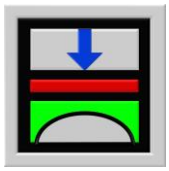
Part C

User's Guide
for the program ELPLA-Data



Determining
contact pressures, settlements, moments
and shear forces of slab foundations by the
method of finite elements

Version 9.2

Program authors: M. El Gendy
A. El Gendy

GEOTEC: GEOTEC Software Inc.
PO Box 14001 Richmond Road PO
Calgary AB, Canada T3E 7Y7

http://www.elpla.com
geotec@elpla.com

Contents Page

1 An overview of ELPLA-Data

ELPLA-Data is used to define the FE-Net, calculation methods, boundary conditions, soil properties, slab properties, project identification, boring fields, etc. for a new problem. The program may also be used to modify the data of a problem that already exists.

The input project data can be defined through ELPLA-Data in the following steps:

- Calculation methods
- Project identification
- FE-Net data
- Girders
- Spring supports
- Supports/ Boundary conditions
- Piles
- Soil properties
- Net of soil elements in z-direction
- Limit depth
- Foundation (or slab) properties
- Reinforcement
- Boring fields
- Loads
- Neighboring foundations
- Temperature change
- Additional settlements

2 Description of ELPLA-Data

ELPLA-Data is a 32-bit, graphical software product that operates under Microsoft Windows XP/ Vista/ 7/ 8. The common "what you see is what you get" of Windows applications makes it easy to learn how to use ELPLA-Data, especially if you are already familiar with the Windows environment.

The program package ELPLA consists of 7 separate programs. The programs can run independently. The name and short description of the seven separate programs are given in Table C-1.

The usage of the program is typically such that first data files are created describing a certain problem by ELPLA-Data, then the project problem is analyzed by using ELPLA-Solver. Finally, the results can be presented as graphical drawing, graphs and tables using the five separate programs ELPLA-Graphic, ELPLA-Section, ELPLA-List, ELPLA-Boring and GEOTEC-Editor.

Table C-1 Names and descriptions of the seven separate programs

Program name	Description of the program
ELPLA-Data	Editing project data
ELPLA-Solver	Analyzing the project problem
ELPLA-Graphic	Displaying data and results graphically
ELPLA-List	Listing project data and calculated results
ELPLA-Section	Displaying results graphically at specified sections
ELPLA-Boring	Editing and displaying boring logs graphically
GEOTEC-Editor	A simple text editor program

Table C-2 gives a list of files, which are read or created by ELPLA-Data. The files can be classified in three groups.

Table C-2 Names of file groups

Group	Saved from the program
A Main data files	ELPLA-Data
B Data files (*.DAT)	ELPLA-Data
C Project data files	ELPLA-Data

Further more, Table C-3 shows the filenames, contents and groups of all files that may be read or created by ELPLA-Data.

Table C-3 Names and contents of files

A Main data files

Filename	Contents
FIRMA	Firm header
STEU	Default directory for files that are saved by ELPLA
NOFORMAT	Number formats
RFT	Design code parameters
UNITS	System of units
PREFEREN.DAT	FE-Net and calculation preferences

B Data files

Filename	Contents
PLOTPAR. DAT	Plot parameters
FONT.DAT	Font data
NODISPLA.DAT	Data of display values
LINEFORM. GRA	Line formats
PAINT.GRA	Fill color data
ORDINATE.GRA	Max. ordinate data

C Project data files

Filename	Contents
* .PO1	System data (Analysis of isolated raft)
* .PO2	System data (Analysis of system of rafts)
*. AUF	Project identification
*. BAU	Soil properties
*. LDH	Data of the limit depth
*. PC1	Load data for slab and grid
*. PCF	Load data for plane frame
*. PCW	Load data for plane stress
*. PL6	Node coordinates and element connectivity
*. PL8	Slab boundary
*. GL1	Girder data (Part 1)
*. GL2	Girder data (Part 2)
*. P21	Data of slab properties/ levels/ coordinates
*. P23	Reinforcement data
*. P31	Data of supports/ boundary conditions for slab and grid
*. P61	Data of supports/ boundary conditions for plane frame
*. P71	Data of supports/ boundary conditions for plane stress
*. P35	Data of spring supports for slab and grid
*. P81	Data of spring supports for plane frame
*. P91	Data of spring supports for plane stress
*. P41	File of boring fields
*. PT1	Data of temperature change
*. PP1	File of neighboring foundations
*. PV1	Data of additional soil settlements
*. DSS	FE-Net in z-direction
*. PIL	Pile properties

The asterisk (*) matches any filename with the specified extension.

The next paragraphs describe the purpose and function of each ELPLA-Data command.

3 Starting ELPLA-Data

Start ELPLA-Data by clicking the program icon in the Windows "Start"-Menu. The introduction screen (Figure C-1) appears.

