Example 8

Analysis of beam

Contents Page

1 Description of the problem

An example for analyzing a beam is selected to illustrate some features of *ELPLA* for analyzing beams.

1.1 Loads and dimensions

The beam is shown in Figure 8-1. All beam members have cross section dimensions of 0.2 [m] * 0.5 [m]. Geometry of the beam and loads are shown in Figure 8-1.
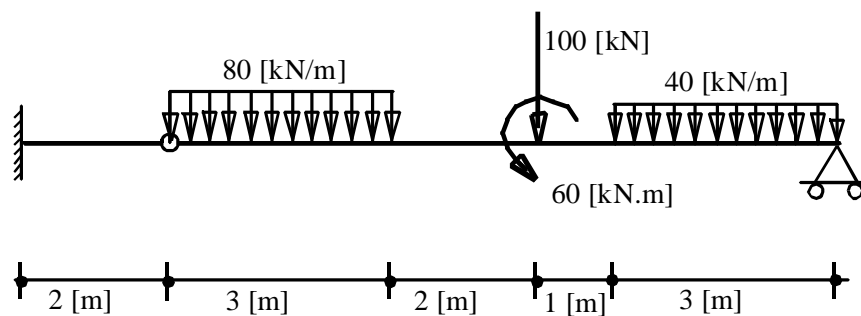

Figure 8-1   Geometry of the beam and loads

1.2 Beam material

Material of the beam is:

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

1.3 Analysis of the beam

To analyze the beam, the beam is divided in beam elements, each has a length of 1.0 [m]. 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 plane beam example. 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 8-2. The wizard will guide you through the steps required to create the system data of the project. As shown in Figure 8-2, the first form of the wizard is the "Analysis type" form.
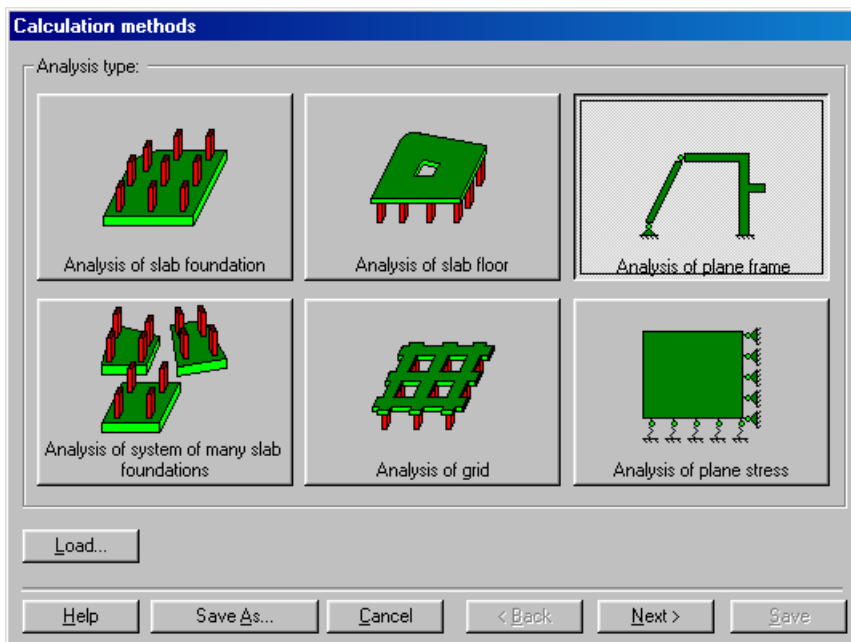
Figure 8-2   "Calculation method" wizard with "Analysis type" form

In the "Analysis type" form in Figure 8-2, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a plane beam problem, select "Analysis of plane beam". Then click "Next" button to go to the next form.

The next form is the "System symmetry" (Figure 8-3). In this form

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

The last form of the wizard assistant contains the "Option" list, Figure 8-4. 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
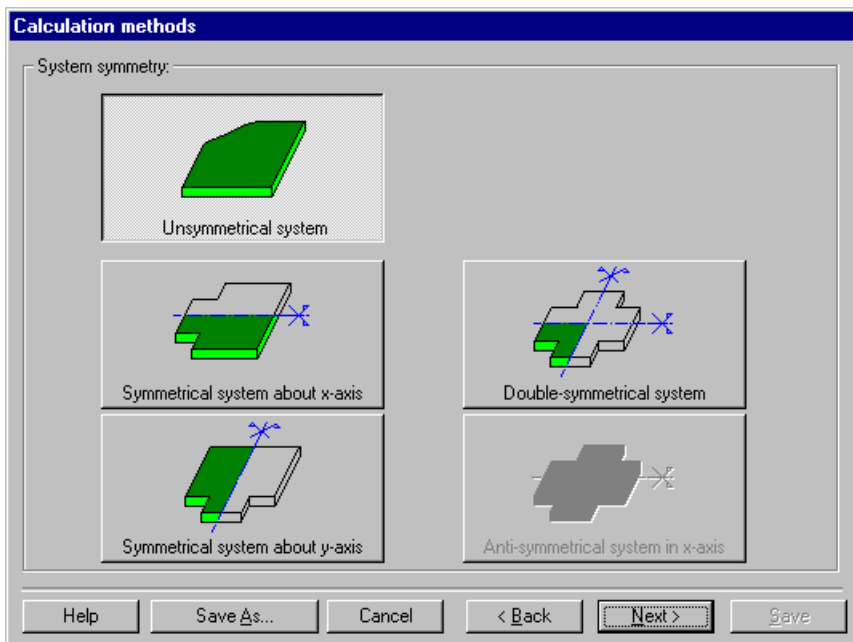-   Click "Save" button

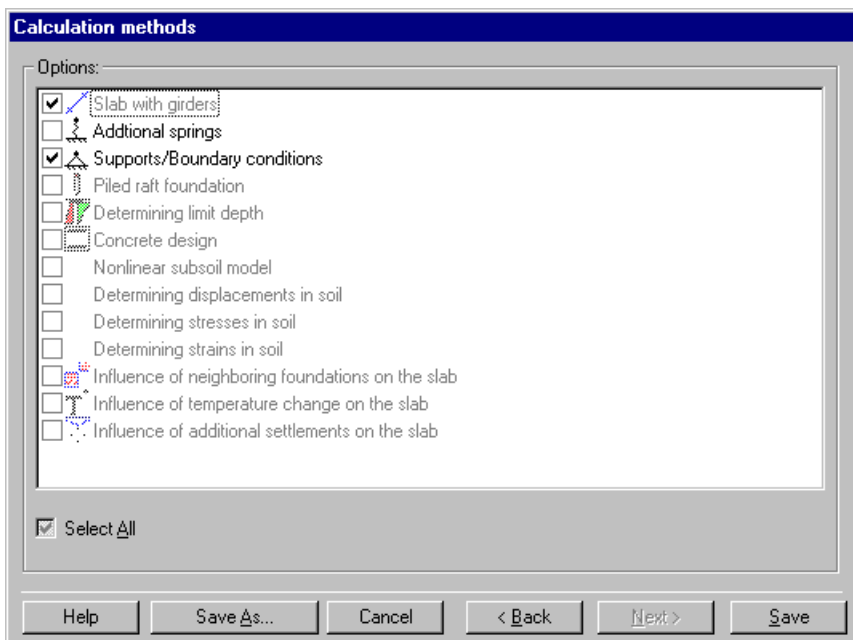Figure 8-3   "System symmetry" form

Figure 8-4   "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 8-5.

In this dialog box

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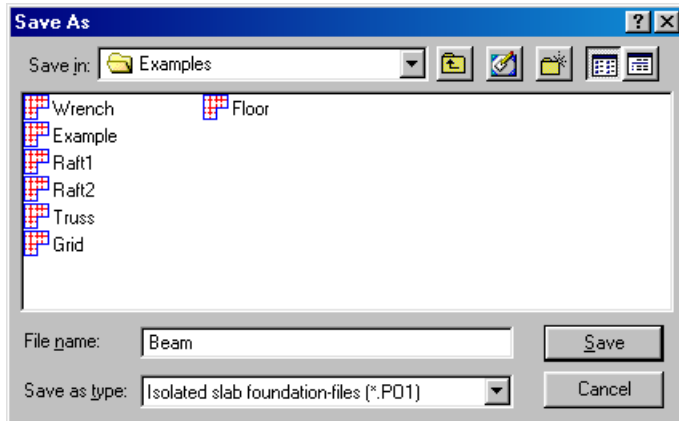

Figure 8-5   "Save as" dialog box

2.2   Project identification

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 8-6 appears.

In this dialog box

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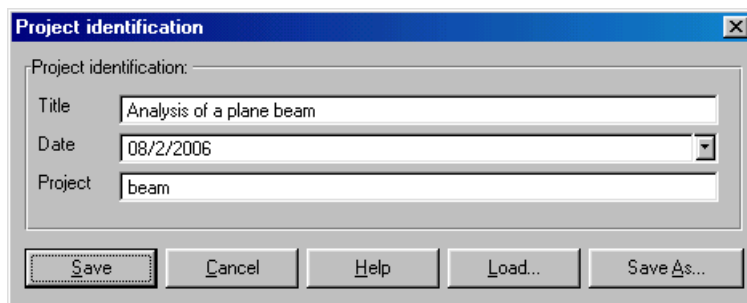- Click "Save" button

Figure 8-6   "Project identification" dialog box

2.3   FE-Net data

To enter the beam geometry, an imaginary net must be first defined with suitable dimensions and then the beam is entered lately by connecting the corresponding nodes on the net that define the beam shape. To define the net for the beam geometry, choose "FE-Net data" command from "Data" menu. The "FE-Net generation" wizard appears as shown in Figure 8-7. 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

- In the "Slab type" options, choose the rectangular slab option to create an imaginary net of a rectangular area
- Type 11 in the "Length of rectangular slab" edit box
- Type 2 in the "Width of rectangular slab" edit box
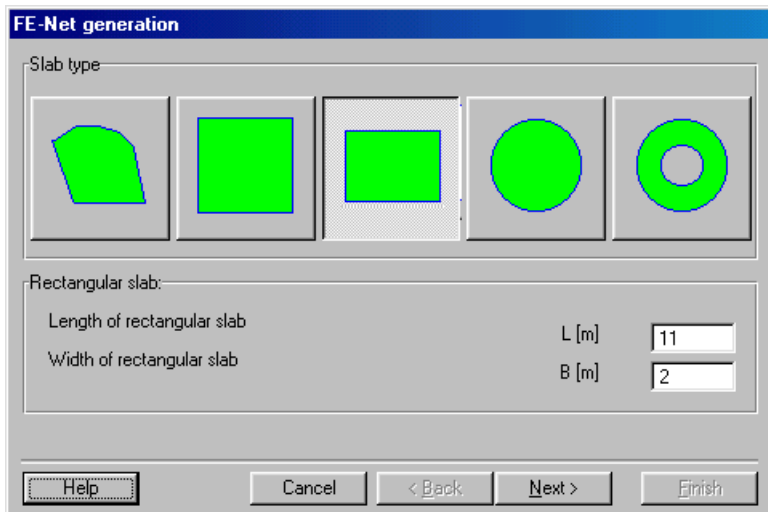- Click "Next" button to go to the next form

