

Example 4

Analysis of a grid

Contents		Page
1	Description of the problem	3
	1.1 Loads and dimensions	3
	1.2 Grid material	3
	1.3 Analysis of the grid	3
2	Creating the project	4
	2.1 Calculation method	4
	2.2 Project identification	6
	2.3 FE-Net data	7
	2.4 Girders	9
	2.5 Supports/ Boundary conditions	13
	2.6 Loads	18
3	Carrying out the calculations	20
	3.1 Starting <i>ELPLA-Solver</i>	20
	3.2 Carrying out all computations	21
4	Viewing data and result graphics	22
5	Index	24

1 Description of the problem

An example of a grid resting on 8 supports is selected to illustrate some features of *ELPLA* for analyzing grids.

1.1 Loads and dimensions

The grid consists of equal girders that have dimensions of 0.15 [cm] * 0.60 [cm]. Geometry of the grid and loads on it including the own weight are shown in Figure 4-1.

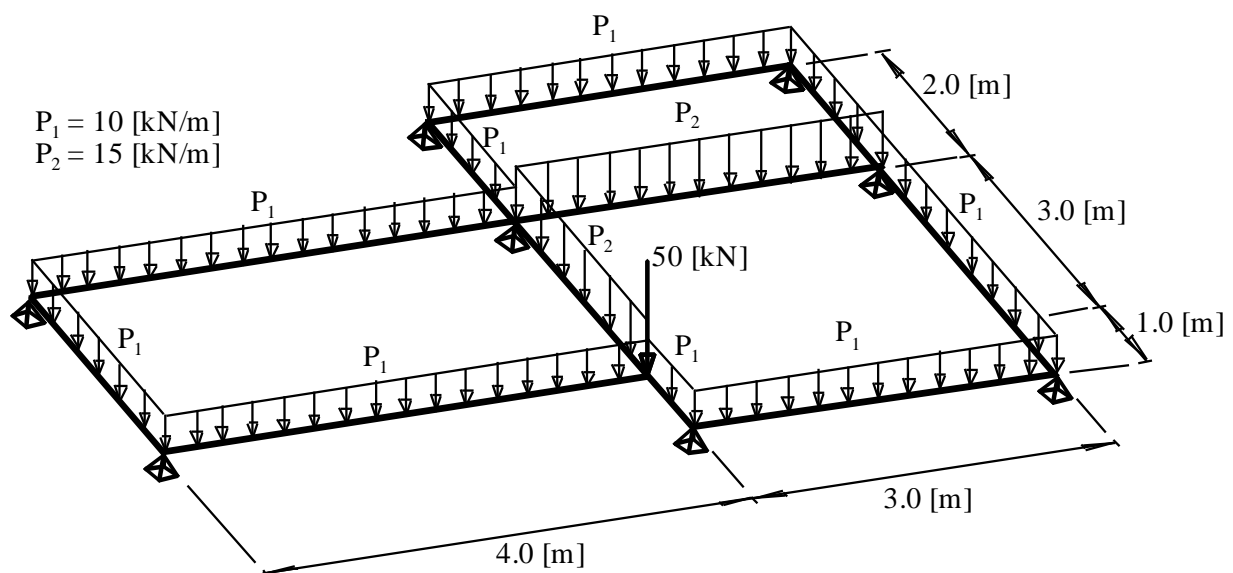


Figure 4-1 Geometry of the grid and loads

1.2 Grid material

Material of the grid has the following parameters:

Young's modulus	E_b	$= 3.2 * 10^7$	[kN/m ²]
Shear modulus	G_b	$= 1.3 * 10^7$	[kN/m ²]

1.3 Analysis of the grid

It is required to analyze the grid using 1.0 [m] beam element in both x - and y -directions and considering the supports to be rigid in z -direction only. 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 are well documented in the User's Guide of *ELPLA*.

2 Creating the project

In this section the user will learn how to create a project for analyzing a grid. The example will be processed step by step to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose "New project" command from the "File" menu. The "Calculation methods" wizard appears, Figure 4-2. The wizard will guide you through the steps required to create the system data of the project. As shown in Figure 4-2, the first form of the wizard is the "Analysis type" form.

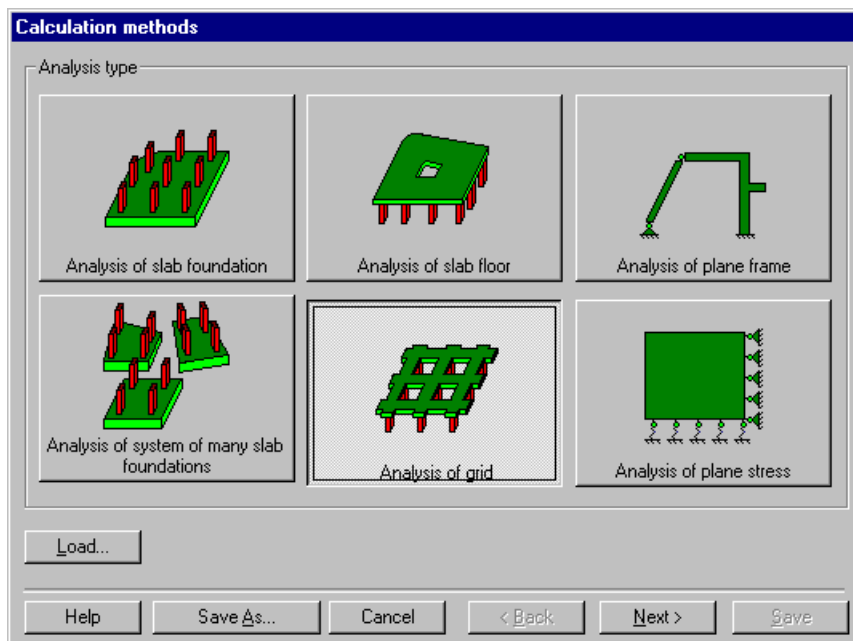


Figure 4-2 "Calculation method" wizard with "Analysis type" form

In Figure 4-2 define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a grid problem, select "Analysis of grid". Then click "Next" button to go to the next form.

The next form is the "System symmetry" (Figure 4-3). In this form

- Choose "Unsymmetrical system"
- Click "Next" button

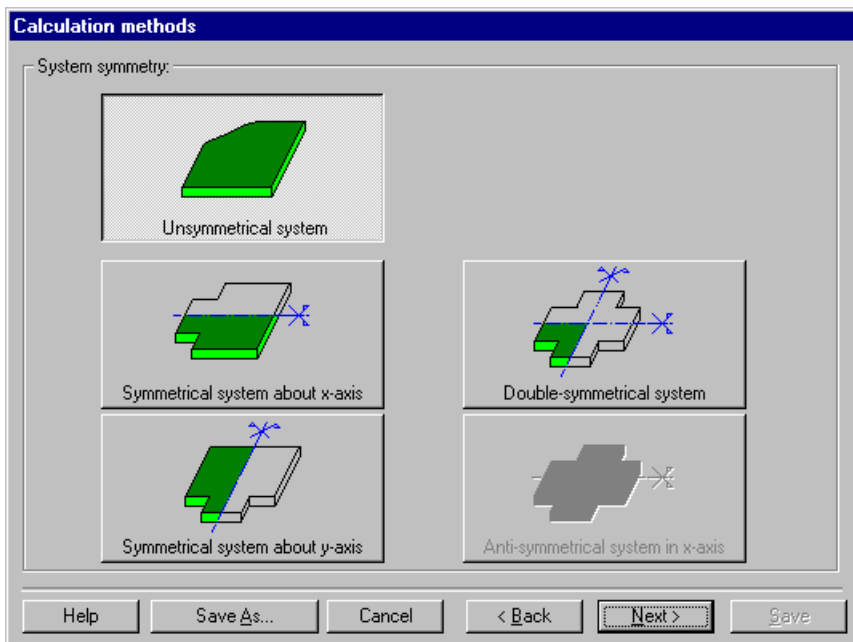


Figure 4-3 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 4-4. 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
- Click "Save" button