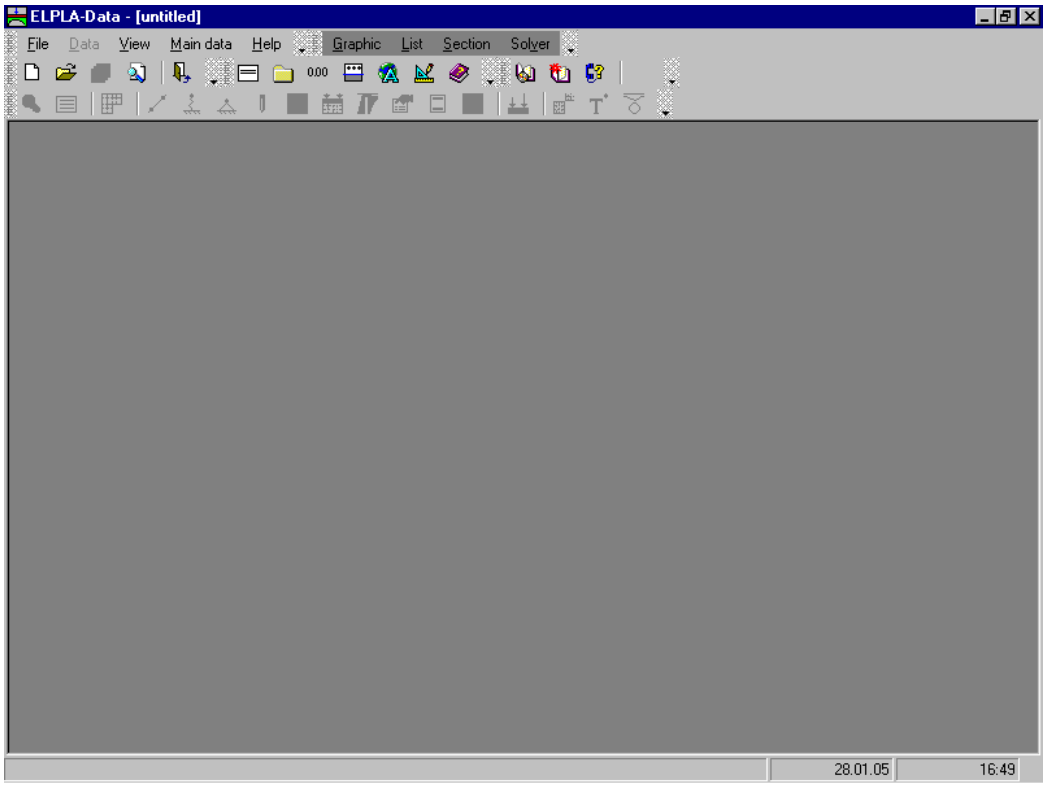


Figure C-1 Introduction screen of the program ELPLA-Data

The menu head of Figure C-1 contains the following five commands:

- File
- Data
- View
- Main data
- Help

After clicking one of the five menu commands, other sub-commands or options become available, which are presented and described in the following paragraphs 4 to 11.

4 File Menu

The File Menu commands are:

- New project
- Open project
- Save project as
- File list
- Files 1, 2, 3, 4
- Exit

4.1 File Menu –"New project" command

By the "New Project" command the current project is closed, if one is loaded, and a new project is initialized for starting a new problem definition. "New project" command has the same action as quitting ELPLA-Data and then restarting ELPLA-Data. After clicking this command, the following Wizard assistance in Figure C-2 appears. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of forms. The first form of "Calculation methods" Wizard is the "Analysis type" Form.

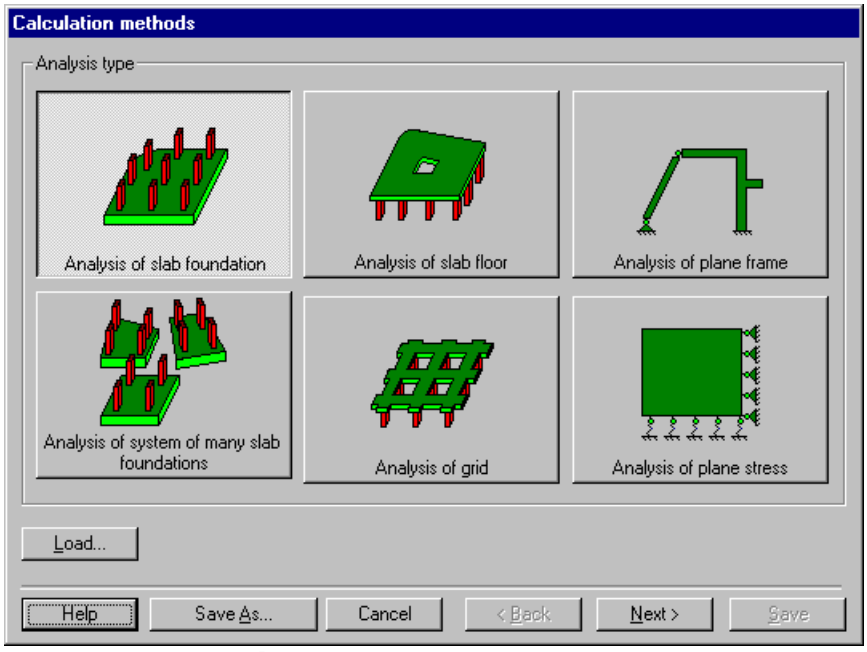


Figure C-2 "Analysis type" Form

ELPLA is used to analyze not only isolated raft but also slab floor, grid, plane frame, plane stress and system of rafts. In the "Analysis type" Dialog box (Figure C-2) choose one of the six analyses.

4.2 File Menu –"Open project" command

By the "Open project" command the current project is closed, if one is loaded, and an existing project is opened. Figure C-3 shows the "Open" Dialog box used to open a specified project. Because ELPLA is used to analyze an isolated raft and system of rafts, ELPLA can read two types of filenames. One has the extension of PO1, which represents the isolated raft and the other has the extension of PO2, which represents the system of rafts.

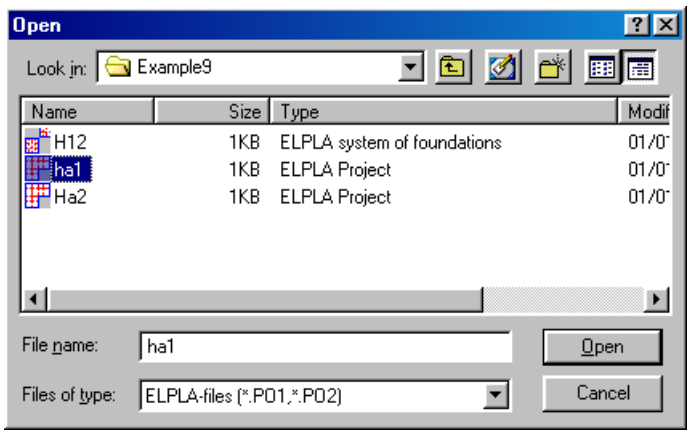


Figure C-3 "Open project" Dialog box

In case of one of the data files is not found, the following Message box appears (Figure C-4). This menu shows the project data that are not found. These data are important for the analysis.