Figure 8-7   "FE-Net generation" wizard with "Slab type" form

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

In this form
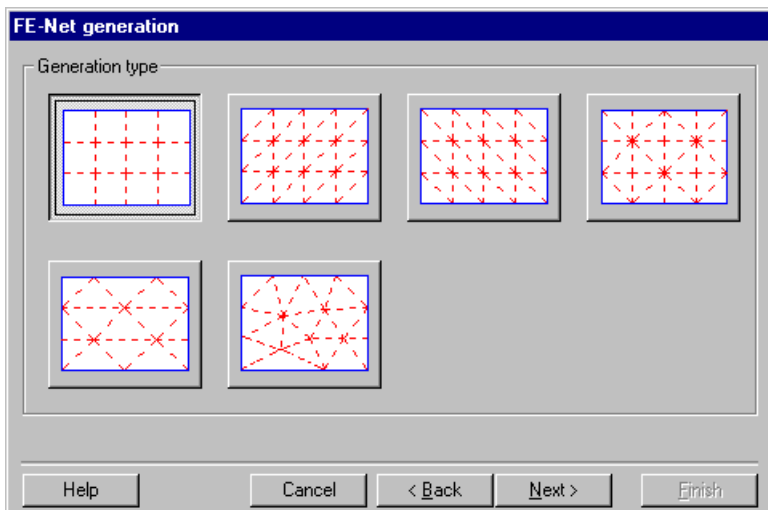
- Choose rectangular elements
- Click "Next"

Figure 8-8   "Generation type" form

After clicking "Next" button in "Generation type" form, the following "Grid definition" dialog box in Figure 8-9 appears with default values of constant element size.

In this dialog box

- Type 11 in the "No. of grid intervals" edit box for grids in *x*-direction
- Type 2 in the "No. of grid intervals" edit box for grids in *y*-direction
- Type 1 in the "Grid interval *Dx*" edit box
- Type 1 in the "Grid interval *Dy*" edit box
- Click "Finish" button

ELPLA will generate an imaginary net for a rectangular area of 11 [m] length and 2 [m] width with square elements of 1.0 [m] each side. The following embedded program in Figure 8-10 appears with the generated imaginary net.
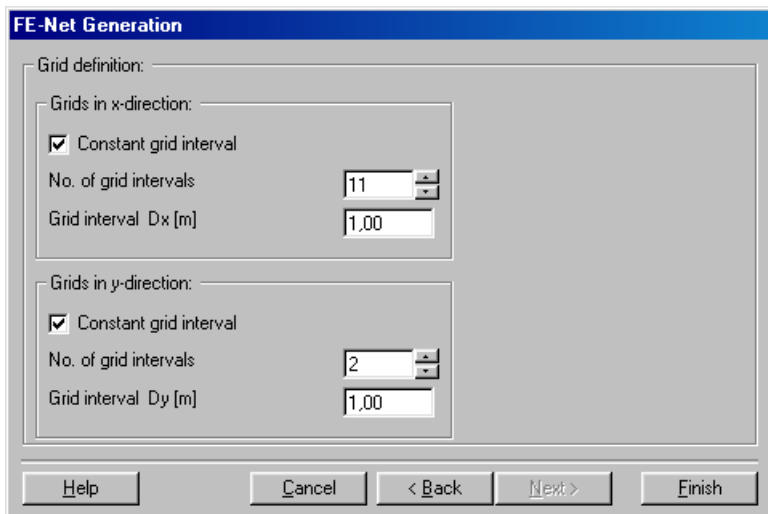

Figure 8-9   "FE-Net generation" dialog box
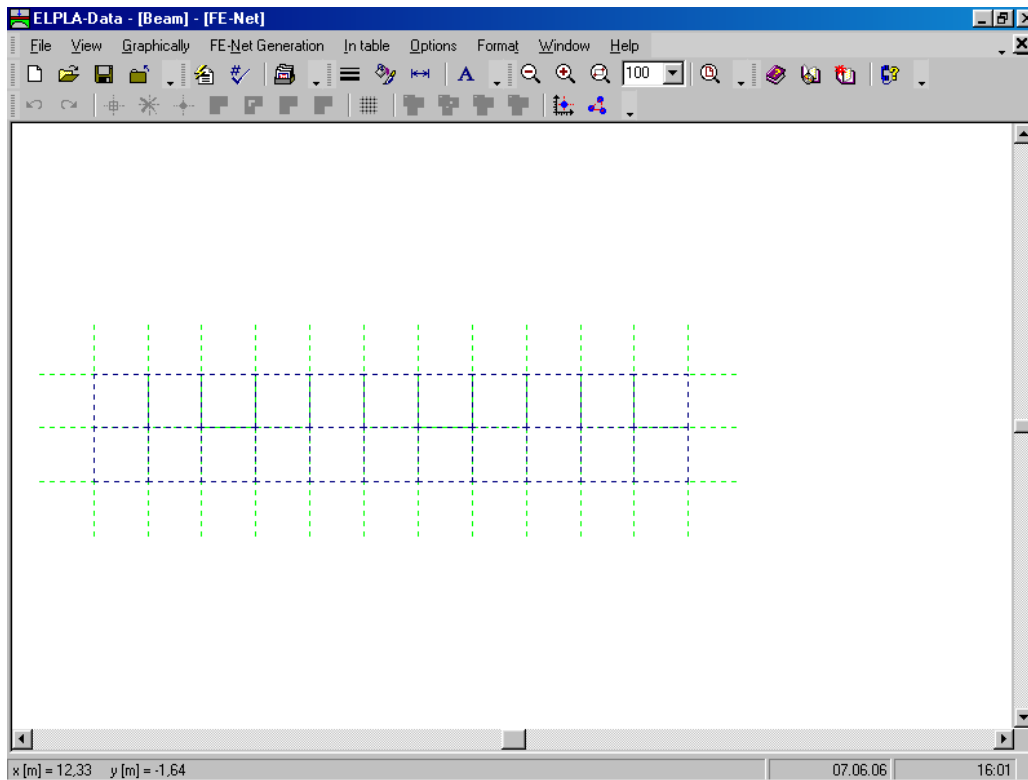
Figure 8-10   Imaginary net of a rectangular area on the screen

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

- Choose "Save FE-Net" command from "File" menu in Figure 8-10 to save the data of the imaginary net
- Choose "Close FE-Net" command from "File" menu in Figure 8-10 to close "FE-Net" embedded program and to return to the main window of *ELPLA-Data*

2.4   Girders

To define the beam members, choose "Girders" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 8-11 appears.
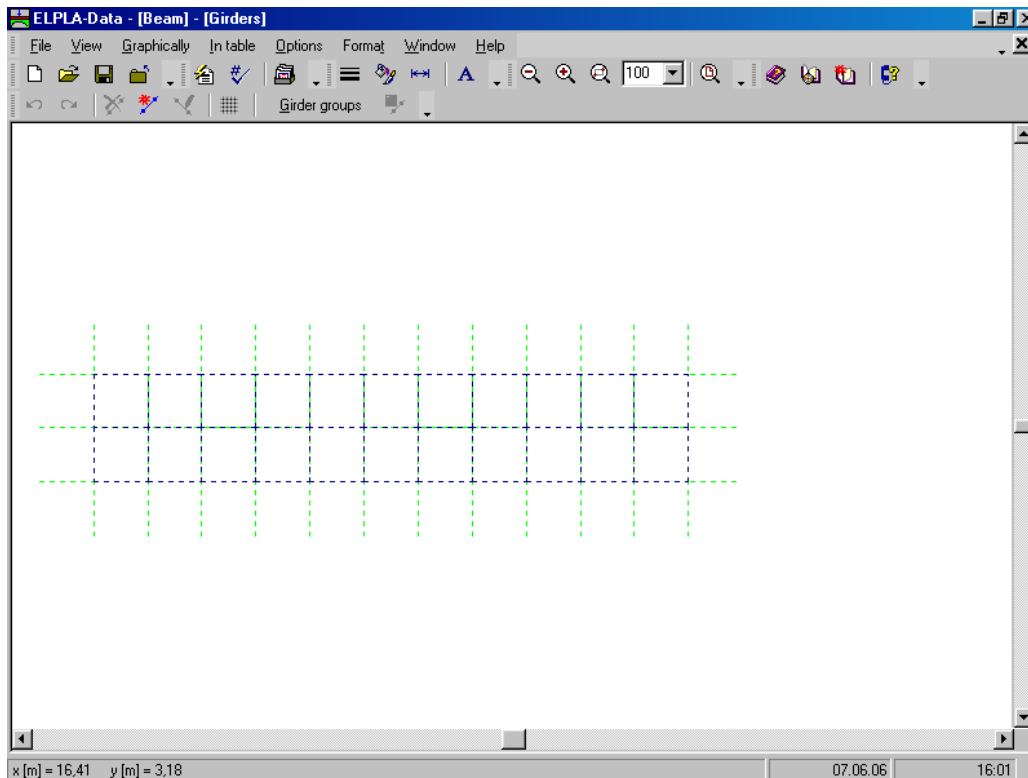
Figure 8-11   "Girders" embedded program

To enter the cross section of the girders

-   Choose "Girder groups" command from "In Table" menu in Figure 8-11.
    The following option box in Figure 8-12 appears
-   In this option box, select "Rectangular cross section"
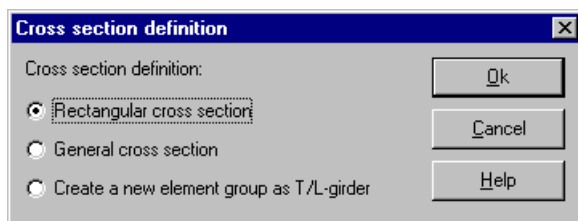-   Click "OK" button

Figure 8-12   "Cross section definition" option box

After clicking "OK" button in the "Cross section definition" option box, the following list box in Figure 8-13 appears.