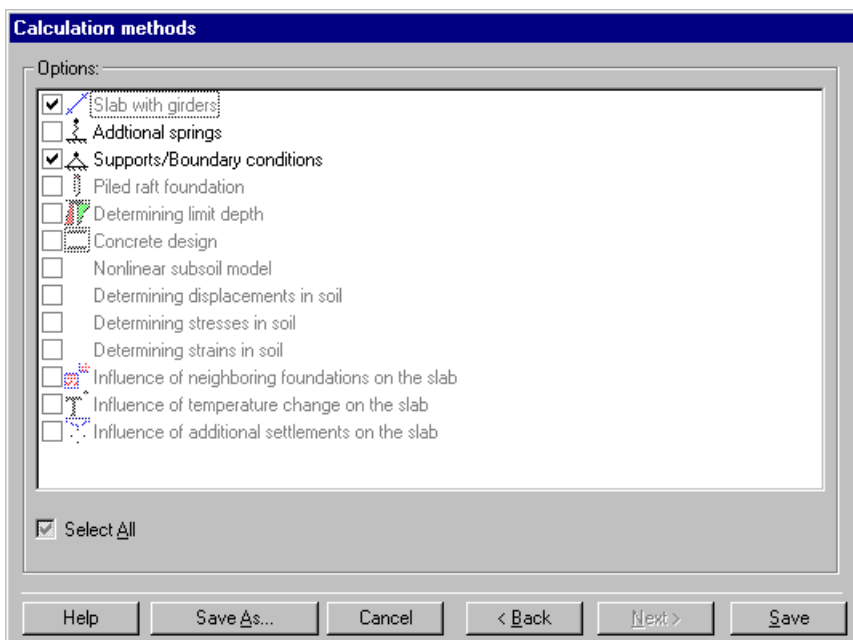


Figure 4-4 "Options" list

After clicking "Save" button the "Save as" dialog box appears (Figure 4-5).

In this dialog box

- Type a file name for the current project in the file name edit box. For example type "Grid". *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" menu. Also the file name of the current project [Grid] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar.

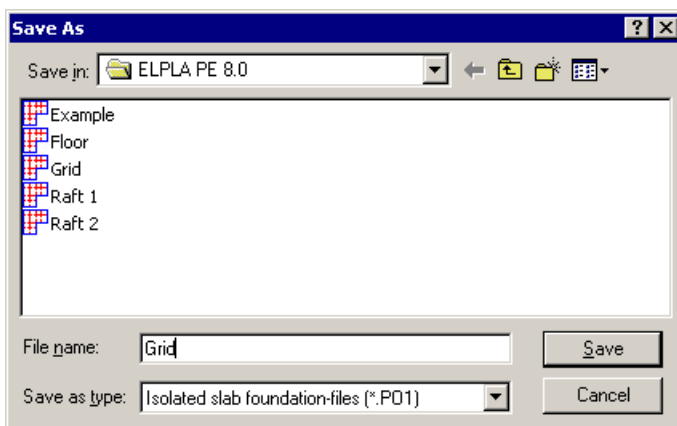


Figure 4-5 "Save as" dialog box

2.2 Project identification

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data* window. The dialog box in Figure 4-6 appears.

In this dialog box

- Type the following line to describe the problem in the "Title edit" box:
"Analysis of a grid"
- Type the date of the project in the "Date" edit box
- Type "Grid" in the "Project edit" box
- Click "Save" button

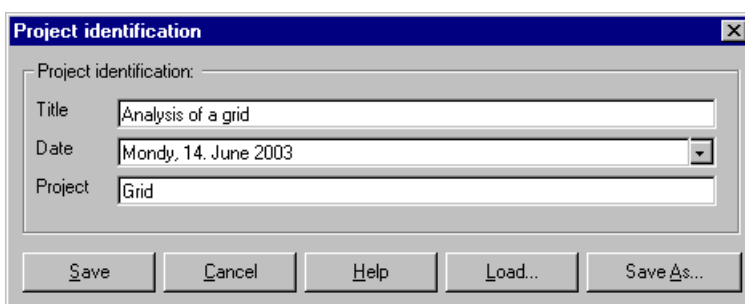


Figure 4-6 "Project identification" dialog box

2.3 FE-Net data

To enter the grid geometry, an imaginary net must be first defined with suitable dimensions and then the grid is entered later by connecting the corresponding nodes on the net that defines the grid shape. To define the net for the grid geometry, choose "FE-Net data" command from "Data" menu. The "FE-Net generation" wizard appears as shown in Figure 4-7. 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 constant size in both x - and y -directions.

To generate an imaginary net

- Choose the rectangular slab option to create an imaginary net of a rectangular area
- 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

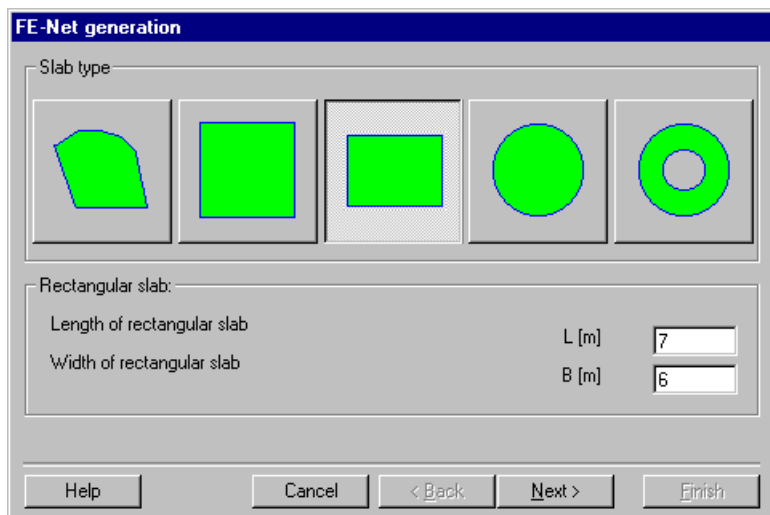


Figure 4-7 "FE-Net generation" wizard with "Slab type" form

After clicking "Next" in the "FE-Net generation" wizard the following "Generation type" form appears, Figure 4-8. *ELPLA* can deal with various types of generations with triangle and/ or rectangular elements.

In this form

- Choose rectangular elements
- Click "Next"

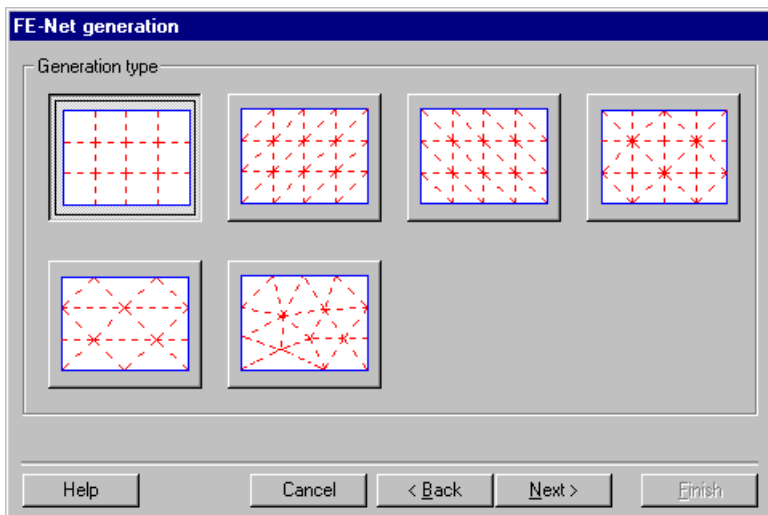


Figure 4-8 "Generation type" form

After clicking "Next" button in "Generation type" form the following "Grid definition" dialog box in Figure 4-9 appears with default values of constant element size.