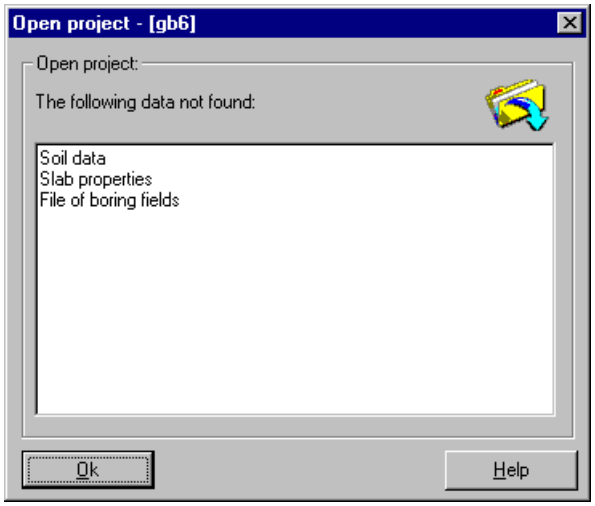


Figure C-4 "Open project" Message box

4.3 File Menu –"Save project as" command

By this command the current project is saved under a new file name. Figure C-5 shows the "Save as" Dialog box used to save the project.

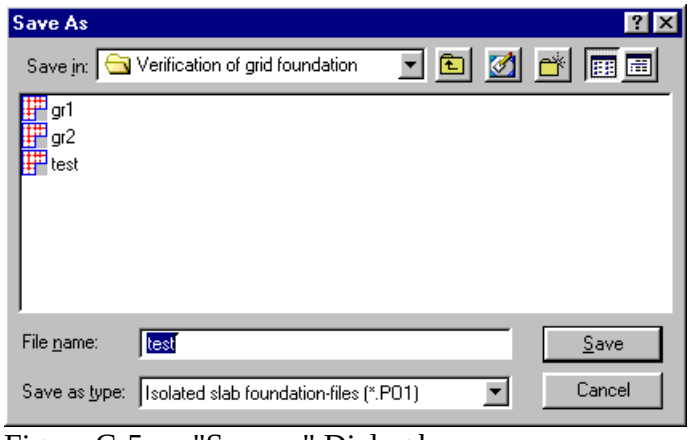


Figure C-5 "Save as" Dialog box

4.4 File Menu –"File list" command

By the "File list" command the user can delete, compress or extract projects, intermediate results, final results or print file lists of projects (Figure C-6). It is possible to sort ELPLA-files according to project identification data (file name, title, date and project).

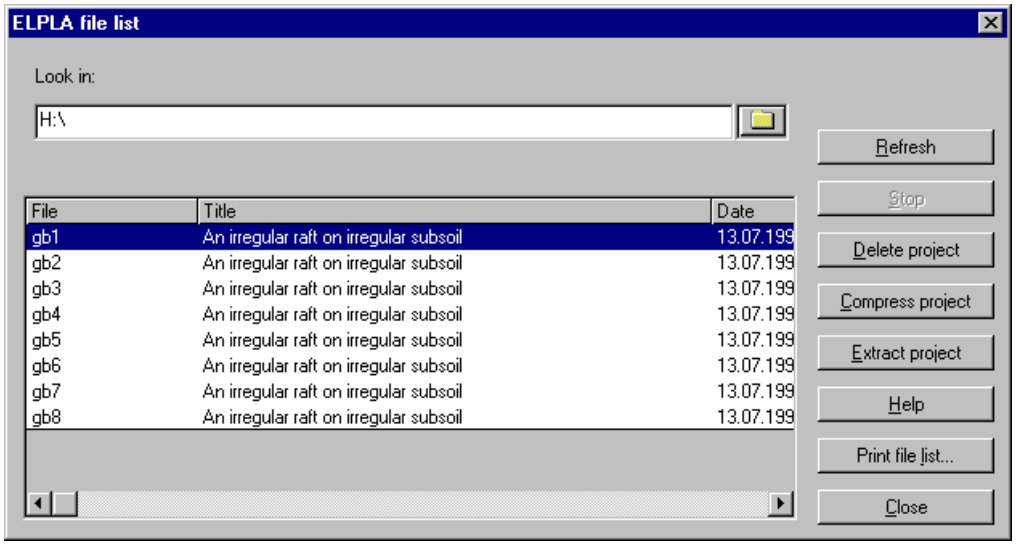


Figure C-6 "ELPLA File list" Dialog box

When the user chooses to delete a project, a Message box will appear (Figure C-7). Not only all project data can be deleted but also intermediate results or final results can be separately deleted.

Note
Deleted files by ELPLA go to the recycled pin.

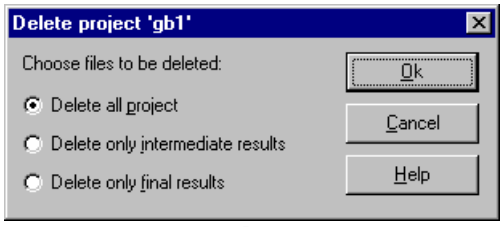


Figure C-7 "Delete project" Message box

4.5 File Menu –"Files 1, 2, 3, 4" command

By the "Files 1, 2, 3, 4" command the user can open one of the last four loaded projects.

4.6 File Menu –"Exit" command

By the "Exit" command the current project is closed and ELPLA-Data is quitted, Figure C-8.

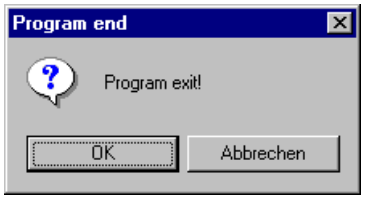


Figure C-8 "Exit" Message box

In case of one of the project data is not defined, the following Message box in Figure C-9 appears. This menu shows the project data that are not defined. These data are important for the analysis.

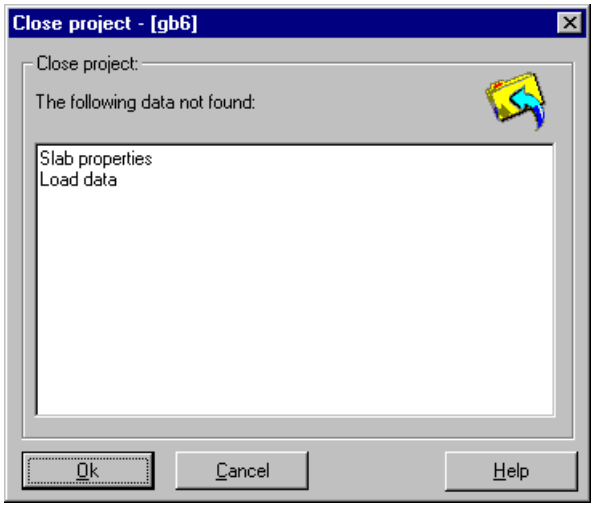


Figure C-9 "Close project" Message box

5 Data Menu

The Data Menu is the main menu, which is used to define the problem.
The Data Menu commands are:

- Calculation methods
- Project identification
- FE-Net data
- Girders
- Spring supports
- Supports/ Boundary conditions
- Piles
- Soil properties
- Net of soil elements in z-direction
- Limit depth
- Foundation (or slab) properties
- Reinforcement data
- Boring fields
- Loads
- Neighboring foundations
- Temperature change
- Additional settlements

5.1 Data Menu–"Calculation methods" command

By this command the analysis method of the project is defined. When choosing the "Calculation method" command, the "Calculation methods" Wizard in Figure C-2 appears with the "Analysis type" Form. In this Form define the analysis type of the problem from different structural systems that are available in ELPLA. After selecting the structural system, click "Next" button to go to the next Form.