In this list box

-   Enter the material properties of the girder, cross-section dimensions and the girder weight as indicated in Figure 8-13. This is done by entering the value in the corresponding cell and press "Enter" button
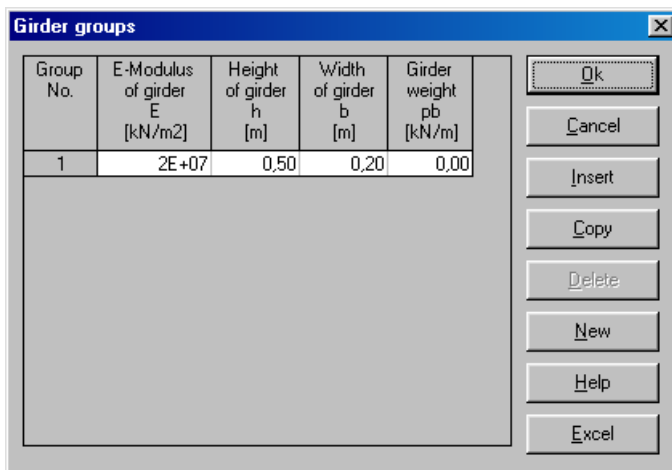-   Click "OK" button

Figure 8-13   "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 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 8-14. 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 and drag the mouse until the end node of that girder (Figure 8-14). Then, click on the end node. The "Girder elements" dialog box in Figure 8-15 appears

In this dialog box

- Select the group No.
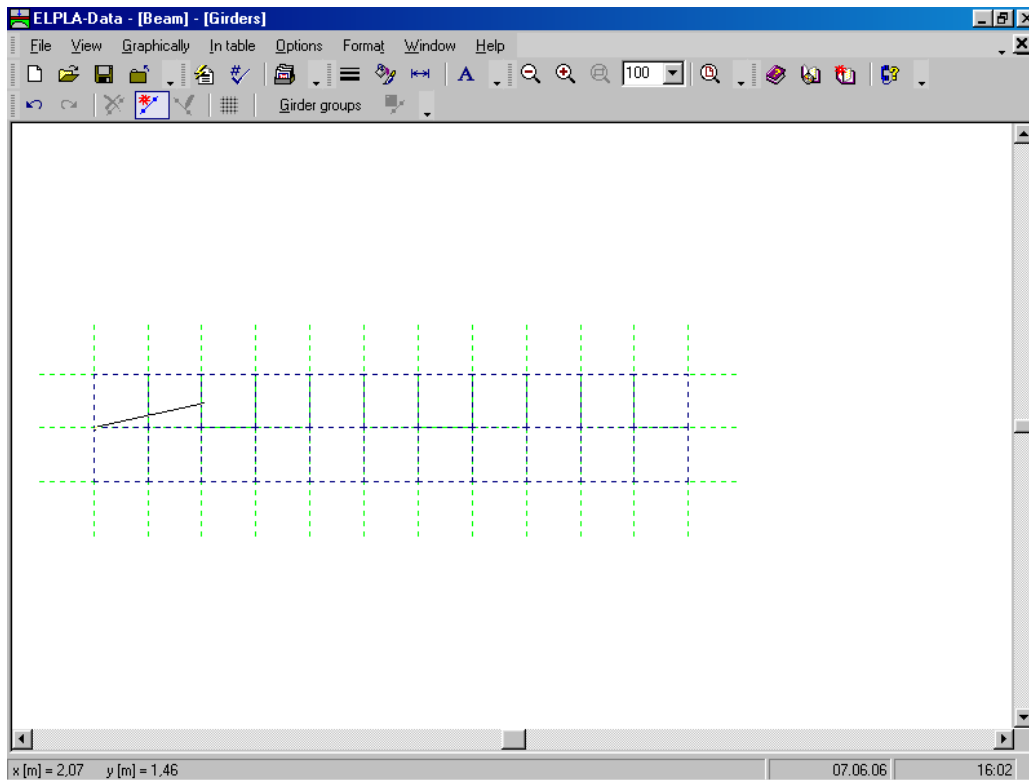- Click "OK" button

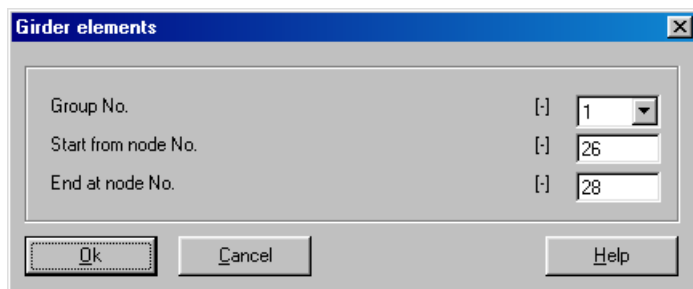Figure 8-14   Add girder by mouse

Figure 8-15   "Girder elements" dialog box

Now, the first girder is defined as shown in Figure 8-16. Note that *ELPLA* has typed automatically the girder type on it indicating the No. of girder group.

One note should be taken into consideration that girder ends are at exchanged nodes, for example supported nodes, hinged nodes and loaded nodes, so we should have five girders in this example.
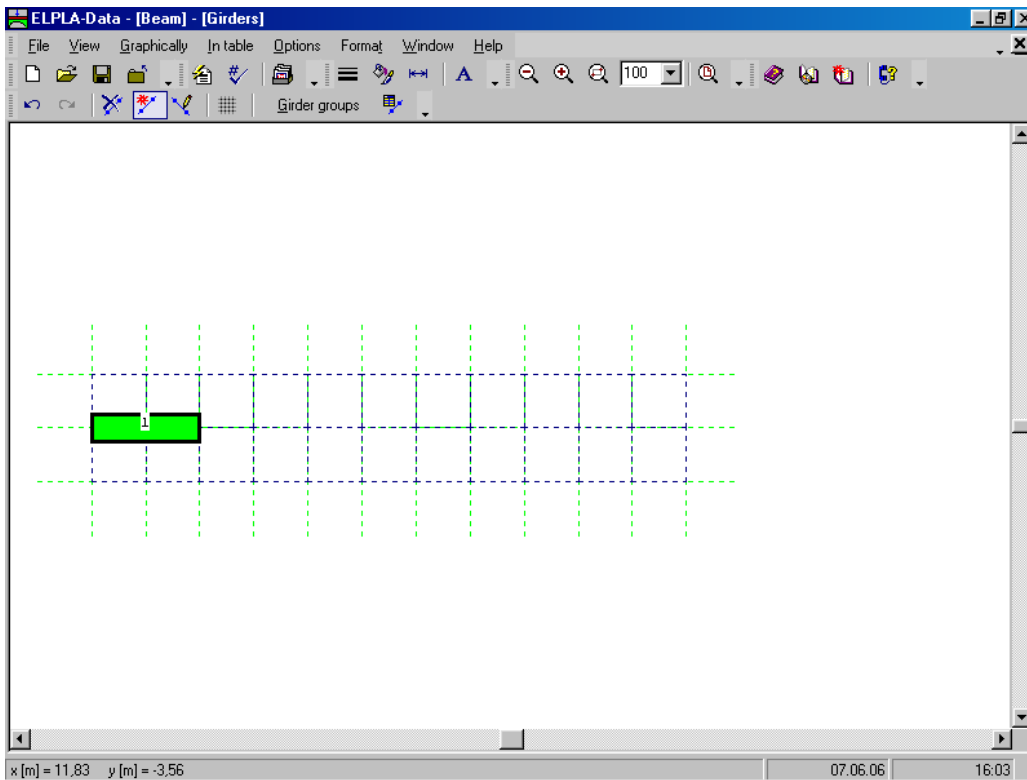
Figure 8-16   First girder on the screen

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 8-17.
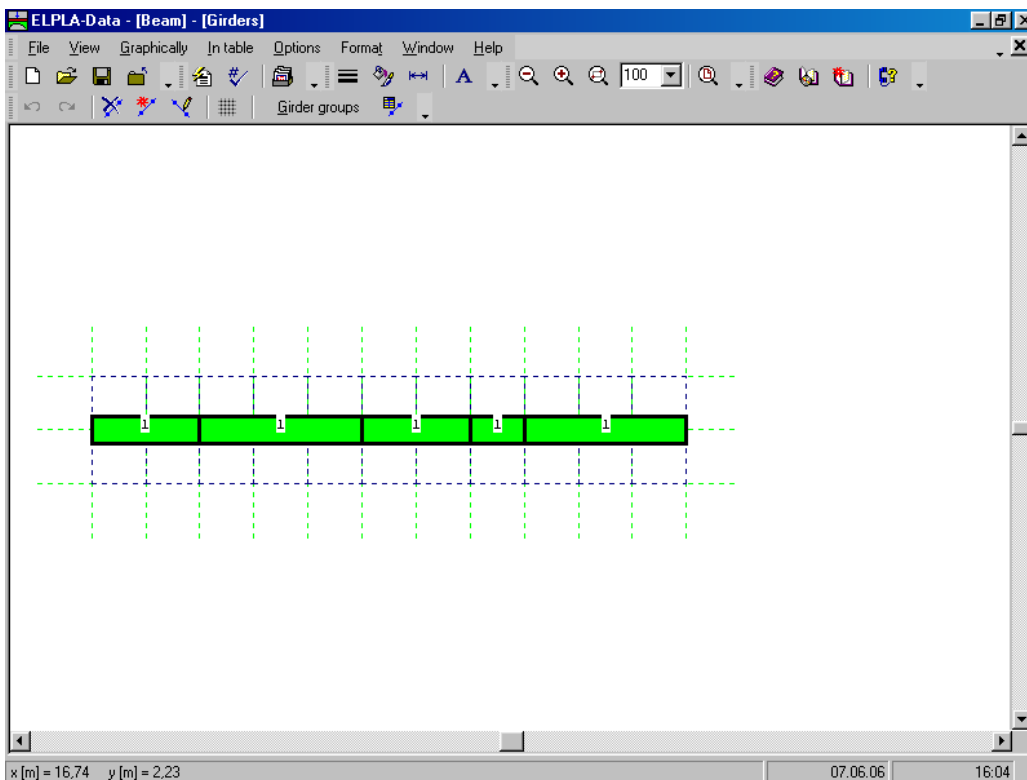