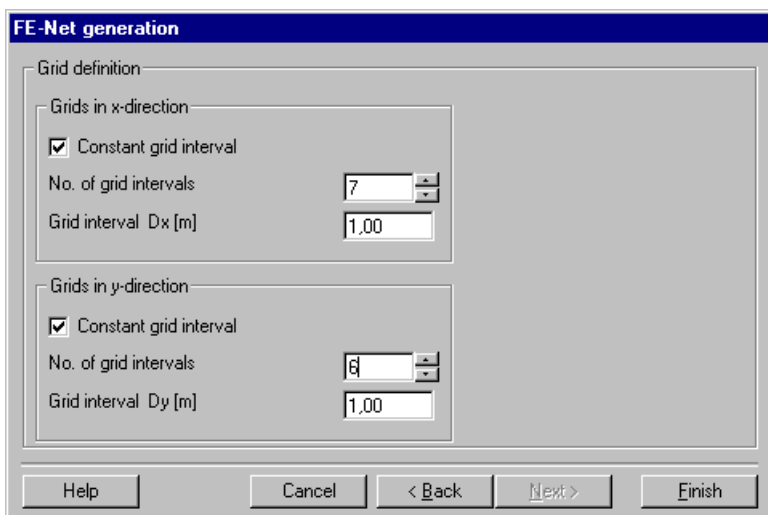


Figure 4-9 "FE-Net generation" dialog box

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 an imaginary net for a rectangular area of 7 [m] length and 6 [m] width with square elements of 1.0 [m] each side. The following embedded program in Figure 4-10 appears with the generated imaginary net.

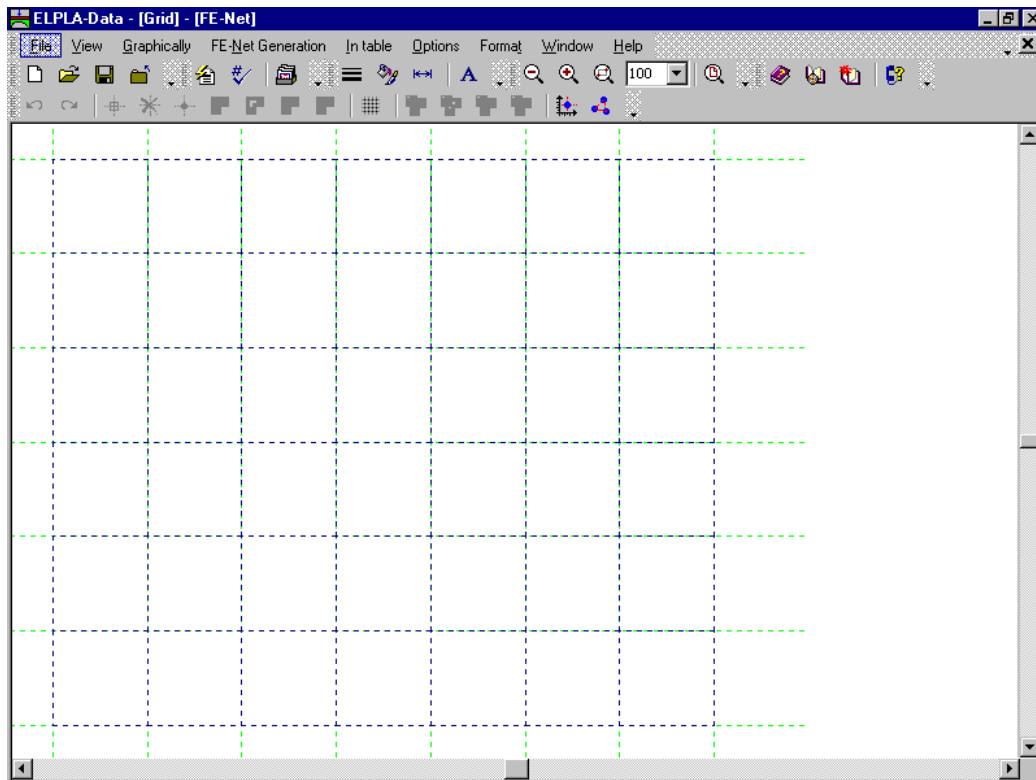


Figure 4-10 Imaginary net of a rectangular area on the screen

After finishing generation of the imaginary net do the following two steps:

- Choose "Save FE-Net" command from "File" menu to save the data of the imaginary net
- Choose "Close FE-Net" command from "File" menu 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* window. The following embedded program in Figure 4-11 appears.

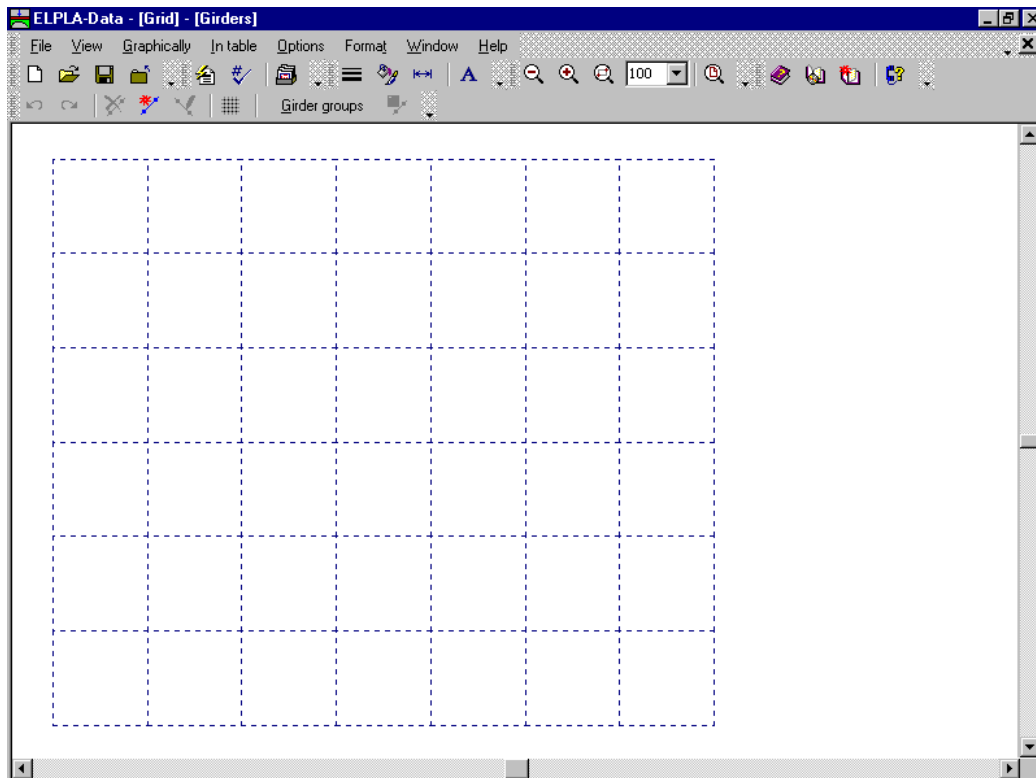


Figure 4-11 "Girders" embedded program

To enter the cross section of the girders

- Choose "Girder groups" command from "In Table" menu in Figure 4-11. The following option box in Figure 4-12 appears
- In this option box select "Rectangular cross section"
- Click "OK" button

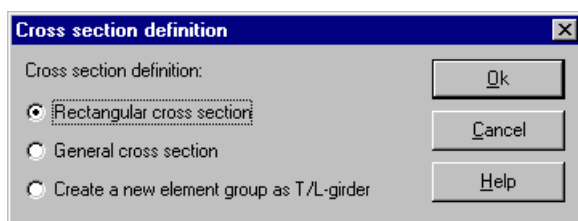


Figure 4-12 "Cross section definition" option box

After clicking "OK" button in the "Cross section definition" option box the following list box in Figure 4-13 appears. In this list box

- Enter the material properties of the girder, cross section dimensions and the girder weight as indicated in Figure 4-13. This is done by entering the value in the corresponding cell and press "Enter" key. For simplicity, the line loads, which include the own weight of the girder, are entered in the cell "Girder weight". Here two groups of girders are defined, the first group for girders that have a line load of 10 [kN/m] while the second group for girders that have a line load of 15 [kN/m]
- Click "OK" button