Calculation methods for an isolated slab foundation

Nine different numerical calculation methods are considered for the analysis of rafts according to Figure C-10. Choose one of the methods No. 1 to 9.

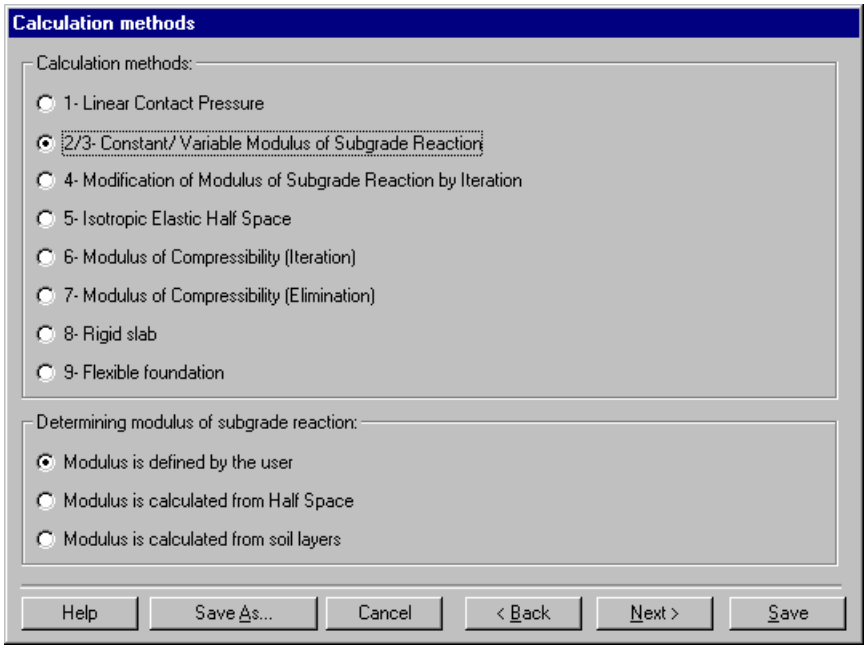


Figure C-10 "Calculation methods" Dialog box

System symmetry

The next Form is the "System symmetry" Form, Figure C-11. In this form select system symmetry and click "Next" button to go to the next Form.

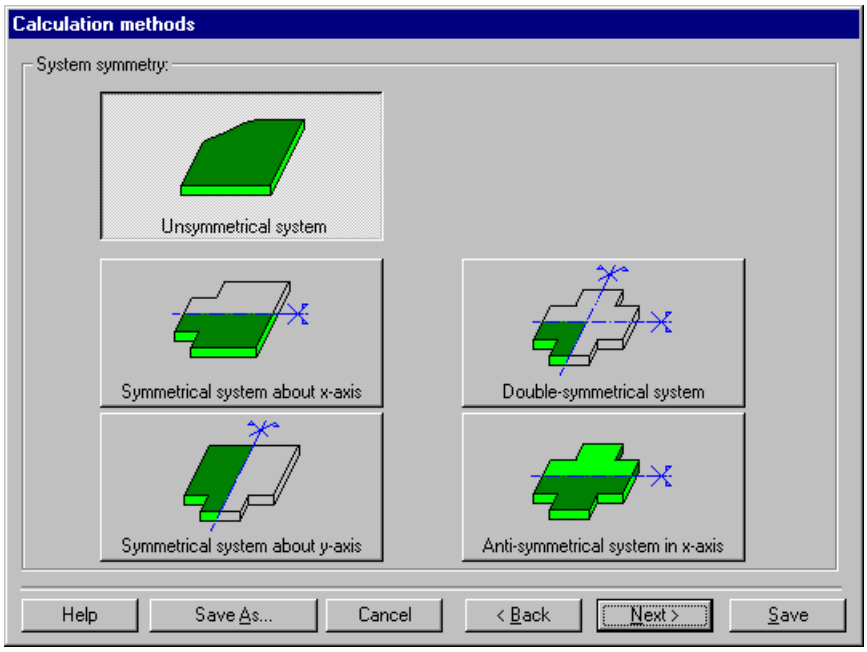


Figure C-11 "System symmetry" Form

By using the system symmetry, if the problem is symmetrical in loading, shape and soil about x- or y-axis, the computational time and computer storage can be considerably reduced.

By defining the project data for simple symmetrical or antisymmetrical slab system the data are defined according to Figure C-12, in which only the lower half slab is considered for symmetry about x-axis while only the left half slab is considered for symmetry about y-axis.

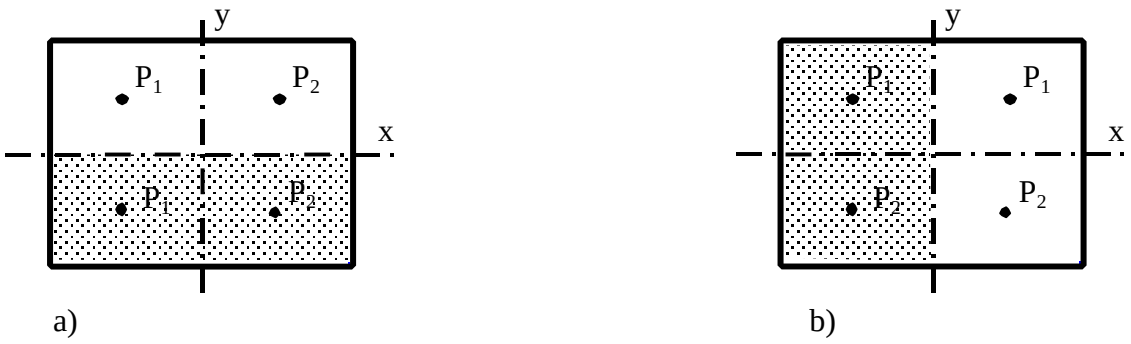


Figure C-12 Simple symmetrical slab system

By defining the project data for double symmetrical slab system the data are defined according to Figure C-13. Only the left lower quarter of the slab is considered.