Figure 8-17   Girders on the screen

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

- Choose "Save girders" command from "File" menu in Figure 8-17 to save the data of girders
- Choose "Close girders" command from "File" menu in Figure 8-17 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*. The following embedded program in Figure 8-18 appears.
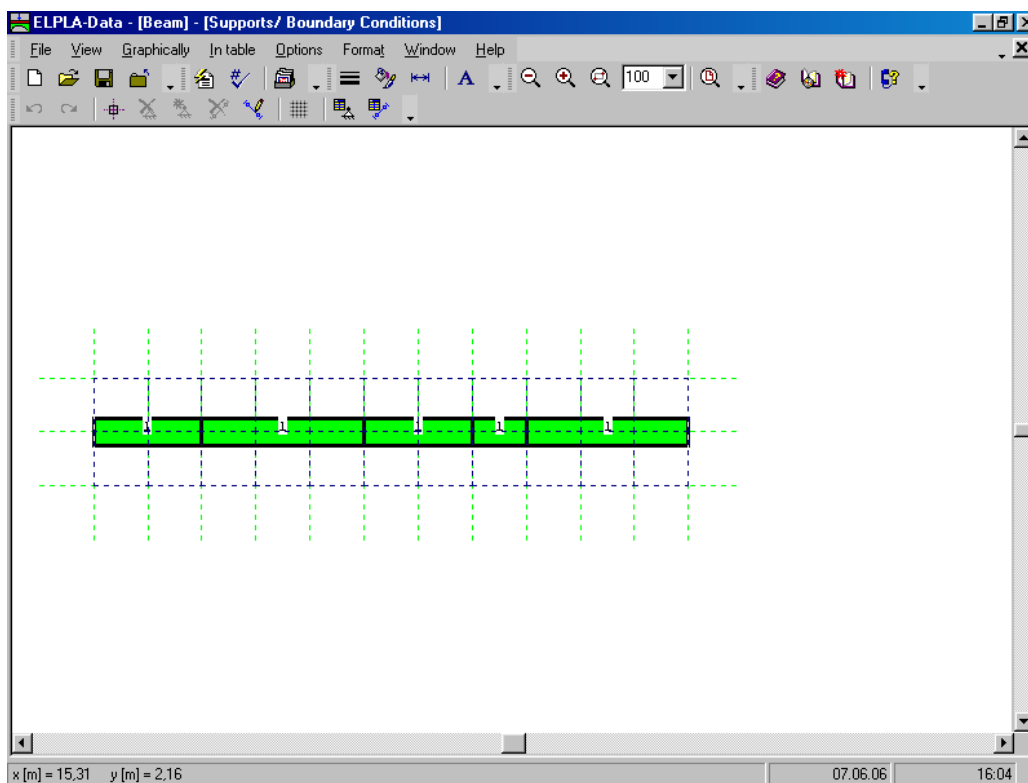

Figure 8-18   "Supports/ Boundary conditions" embedded program

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 8-18, 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 8-19 appears
- In the "General plot parameters" tab, uncheck the "Draw girder thickness" check box in the "Girder systems" dialog box
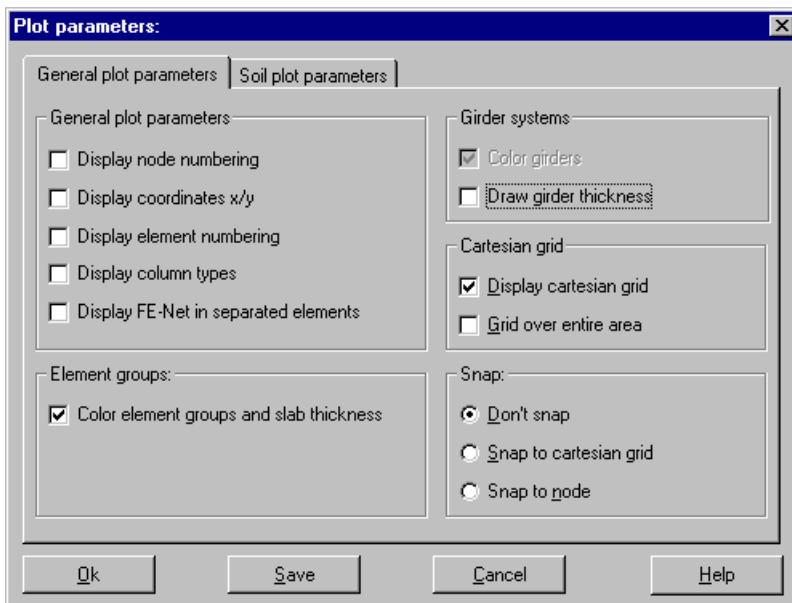- Click "OK" button

Figure 8-19   "Plot parameters" dialog box

After clicking "OK" in the "Plot parameters" dialog box, the girders are drawn as simple lines as shown in Figure 8-20.
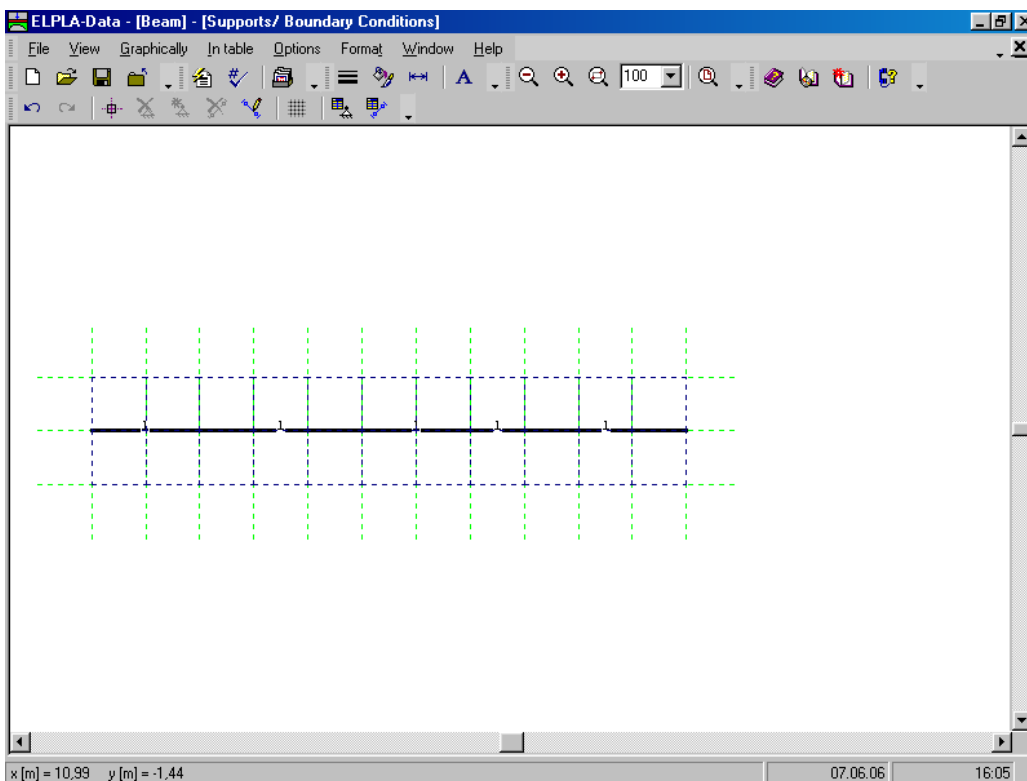

Figure 8-20   Girders are drawn as simple lines

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 the fixed support

- Choose "Select nodes" command from the "Graphically" menu in Figure 8-20. 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 the fixed supports as shown in Figure 8-21
- Choose "Add boundaries" command from "Graphically" menu in Figure 8-21
- "Supports/ Boundary conditions" dialog box in Figure 8-22 appears
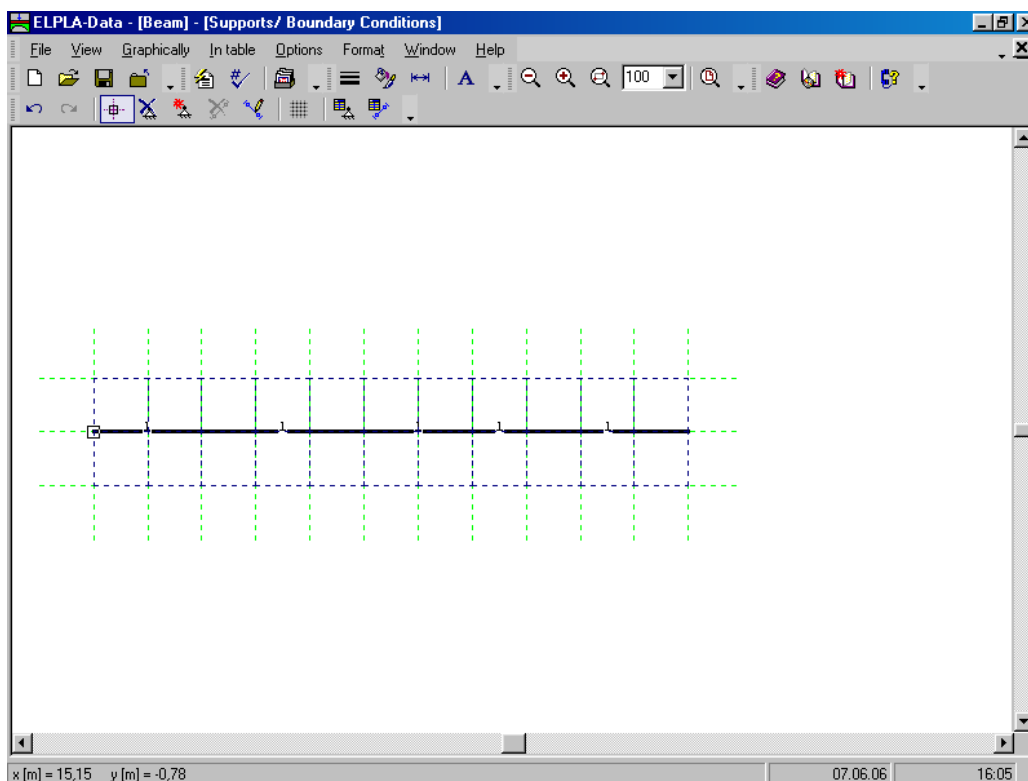