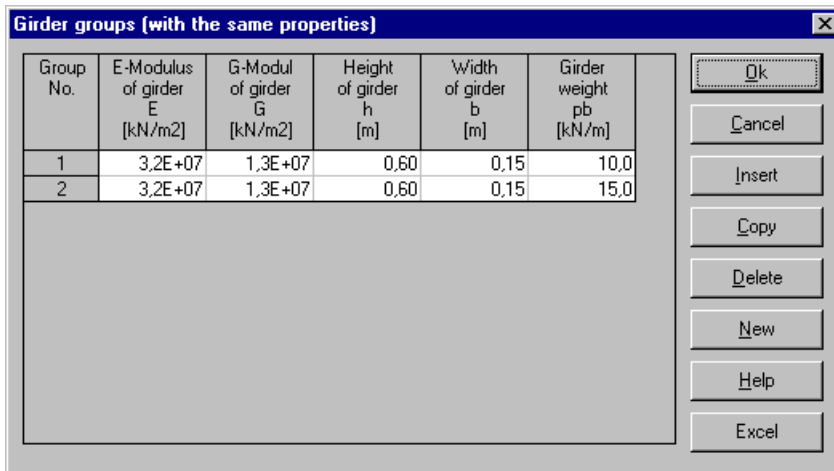


Figure 4-13 "Defining girder groups" list box

Defining the girder locations on the net

This 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 4-11. 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 and drag the mouse until the end node of that girder (Figure 4-14). Then click on the end node. The "Girder elements" dialog box in Figure 4-15 appears

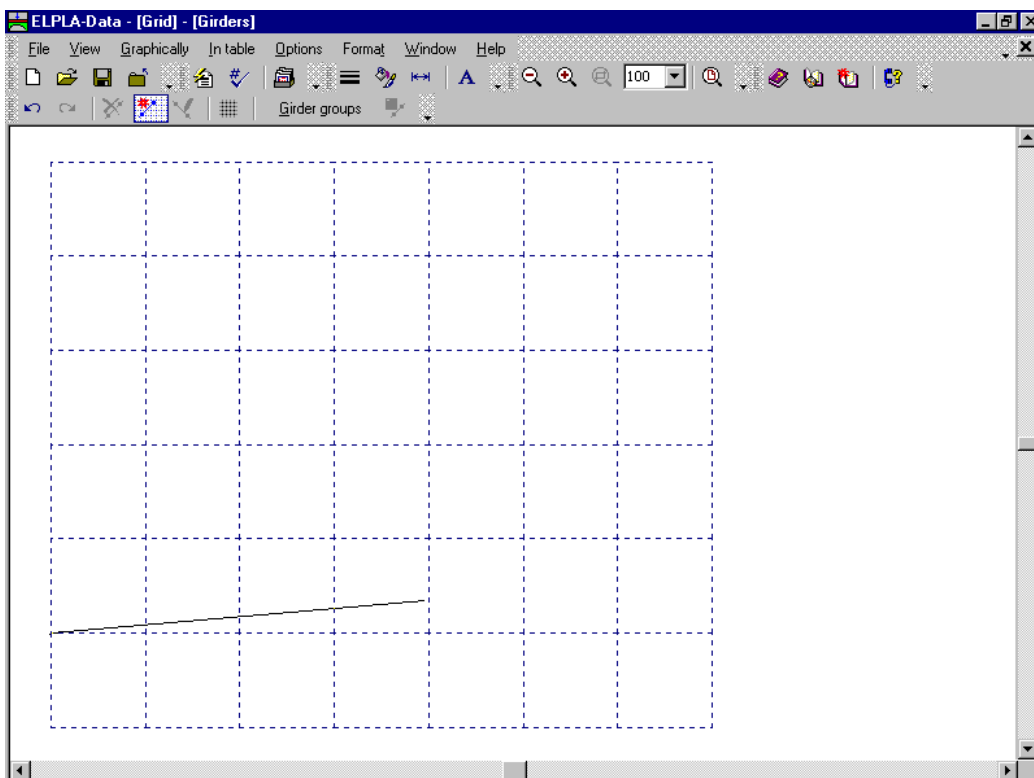


Figure 4-14 Add girder by mouse

In this dialog box

- Select the group No
- Click "OK" button

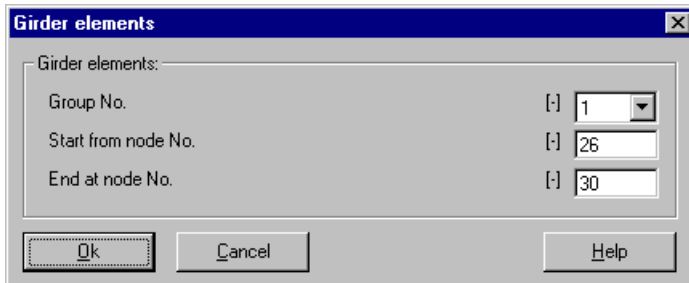


Figure 4-15 "Girder elements" dialog box

Now the first girder is defined as shown in Figure 4-16. Note that *ELPLA* has typed automatically the girder type on it indicating the No. of girder group.

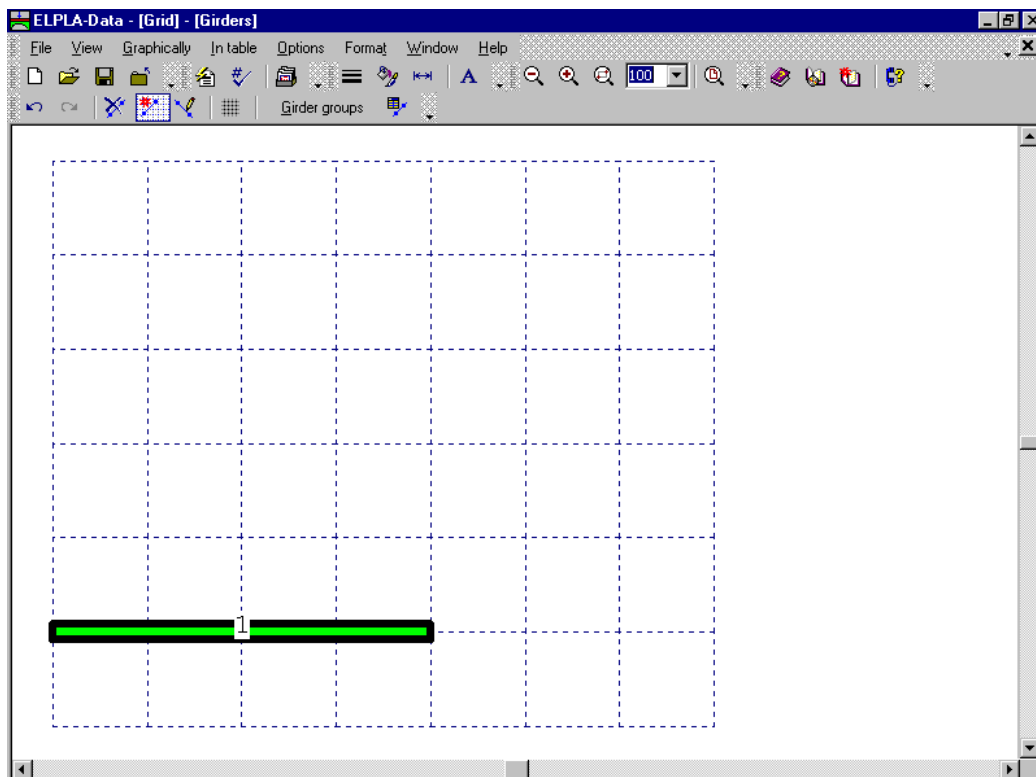


Figure 4-16 First girder on the screen

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 4-17.