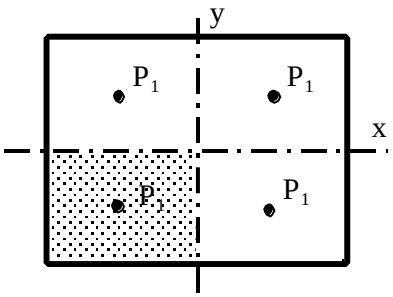


Figure C-13 Double symmetrical loaded slab

If the slab is symmetrical in shape and unsymmetrical in loading, it is also possible to divide this general case of loading into two cases having symmetrical and antisymmetrical loading, Figure C-14.

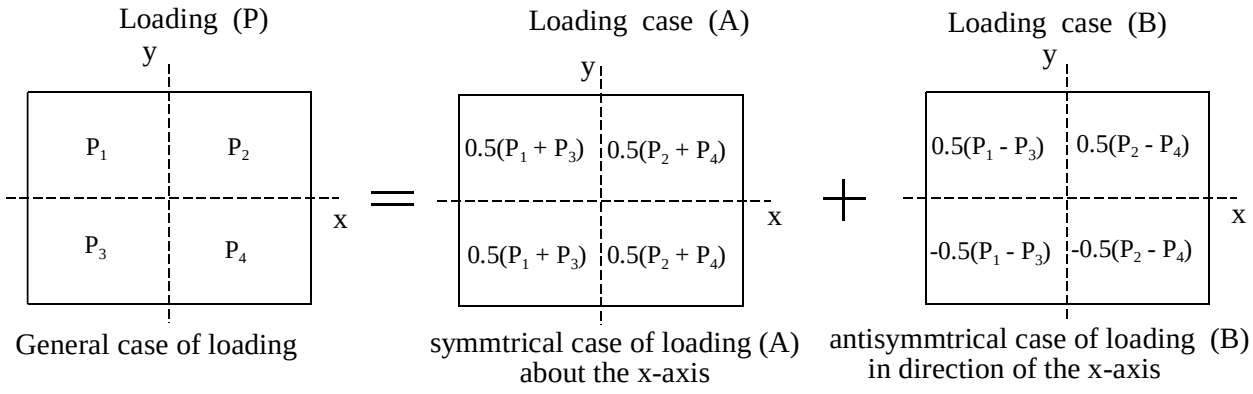


Figure C-14 General case of loading by symmetrical and antisymmetrical loading

The symmetrical cases are available for all calculation methods 1 to 9. The antisymmetrical case is only possible for calculation methods 4 to 8.

Determination of modulus of subgrade reactions
In ELPLA it is possible to analyze the raft by modulus of subgrade reaction method in which the modulus can be determined by three ways:

a) Modulus is defined by the user
b) Modulus is calculated from Half-space
c) Modulus is calculated from soil layers

In item a) the user can define a constant modulus for the entire raft (Method 2) or a variable modulus at nodes (Method 3).

In items b) and c) the modulus is calculated through the settlement calculation of the soil depending on boring logs and soil properties.

By the modulus of subgrade reaction method, specify the way for determining the modulus of subgrade reaction by selecting one of the available three options in Figure C-2.

Options
Some options are available in ELPLA such as concrete design of sections, additional springs, supports, girders and piles, determining the limit depth, nonlinear subsoil model, determining displacements, stresses and strains and in soil. Also, ELPLA can study some external influences on the raft such as temperature change, additional settlements or neighboring foundations. In the menu of Figure C-15 check the options that you want to consider in the analysis.

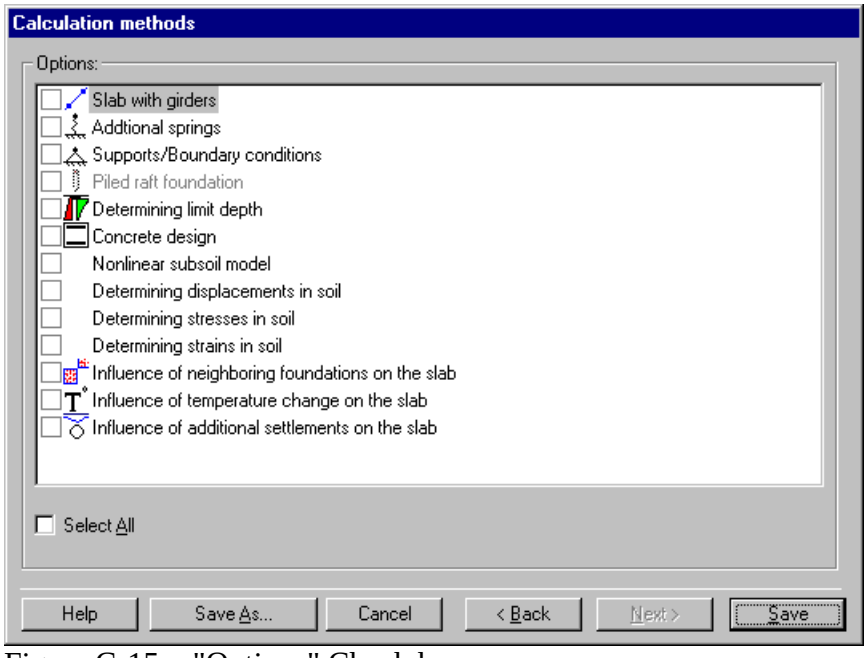


Figure C-15 "Options" Check box

Analysis of system of many slab foundations
In the "Analysis type" Form (Figure C-2), if the option "Analysis of system of many slab foundations" is chosen, the following Dialog box in Figure C-16 appears. Three different numerical calculation methods are considered for the analysis of system of slab foundations, flexible, elastic or rigid. For the analysis of system of many slab foundations the project filenames (slab foundations) are required.