Figure 8-21   Selection of fixed support node

In this dialog box

- Type 0 in the "Displacement u" edit box to define the horizontal supports
- Type 0 in the "Displacement w" edit box to define the vertical supports
- Type 0 in the "Rotation Theta" edit box to define the moment supports
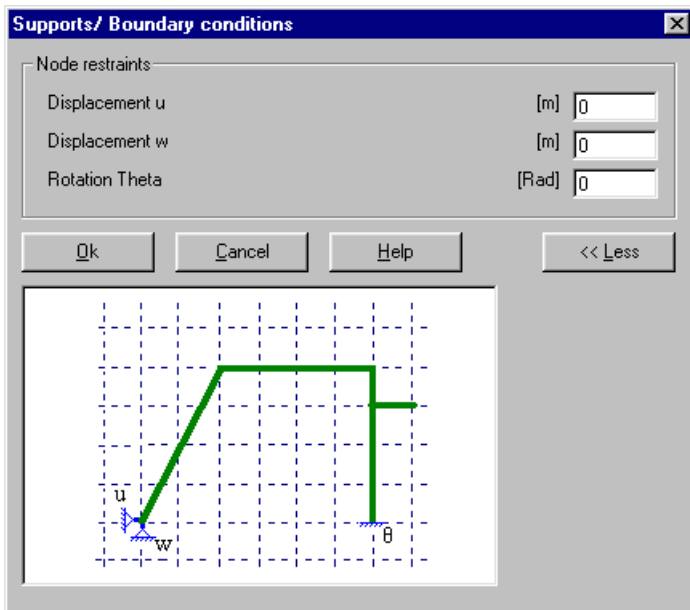- Click "OK" button

Figure 8-22   "Supports/ Boundary conditions" dialog box

To define the roller support, repeat the steps used to define the fixed support except that the support is free to rotate and the displacement in *x*-direction is free. To make the rotation of the roller support is free, type "F" in the "Rotation Theta" instead of typing "0". To make the displacement of the roller support is free in *x*-direction, type "F" in the "Displacement u" instead of typing "0". After you have completed the definition of supports, the screen should look like the following Figure 8-23.
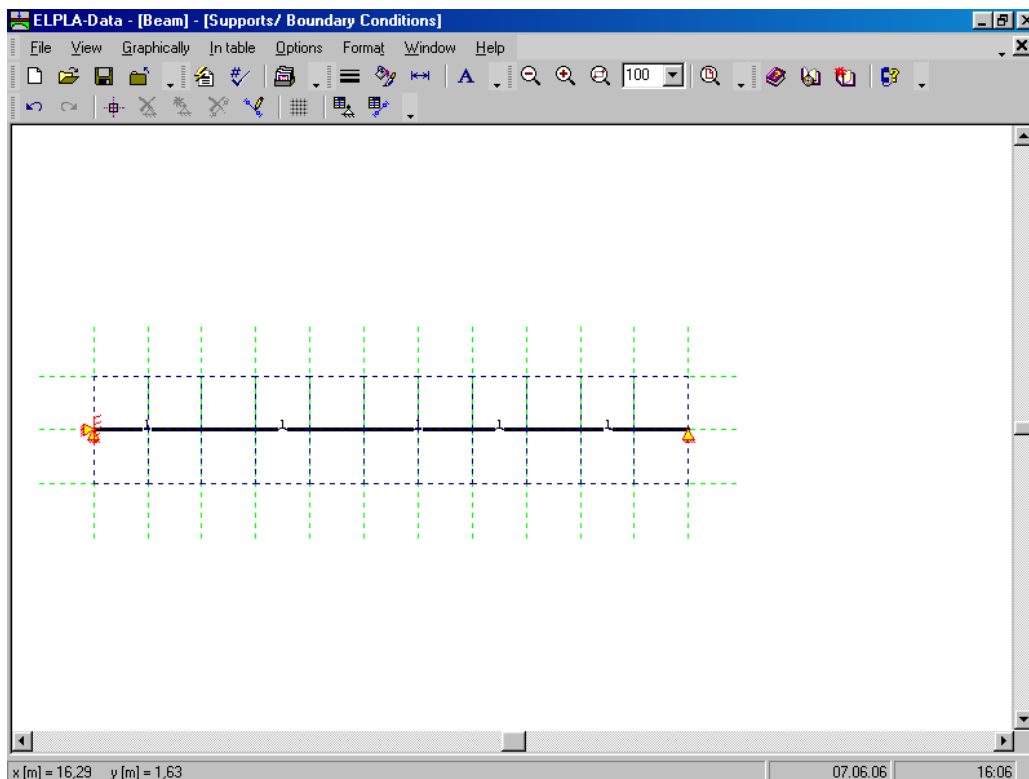

Figure 8-23   Supports on the screen

Defining hinges for beam members

Defining element hinges 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 element hinges on the net graphically.

To define the hinged node, the hinge is defined from two sides on both left and right elements that have a hinged end. Choose "Edit hinges" command from the "Graphically" menu in Figure 8-20. When "Edit hinges" command is chosen, the cursor will change from an arrow to a cross hair. Double click on the element that has a hinged end. "Edit hinges" dialog box in Figure 8-24 appears.
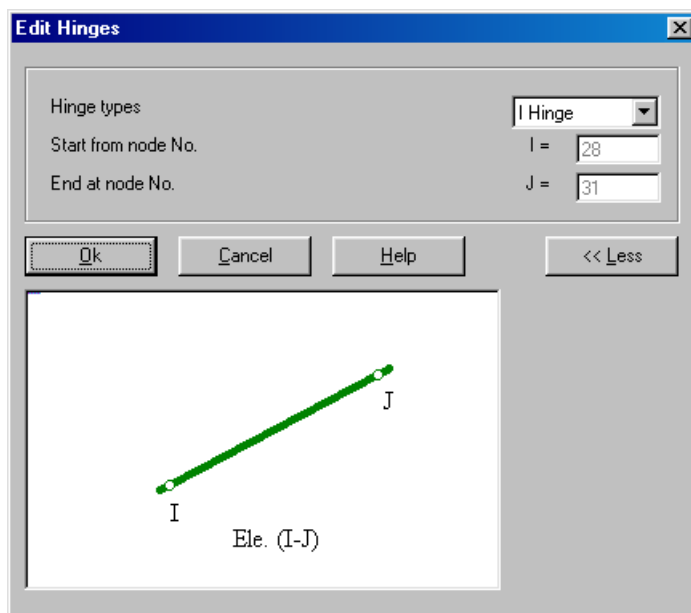

Figure 8-24   "Edit hinges" dialog box

In this dialog box, choose "I Hinge" from the hinge types list to define one end of the element as hinge. Then click "OK" button. The screen should look like Figure 8-25.
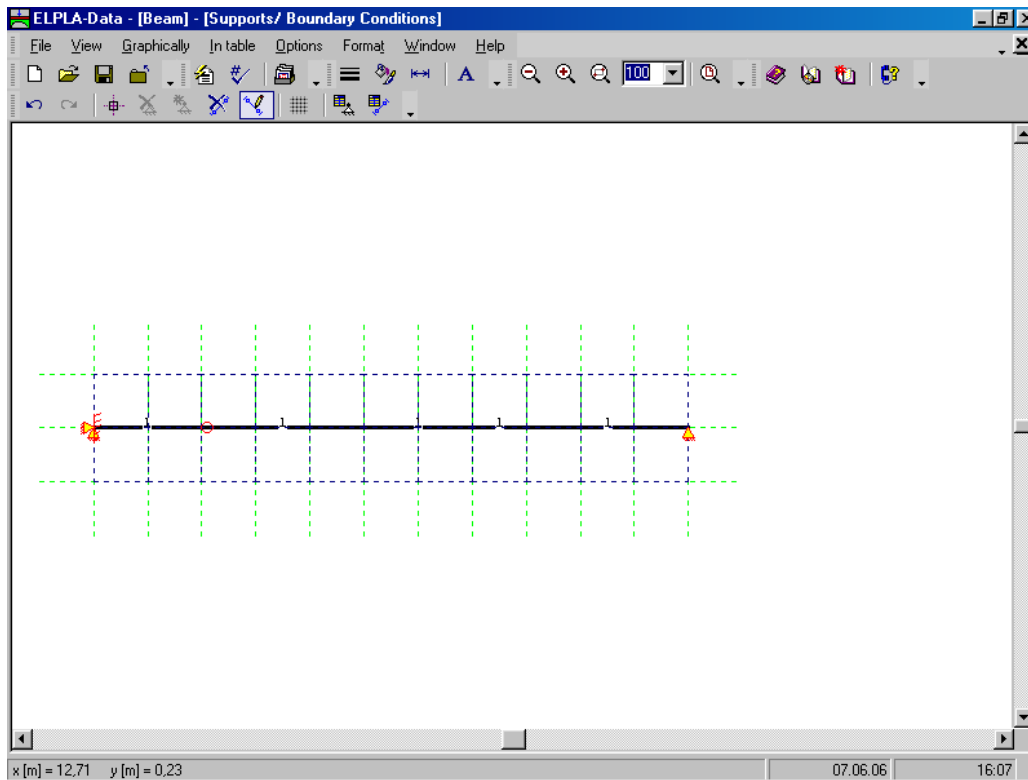
Figure 8-25   Hinged end of the element

Double click on the other element that has the hinged end. "Edit hinges" dialog box in Figure 8-24 appears.

Choose "J Hinge" from the hinge types list to define one end of the element as hinge. Then click "OK" button.

After you have completed the definition of element hinges, the screen should look like the following Figure 8-26.
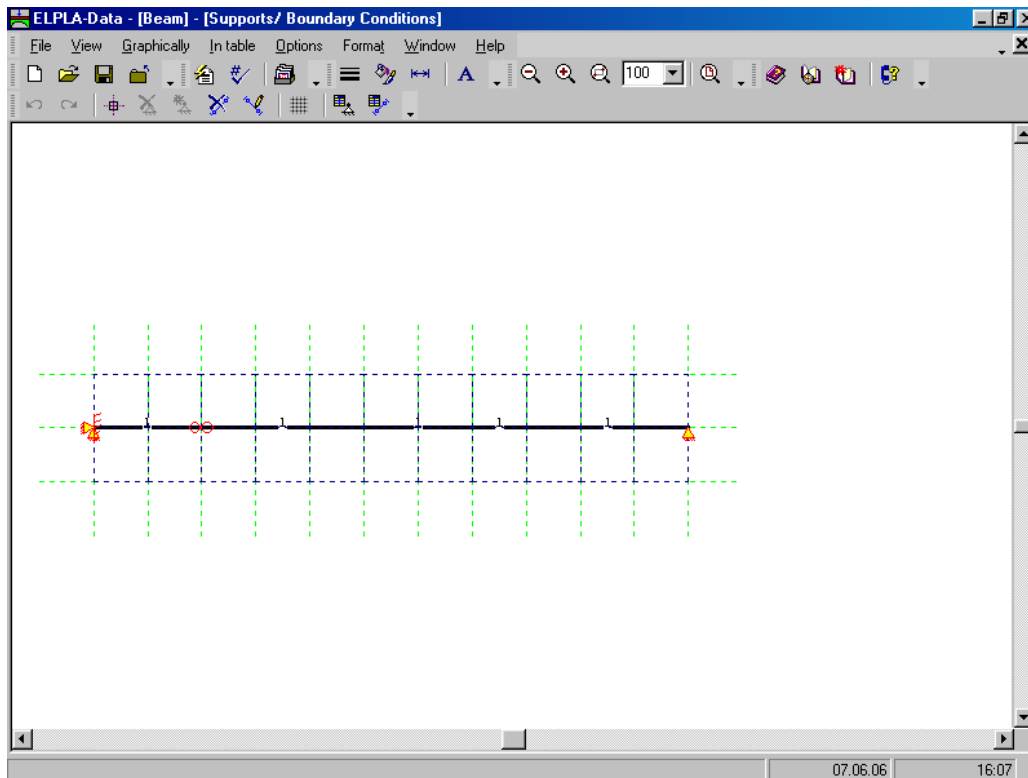
Figure 8-26   Supports and hinges on the screen