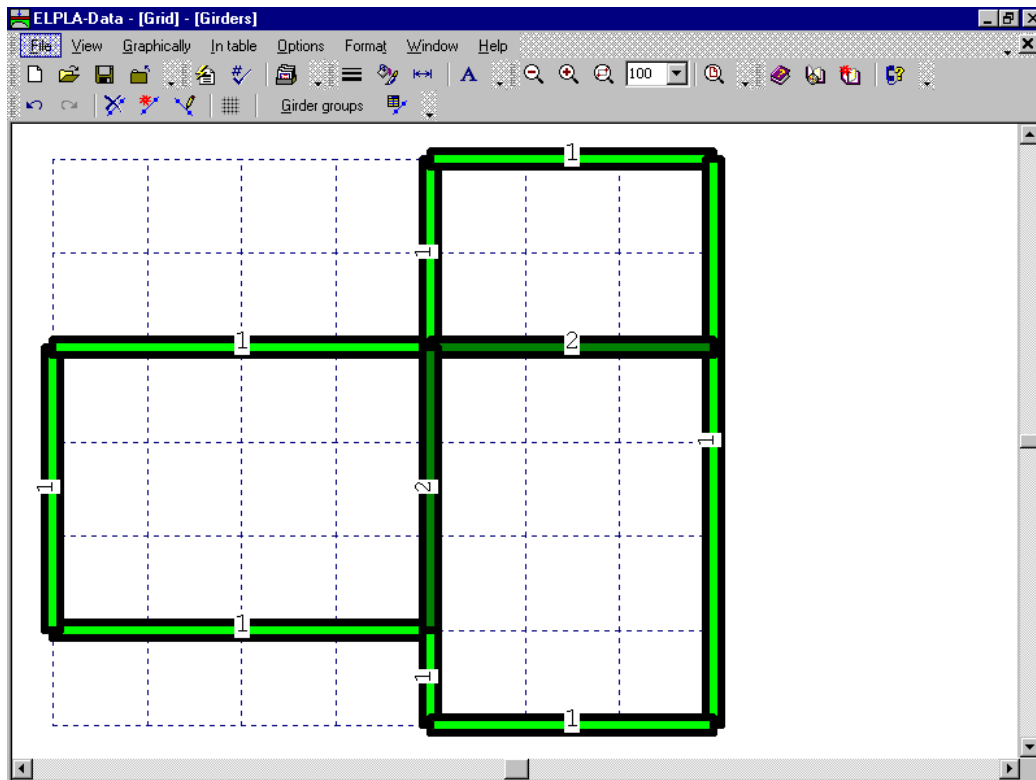


Figure 4-17 Girders on the screen

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

- Choose "Save girders" command from "File" menu in Figure 4-17 to save the data of girders
- Choose "Close girders" command from "File" menu in Figure 4-17 to close the "Girders" embedded program and to return to the main window of *ELPLA-Data*

2.5 Supports/ Boundary conditions

To define supports, choose "Supports/ Boundary conditions" command from "Data" menu of *ELPLA-Data* window. The following embedded program in Figure 4-18 appears.

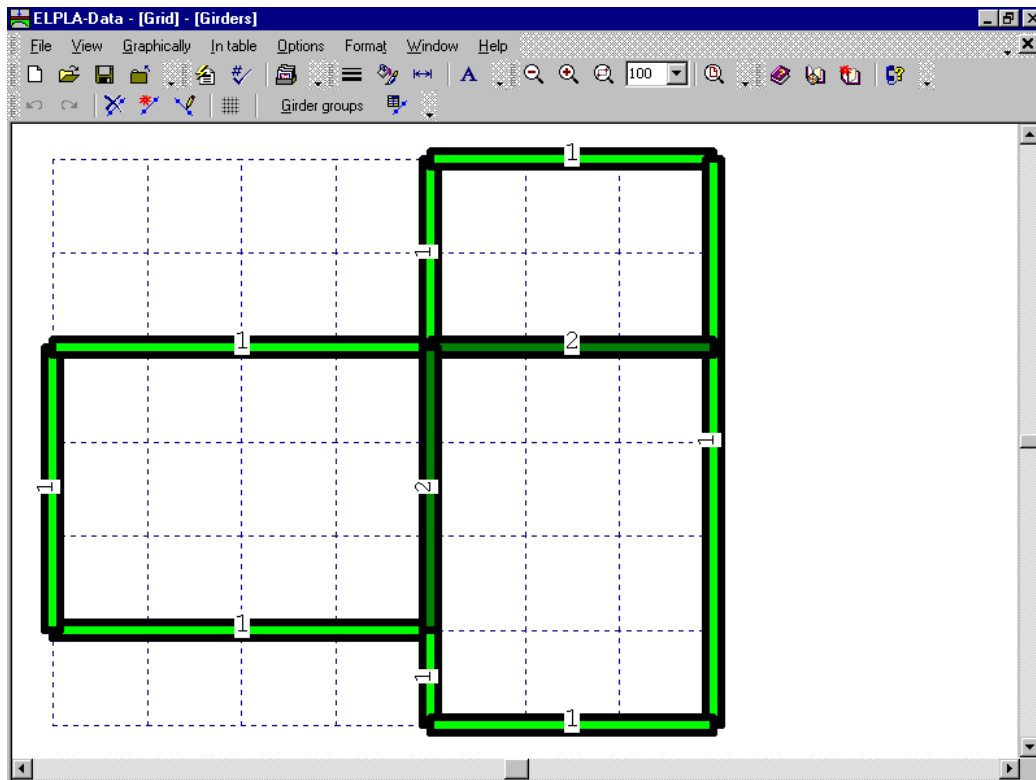


Figure 4-18 "Supports/ Boundary conditions" embedded program

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. In the case of analysis of girder problems, *ELPLA* displays girders during input of other data. As shown in Figure 4-18, girders are drawn with the actual thickness.

To view girders as simple lines

- Choose "Plot parameters" command from "Options" menu.
The "Plot parameters" dialog box in Figure 4-19 appears
- In the "General plot parameters" tab uncheck the "Draw thickness" check box in the "Girder systems" dialog box
- Click "OK" button

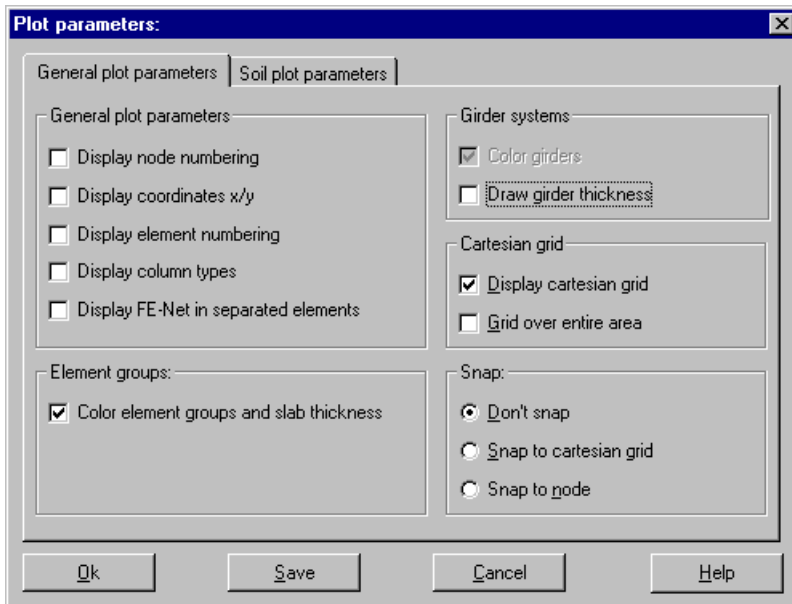


Figure 4-19 "Plot parameters" dialog box

After clicking "OK" in the "Plot parameters" dialog box the girders are drawn as simple lines as shown in Figure 4-20.