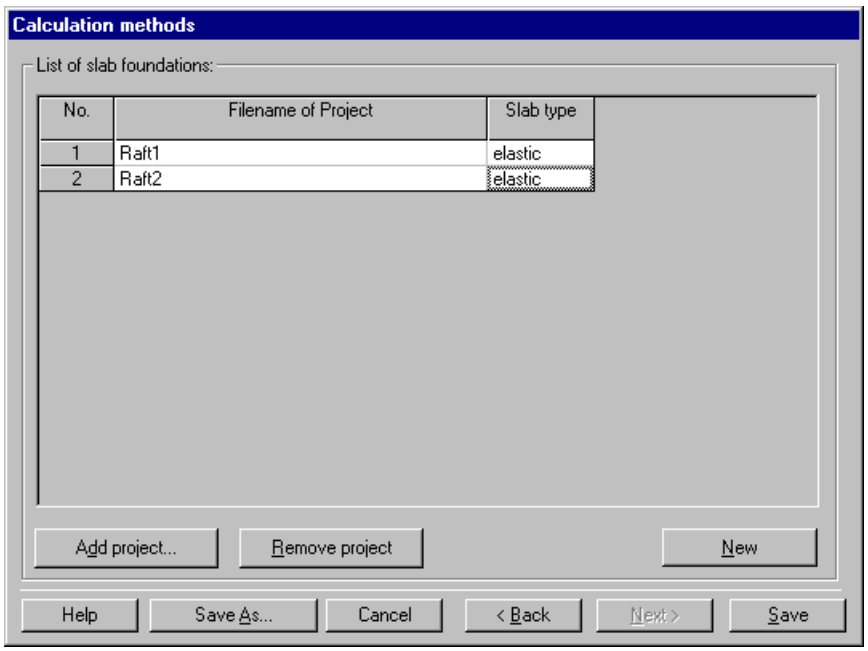


Figure C-16 "Analysis of system of many slab foundations" Dialog box

Analysis of a slab floor, plane frame or plane stress
By choosing the analysis of a slab floor, plane frame or plane stress only the "System symmetry" Form and "Options" Check box will appear.

5.2 Data Menu–"Project identification" command

By the "Project identification" command the information to identify the problem can be specified, Figure C-17. This information is required for printing and plotting the data and results. The date can be defined from the computer calendar, Figure C-18.

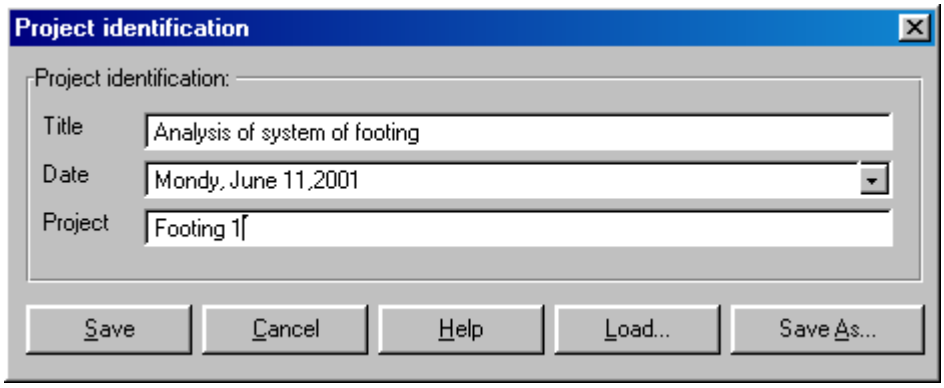


Figure C-17 "Project identification" Dialog box

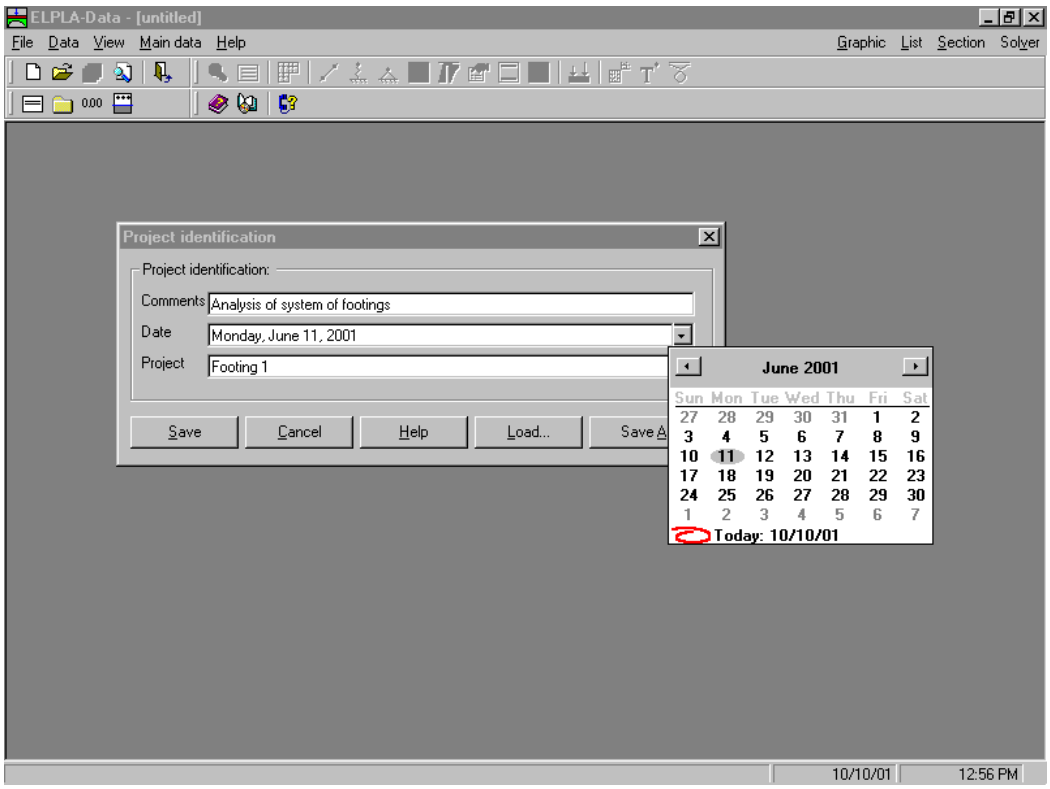


Figure C-18 Input Date information

5.3 Data Menu–"FE-Net data" command

By the "FE-Net data" command the net of the finite elements is defined. When the "FE-Net data" command is chosen, the following embedded program appears (Figure C-19).