After entering supports and hinges, do the following two steps:

- Choose "Save supports/ Boundary conditions" command from "File" menu in Figure 8-23 to save the data of supports
- Choose "Close supports/ Boundary conditions" command from "File" menu in Figure 8-23 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*. The following embedded program in Figure 8-27 appears with girders on the net.
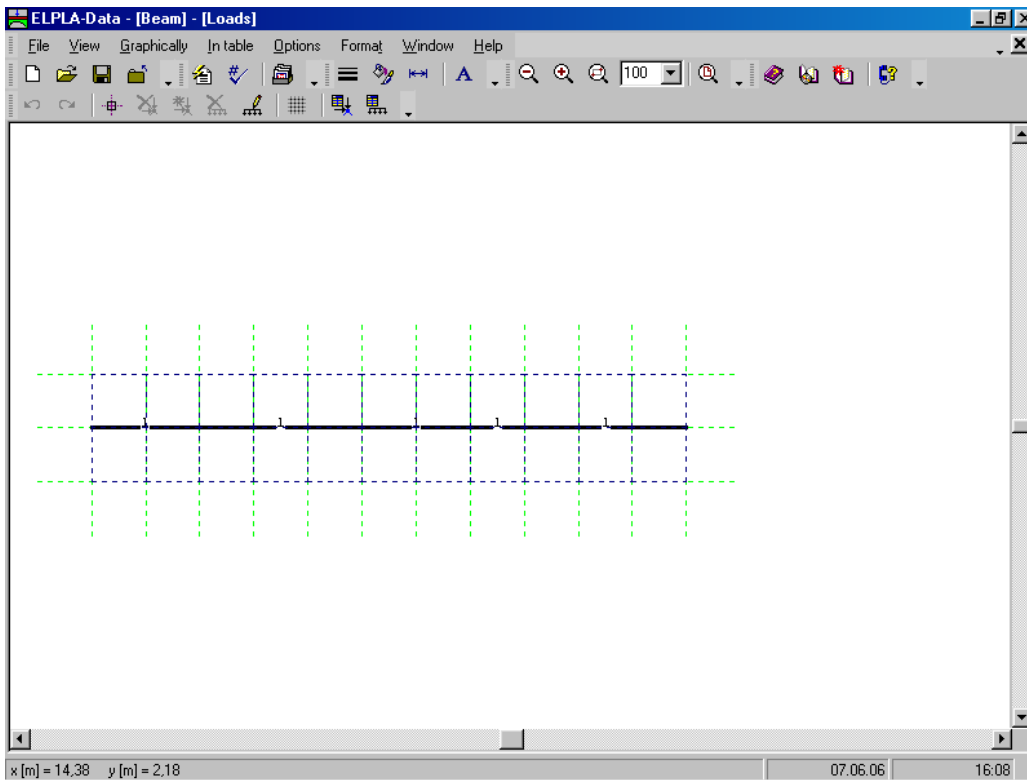
Figure 8-27   "Loads" embedded program

To enter the nodal loads, choose "Select nodes" command from "Graphically" menu in the window of Figure 8-27. When "Select nodes" command is chosen, the cursor is changed from an arrow to a cross hair. Choose a node to be loaded as shown in Figure 8-28.
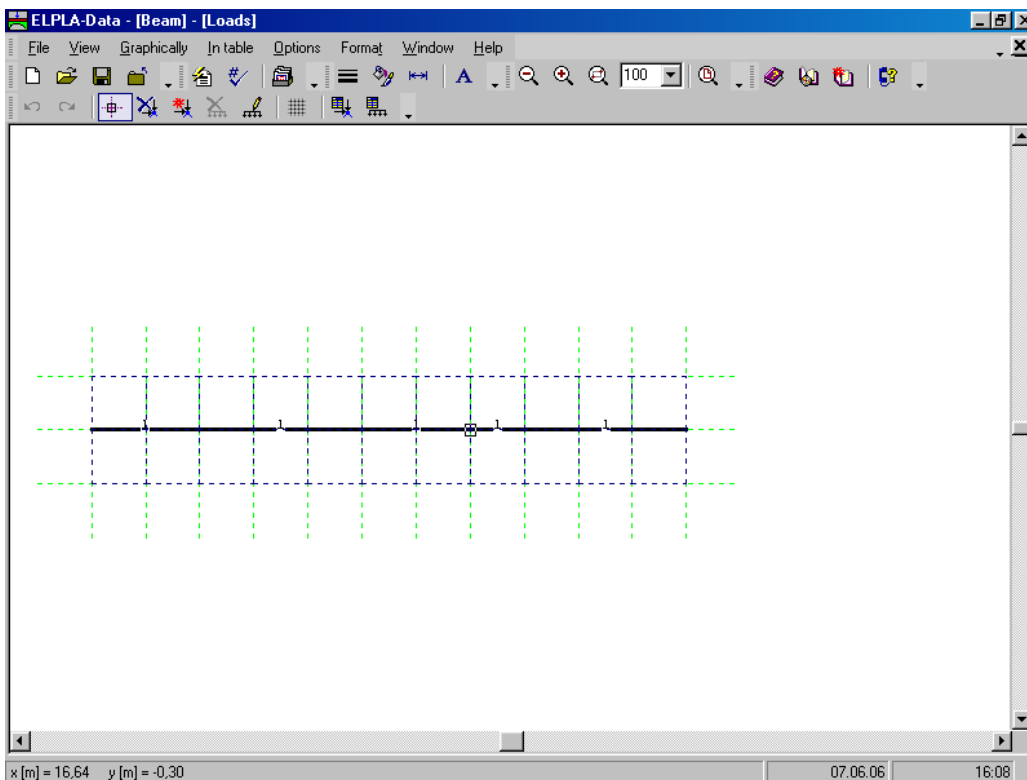
Figure 8-28   Selection of loaded node

Then choose "Add nodal loads" command from "Graphically" menu, the "Nodal loads" dialog box shown in Figure 8-29 appears. In this dialog box, type 100 in the "Load Pv" edit box to define the vertical nodal load, type also −60 in the "Moment M" edit box then click "OK" button. The node should be loaded as shown in Figure 8-30.
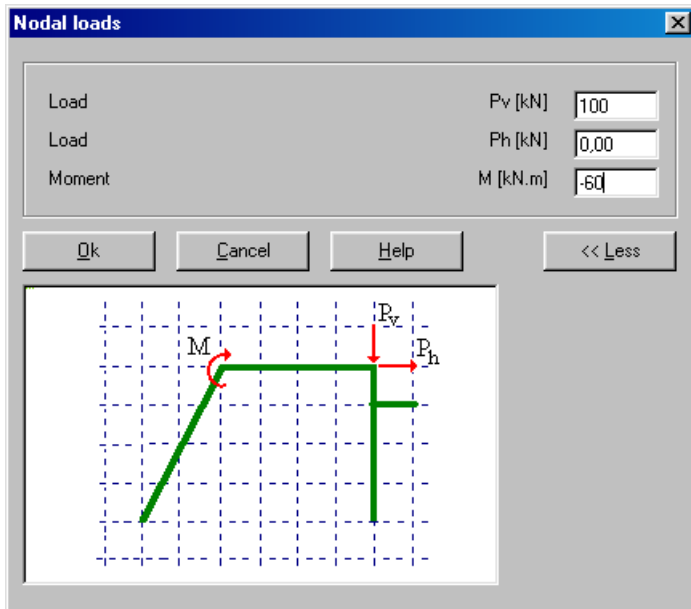