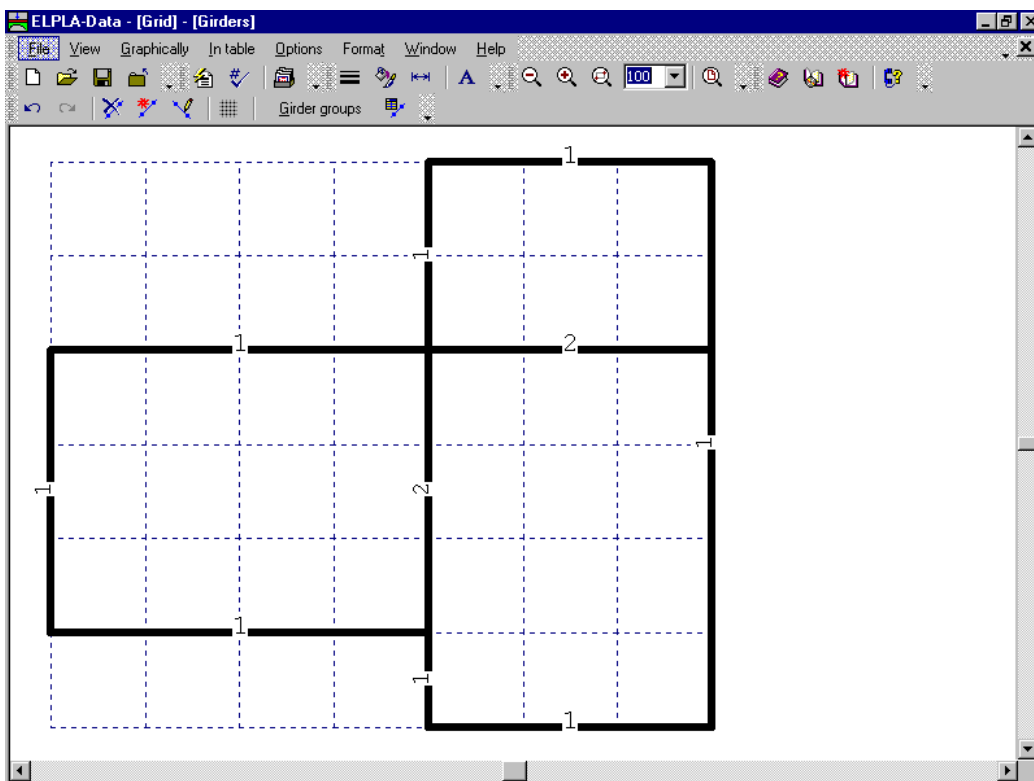


Figure 4-20 Girders are drawn in simple lines

Defining supports on the net

Defining supports or boundary conditions on the net may be carried out either graphically or numerically (in 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 4-20. 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 4-21
- After selecting nodes of supports choose "Add boundaries" command from "Graphically" menu in Figure 4-21. "Supports/ Boundary conditions" dialog box in Figure 4-22 appears

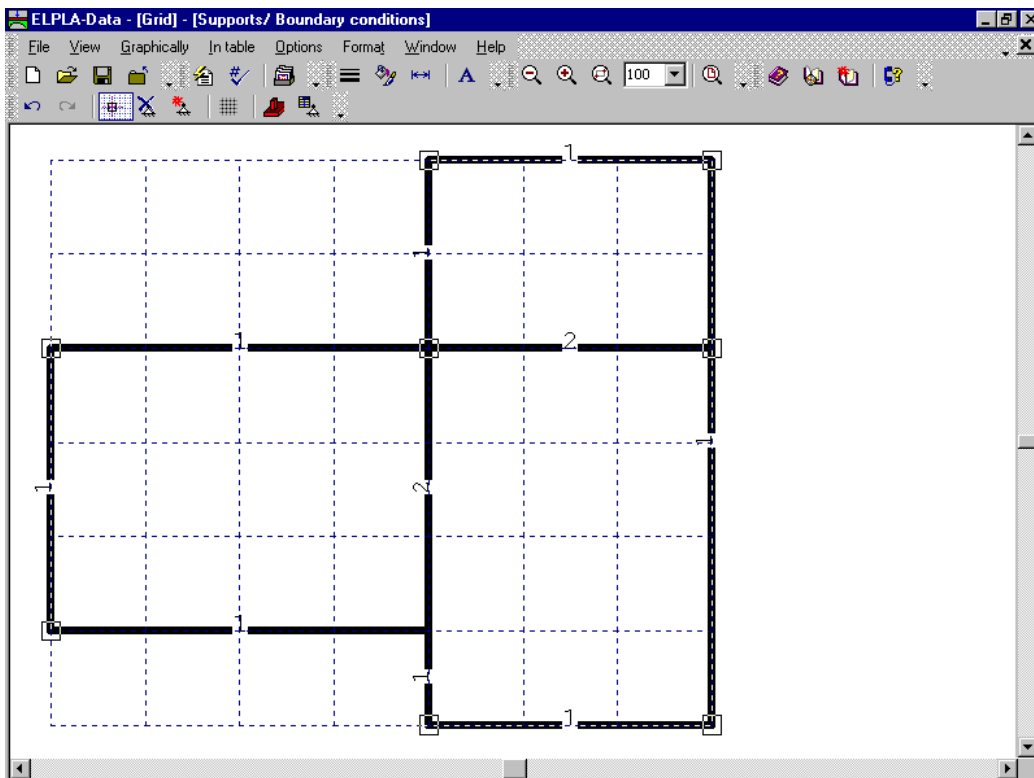


Figure 4-21 Selection of nodes that have supports

In this dialog box

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

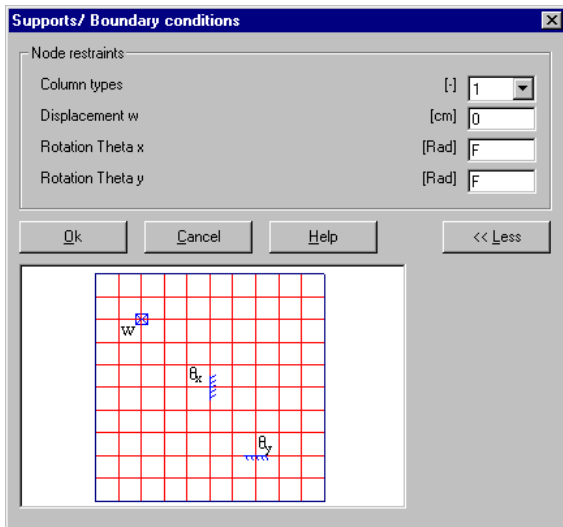


Figure 4-22 "Supports/ Boundary conditions" dialog box

ELPLA can calculate the punching stresses due to column reactions. 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. After you have completed the definition of supports, the screen should look like the following Figure 4-23.

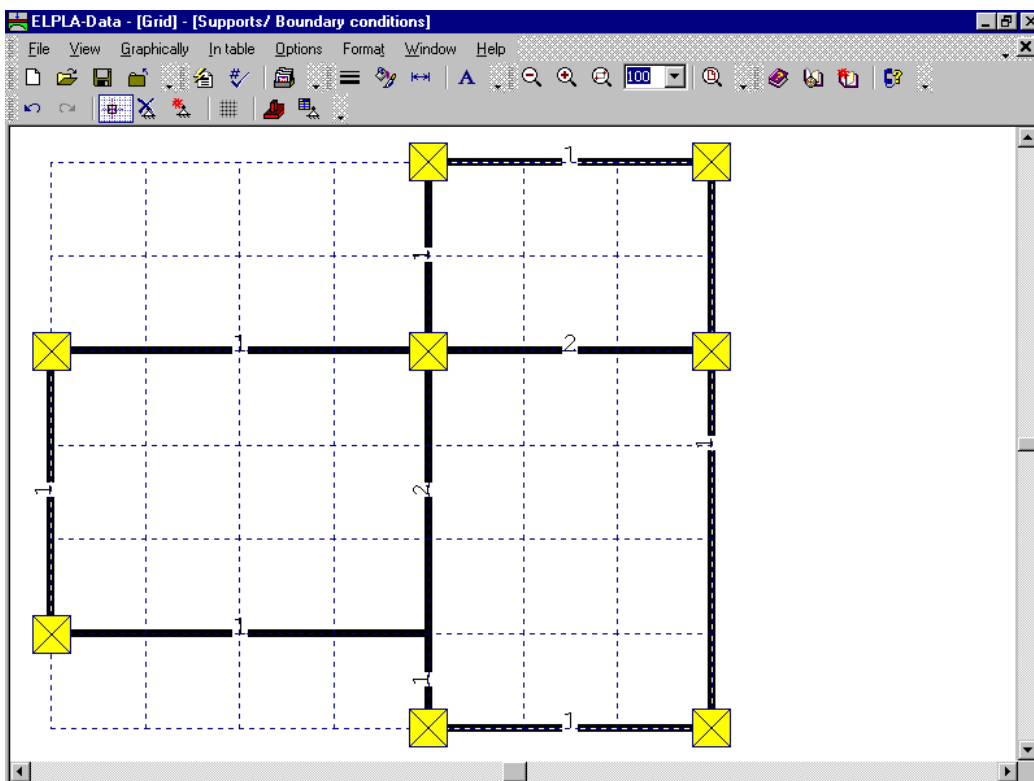


Figure 4-23 Supports on the screen

After entering supports, do the following two steps:

- Choose "Save supports/ Boundary conditions" command from "File" menu in Figure 4-23 to save the data of supports
- Choose "Close supports/ Boundary conditions" command from "File" menu in Figure 4-23 to close the "Supports/ Boundary conditions" embedded program and to return to the main window of *ELPLA-Data*

2.6 Loads

To define the concentrated load, choose "Loads" command from "Data" menu of *ELPLA-Data* window. The following embedded program in Figure 4-24 appears with girders on the net.