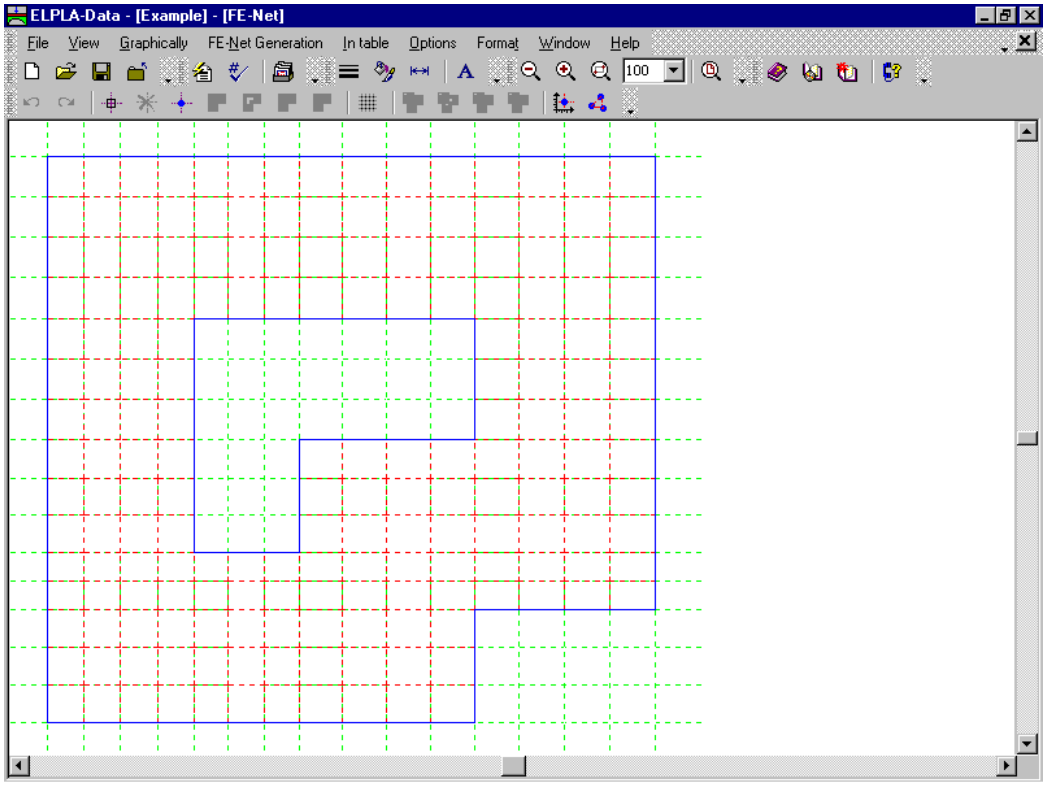


Figure C-19 "FE-Net" embedded program

The menu head of Figure C-19 contains the following nine menus:

- File
- View
- Graphically
- FE-Net Generation
- In table
- Options
- Format
- Window
- Help

After clicking one of the nine menus other sub-menus or commands become available. The following paragraph presents and describes the nine menu commands and their sub-commands.

5.3.1 File Menu

This menu contains the following commands:

- New FE-Net
- Open FE-Net
- Save FE-Net
- Save FE-Net as
- Close FE-Net

File Menu–"New FE-Net" command
A new FE-Net is created. When choosing this command, the following templates for different types of FE-Net appear (Figure C-20).

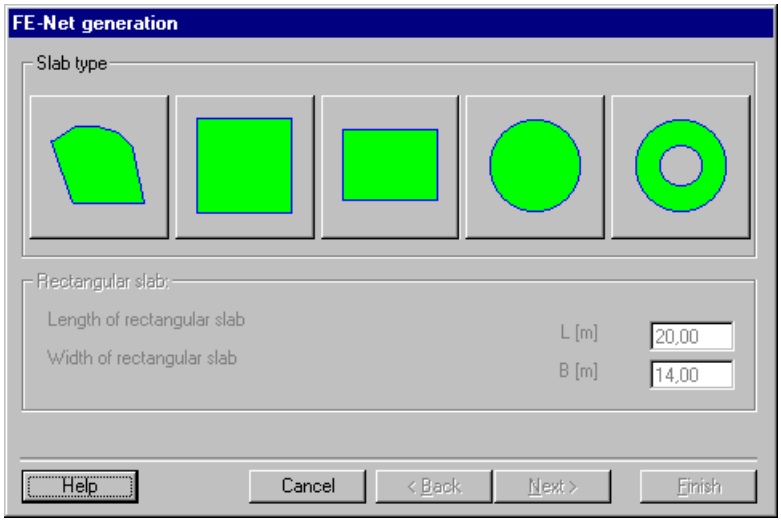


Figure C-20 Templates for different types of FE-Net

File Menu–"Open FE-Net" command
Loads are saved as FE-Net again. Then the loaded FE-Net, if desired, can be redefined.

File Menu–"Save FE-Net" command
The command saves the active FE-Net under the available name.

File Menu–"Save FE-Net as" command
The active FE-Net is saved under a new name.

File Menu–"Close FE-Net" command
Closes the FE-Net embedded program and returns to ELPLA-Data.

5.3.2 View Menu

The View Menu command is Tool bars.

View Menu–"Tool bars" command
The "Tool bars" command displays tool bars located just below the menu head. Tool bars contain icons of program menus.

5.3.3 Graphically Menu

This menu contains the following commands:

- Undo
- Redo
- Select nodes
- Remove nodes
- Add nodes
- Slab corners by Mouse
- Add opening
- Add reference points
- Add reference lines
- Cartesian grid

Graphically Menu–"Undo" command
This command is used to undo the effects of a selected command and return to a previous state.

Graphically Menu–"Redo" command
This command is used to redo the last action of "Undo" command.