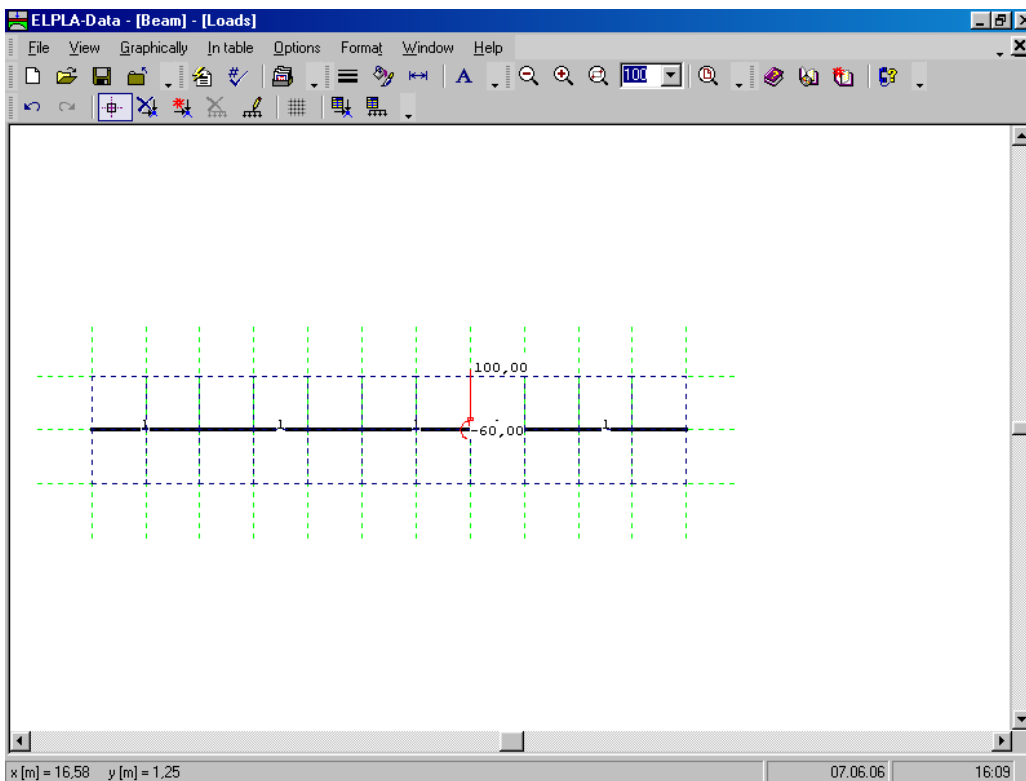Figure 8-29   "Nodal loads" dialog box

Figure 8-30   Nodal loads

To input the uniform distributed loads, choose "Edit member load" command from "Graphically" menu. Then the cursor will change from an arrow to a cross hair. Double click on the element that has the uniform distributed load. "Edit member load" dialog box in Figure 8-31 appears. In this dialog box, type 80 in the "Member loads" edit box to define the member load then click "OK" button. The loaded member is shown in Figure 8-32. The distributed loads in this example are "Self weight" types. To more about distributed load types, see the *ELPLA* User's Guide.
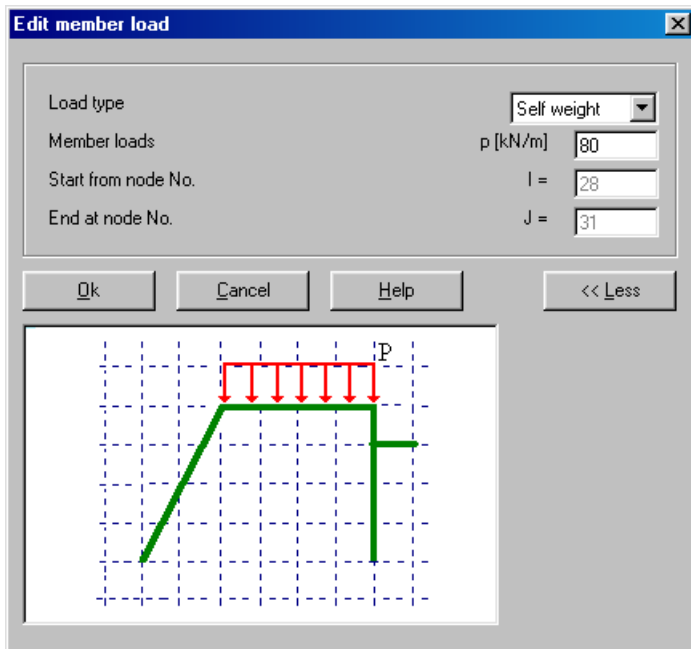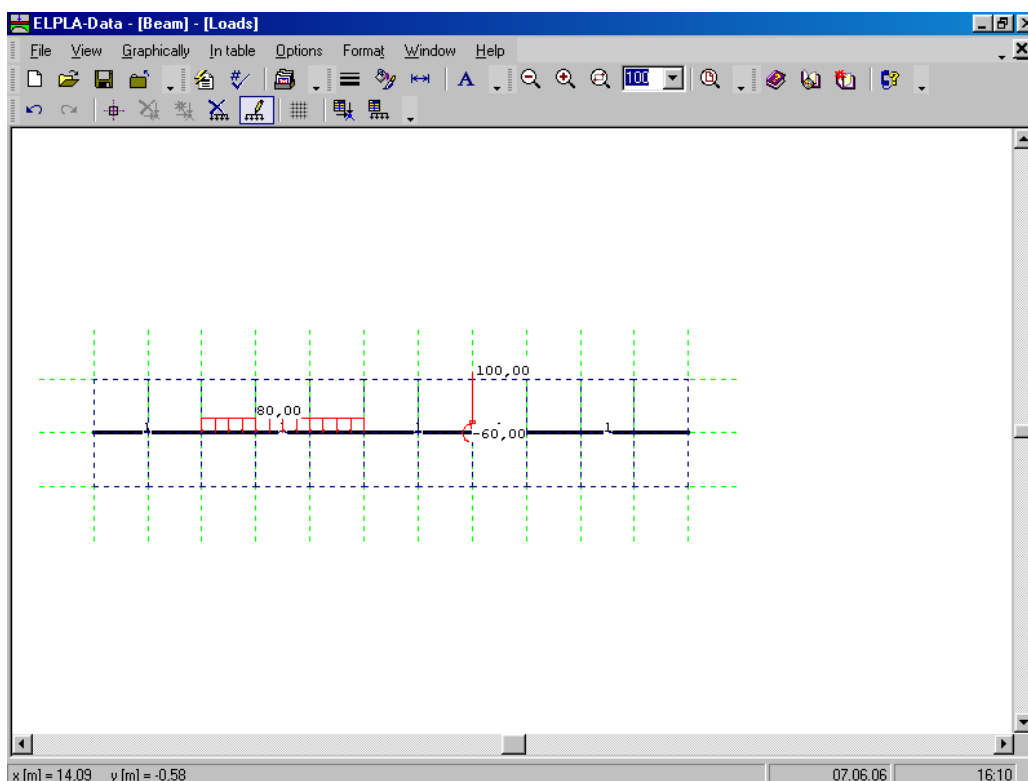

Figure 8-31   "Edit member load" dialog box

Figure 8-32   Loaded member

Use the same steps to input the uniform distributed load on the other loaded member. After you have completed the definition of the load data, the screen should look like the following Figure 8-33.
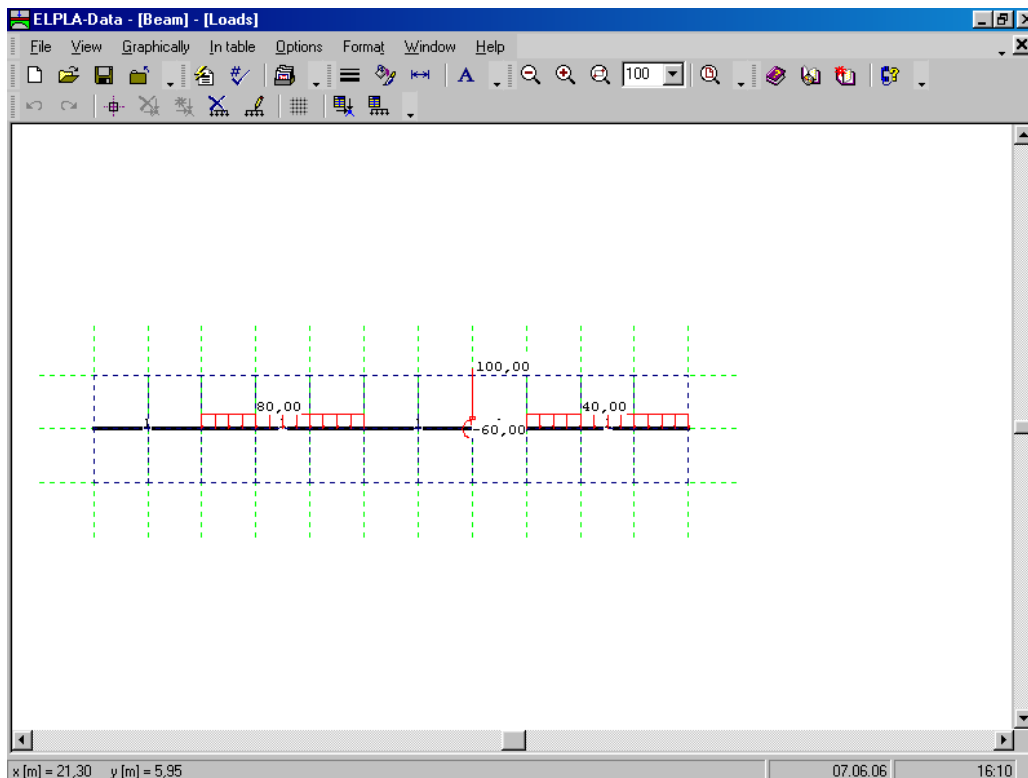

Figure 8-33   Beam loads on the screen

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

- Choose "Save loads" command from "File" menu in Figure 8-33 to save the load data
- Choose "Close loads" command from "File" menu in Figure 8-33 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

You can get the supports and the system of loading together on the screen. Choose "View grouping" command from "Options" list and the following dialog box in Figure 8-34 appears.
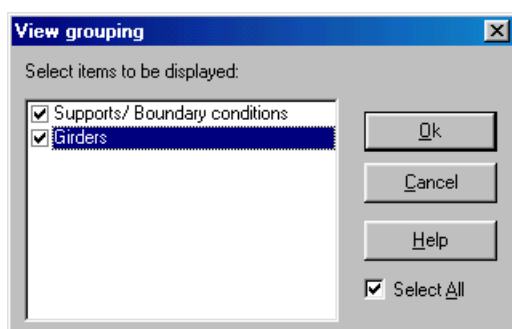

Figure 8-34   View grouping dialog box

Check the "Supports/Boundary conditions" check box then click the "OK" button. The screen should look like Figure 8-35.
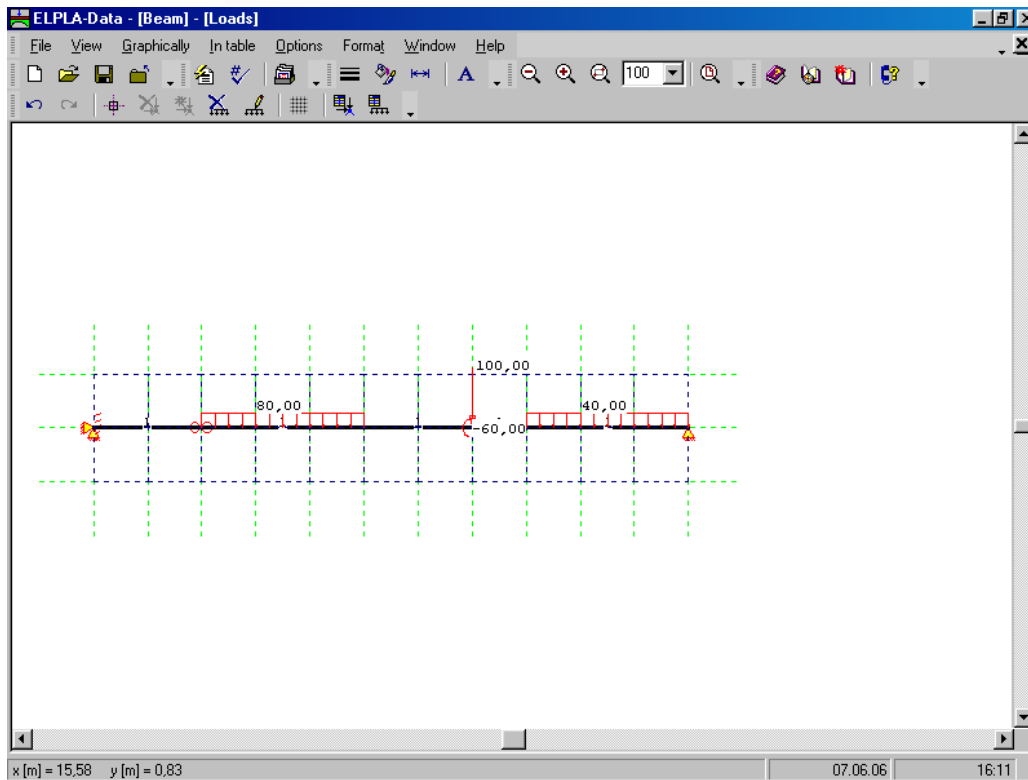
Figure 8-35   Beam supports and system of loading on the screen