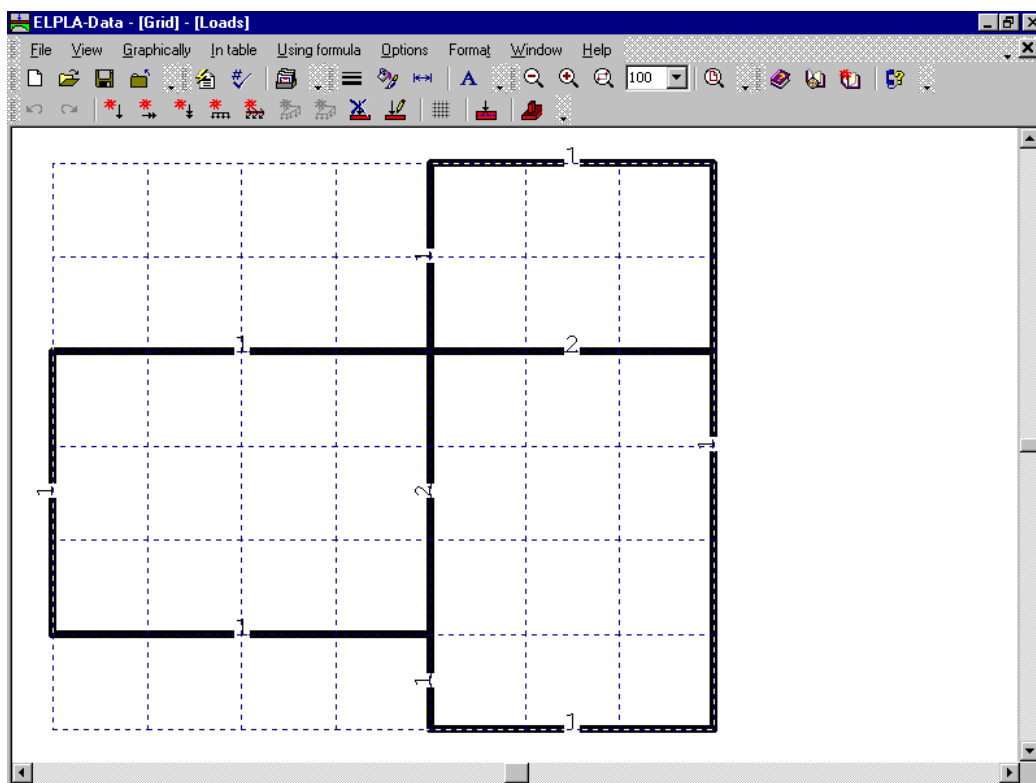


Figure 4-24 "Loads" embedded program

The vertical concentrated loads on the grid can be defined at any position (x , y) on the grid. The position of the load is independent on nodes of the net.

To enter the concentrated load choose "Point loads" command from "Graphically" menu in the window of Figure 4-24. When "Point loads" command is chosen, the cursor is changed from an arrow to a cross hair. Then the load can be defined by clicking on the screen (grid). The following dialog box in Figure 4-25 appears with the load value and coordinates.

In this dialog box

- Type 50 in the "Load value" edit box
- Click "OK" button

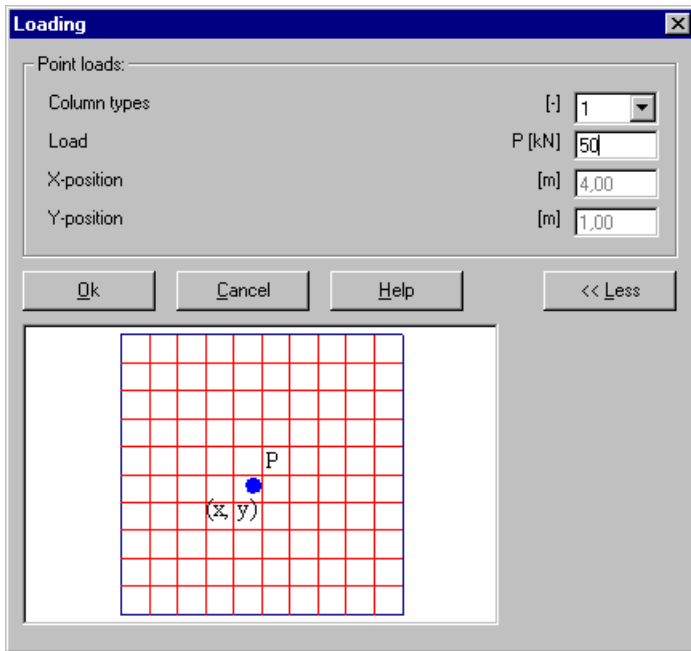


Figure 4-25 "Point loads P" dialog box

After you have completed the definition of the concentrated load, the screen should look like the following Figure 4-26.

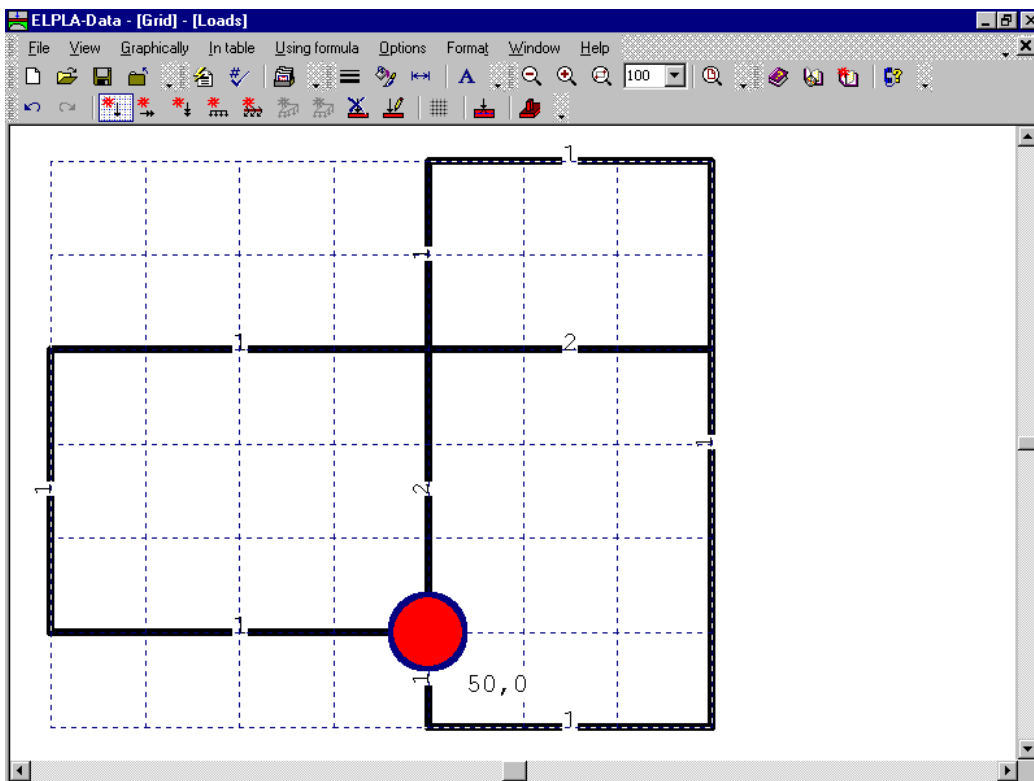


Figure 4-26 Concentrated load on the screen

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

- Choose "Save loads" command from "File" menu in Figure 4-26 to save the load data
- Choose "Close loads" command from "File" menu in Figure 4-26 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

Creating the project of the grid is now complete. See 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* by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data* window (Figure 4-27).



Figure 4-27 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, which 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

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 4-28 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.

