... 4

D

Deformation 26
 Design codes 21, 22

E

Element groups..... 21

F

FE-Net..... 8, 9

G

Generation type 8
 Girder locations..... 13
 Girders..... 12, 15

I

Internal forces 22, 26

L

Load vector 26
 Loads 3, 23

M

Material properties 13

N

New project..... 4

P

Project identification..... 7

R

Reinforcement..... 21
 Remove nodes..... 10, 11

S

Select nodes 10
 Slab properties 20
 Supports 5, 16, 18, 20
 System symmetry 5

U

Unit weight of the slab..... 21