Creating the project of the beam 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 8-36 appears.
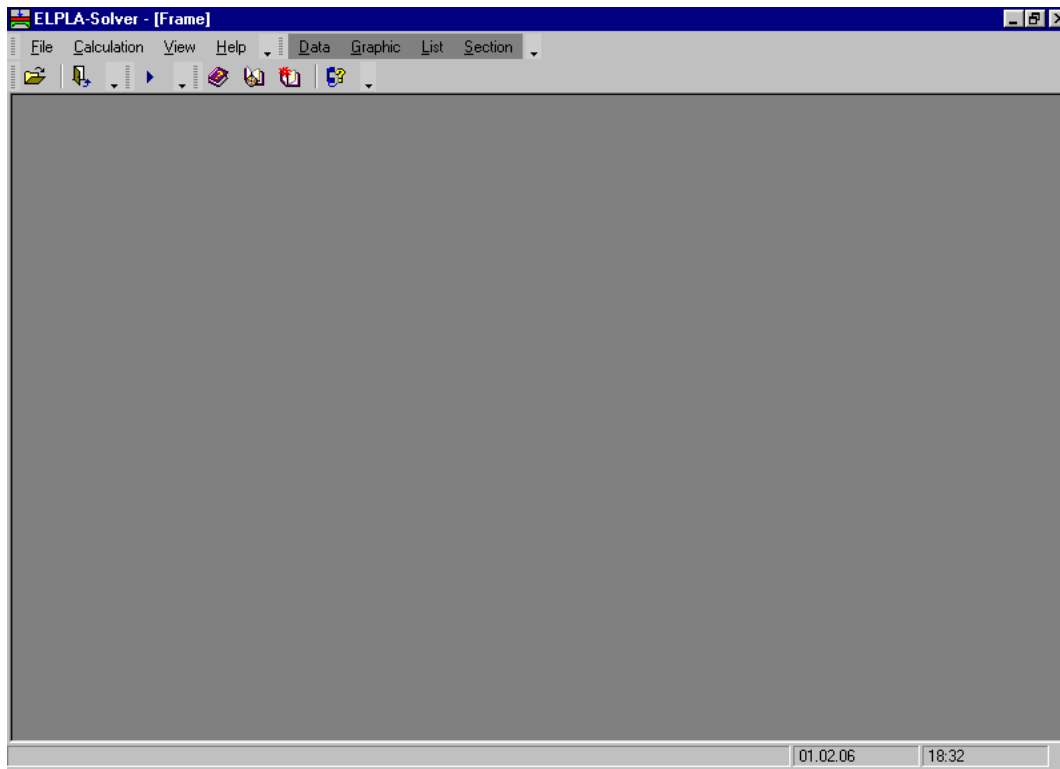
Figure 8-36   Opening screen of the sub program *ELPLA-Solver*

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 active 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:

- Preparing the calculation
- Analysis of plane beam

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 8-37 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.
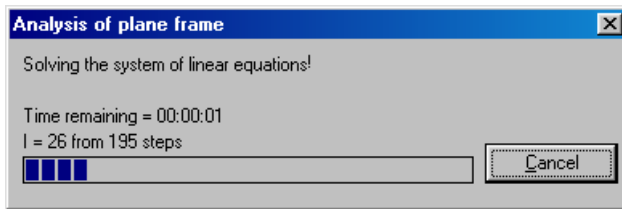
Figure 8-37   Analysis progress menu

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 8-38 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.
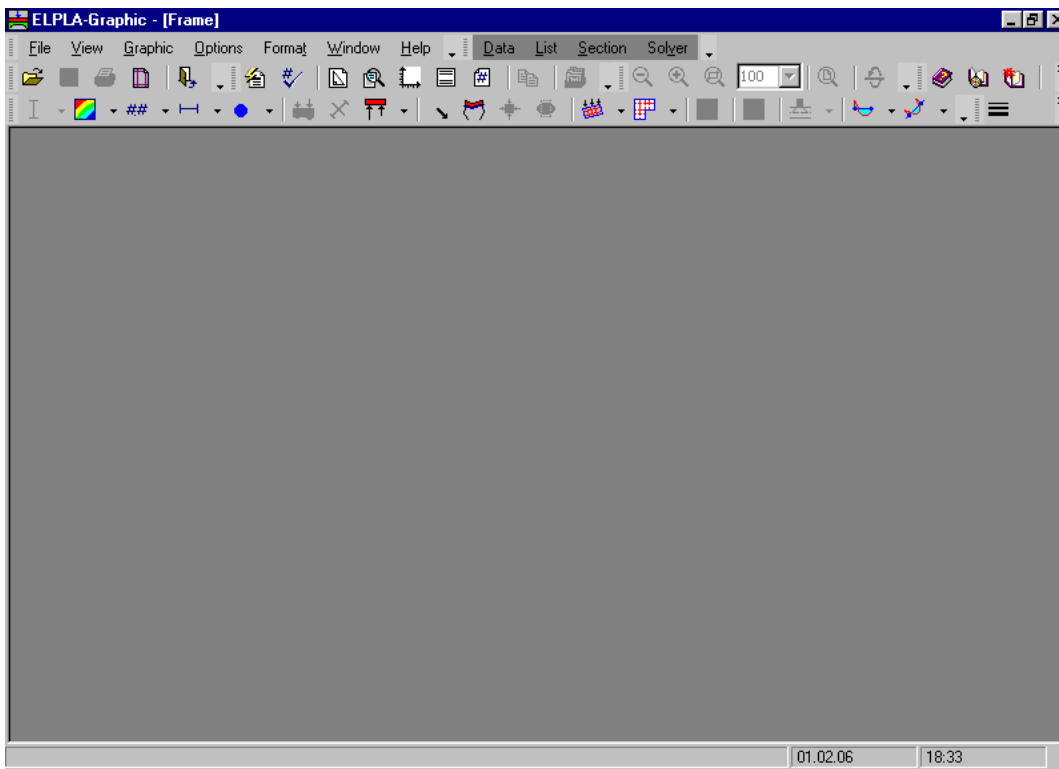
Figure 8-38   Opening screen of the sub program *ELPLA-Graphic*

To view the results of girders, choose "Beam results" command and then "Distribution of internal forces (in plane)" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 8-39 appears.

In this option box

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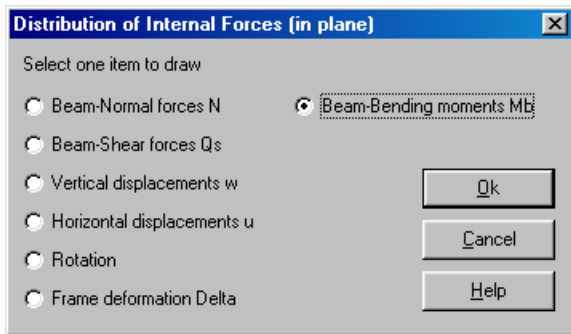- Click "OK" button

Figure 8-39   "Distribution of internal forces (in plane)" option box

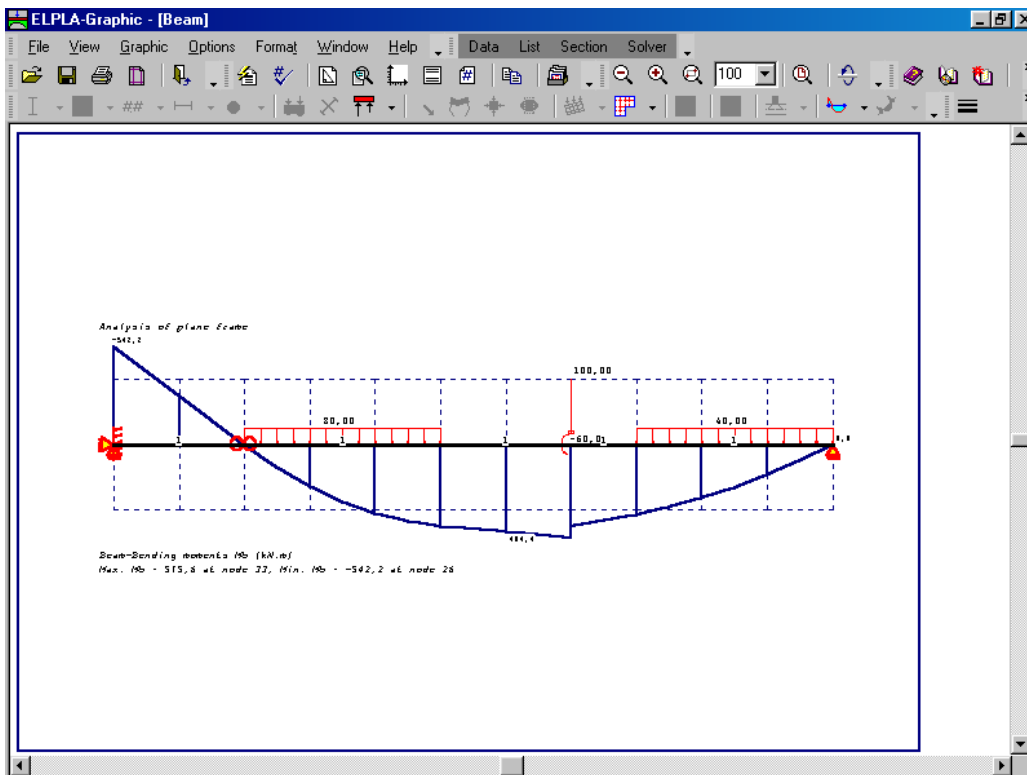The bending moments are now displayed for the beam as shown in Figure 8-40.

Figure 8-40   Beam bending moments

5   Index

A

B

C

F

G

L

N

P

S