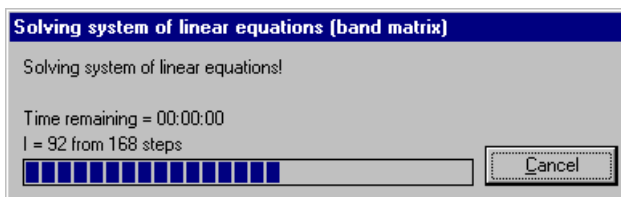


Figure 4-28 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution in Figure 4-29 appears. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

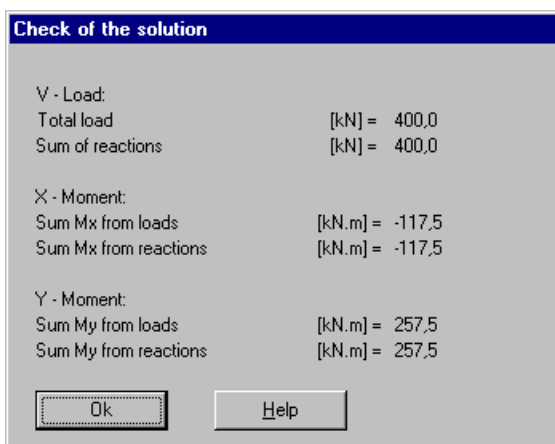


Figure 4-29 Menu "Check of the solution"

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

4 Viewing data and result graphics

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 4-30 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.

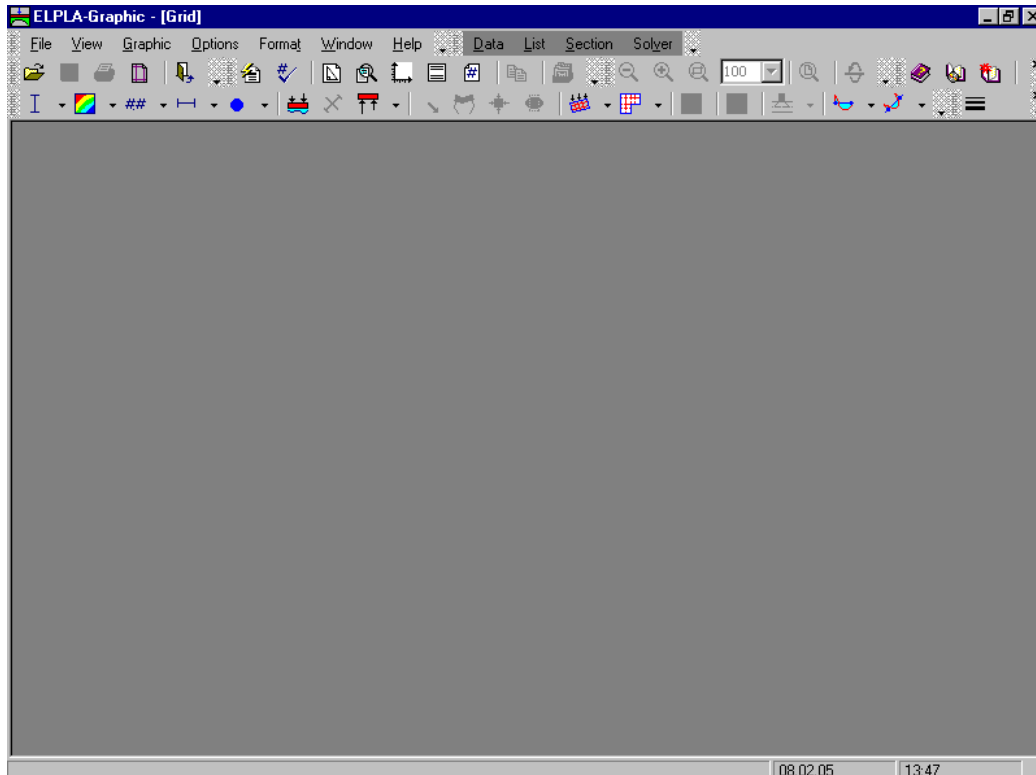


Figure 4-30 Opening screen of the sub program *ELPLA-Graphic*

To view the results of girders, choose "Girders" command and then "Internal forces in isometric view" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 4-31 appears.

In this option box

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

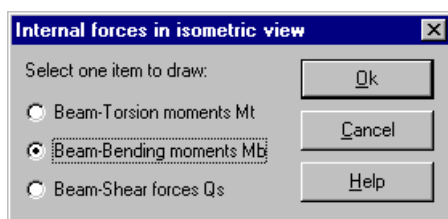


Figure 4-31 "Internal forces in isometric view" option box

The moments are now displayed for the girders as shown in Figure 4-32.

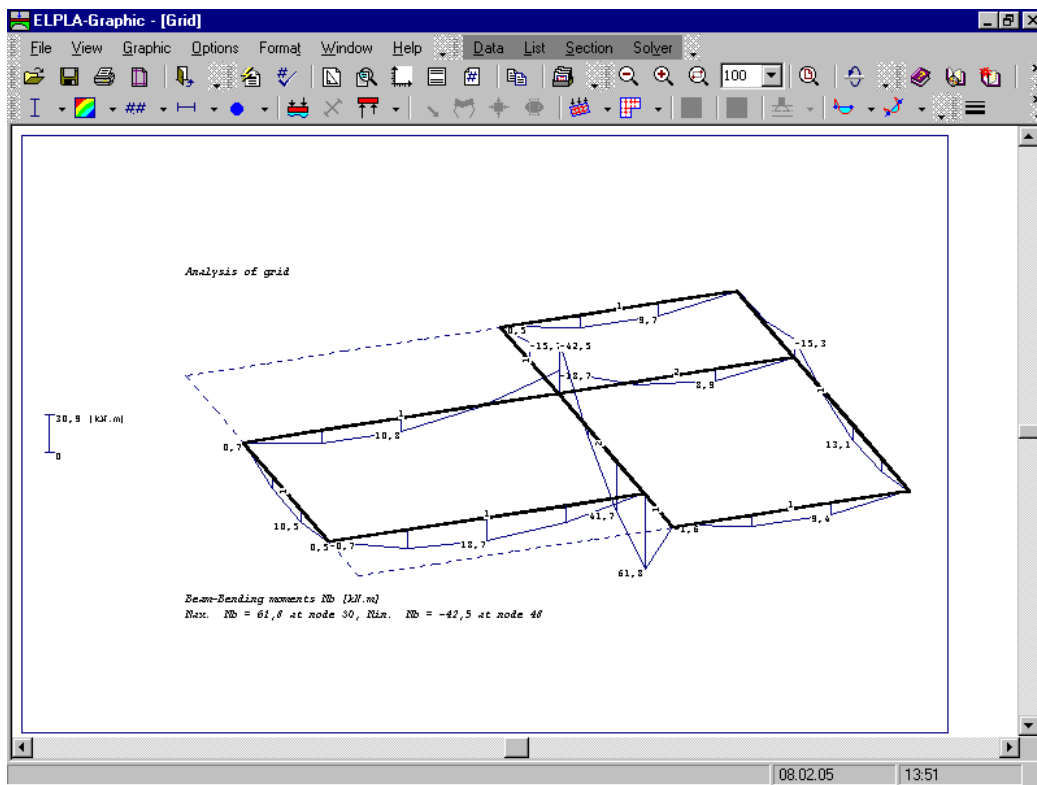


Figure 4-32 Beam-Bending moments M_b

5 Index

A

Analysis	3
Analysis type	4

B

Boundary conditions	13, 16
---------------------------	--------

C

Calculation	20
Calculation methods	4
Check of the solution	21
Computation of all	21
Cross section	10

D

Deformation	20
-------------------	----

F

FE-Net generation	7
-------------------------	---

G

Generation type	7
Geometry	3
Girder locations	11
Girder weight	10
Girders	9, 12
Girders as simple lines	14
Graphic	22
Grid	3

I

Internal forces	20, 22
-----------------------	--------

L

Load vector	20
Loads	3, 14, 18

N

New project	4
-------------------	---

P

Point loads	18
Project identification	6

S

Save as	6
Slab stiffness matrix	20
Solver	20
Supports	3, 5, 13, 16