Graphically Menu–"Select nodes" command
The function of "Select nodes" command is to provide a method for deleting nodes. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair. The commands "Remove nodes" in the menu "Graphically" will be enabled, indicating the mode in which is being operated. The desired nodes are selected by clicking on each node individually or selecting a group of nodes as shown in Figure C-21. A group of nodes can be selected by holding the left mouse button down at the corner of the region and dragging the mouse until a rectangle encompasses the desired group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

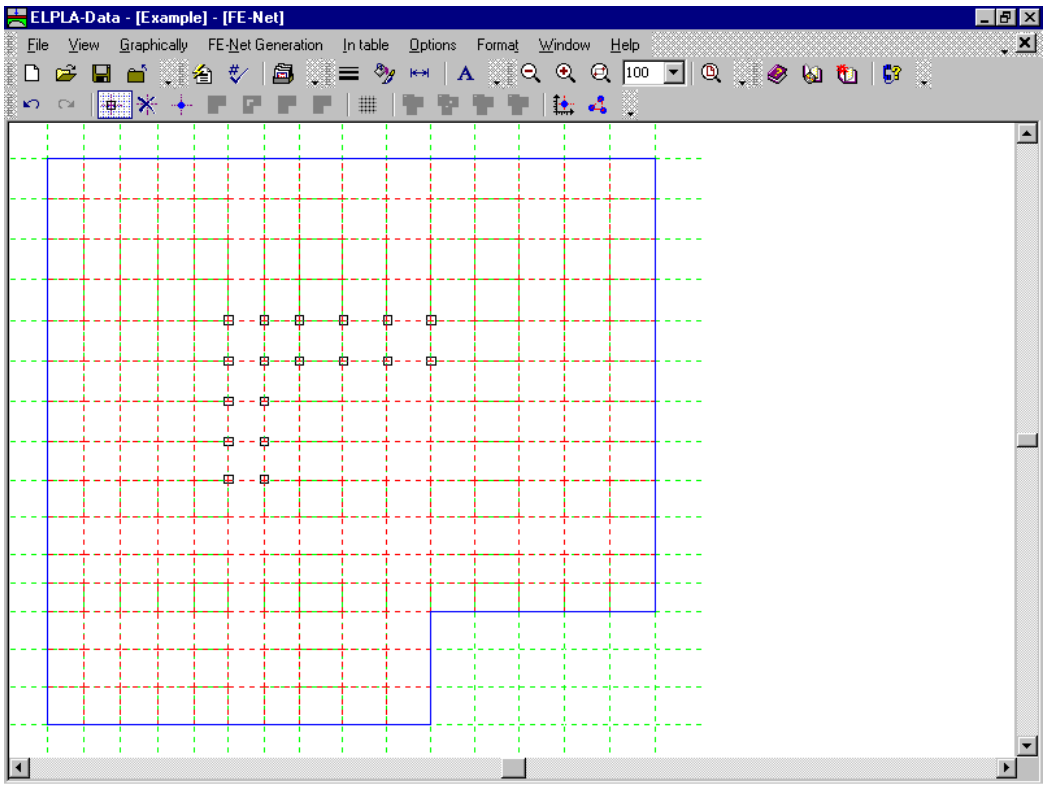


Figure C-21 Select nodes to be removed

Graphically Menu–"Remove nodes" command

"Remove nodes" command is used to remove the selected nodes. Any elements bonded the deleted nodes are also deleted (Figure C-22).

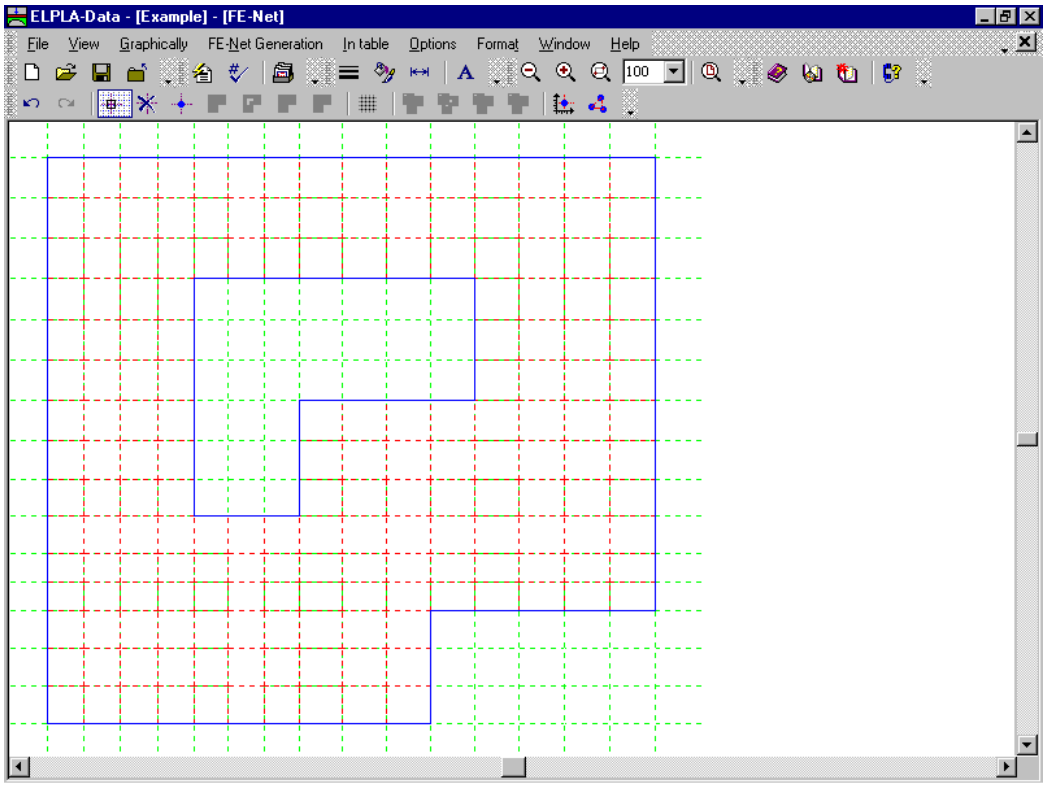


Figure C-22 Selected nodes were removed

Graphically Menu–"Add nodes" command

By the "Add nodes" command nodes can be added at anywhere to the FE-Net. Clicking by Mouse inside the required element adds a new node in the element (Figure C-23).

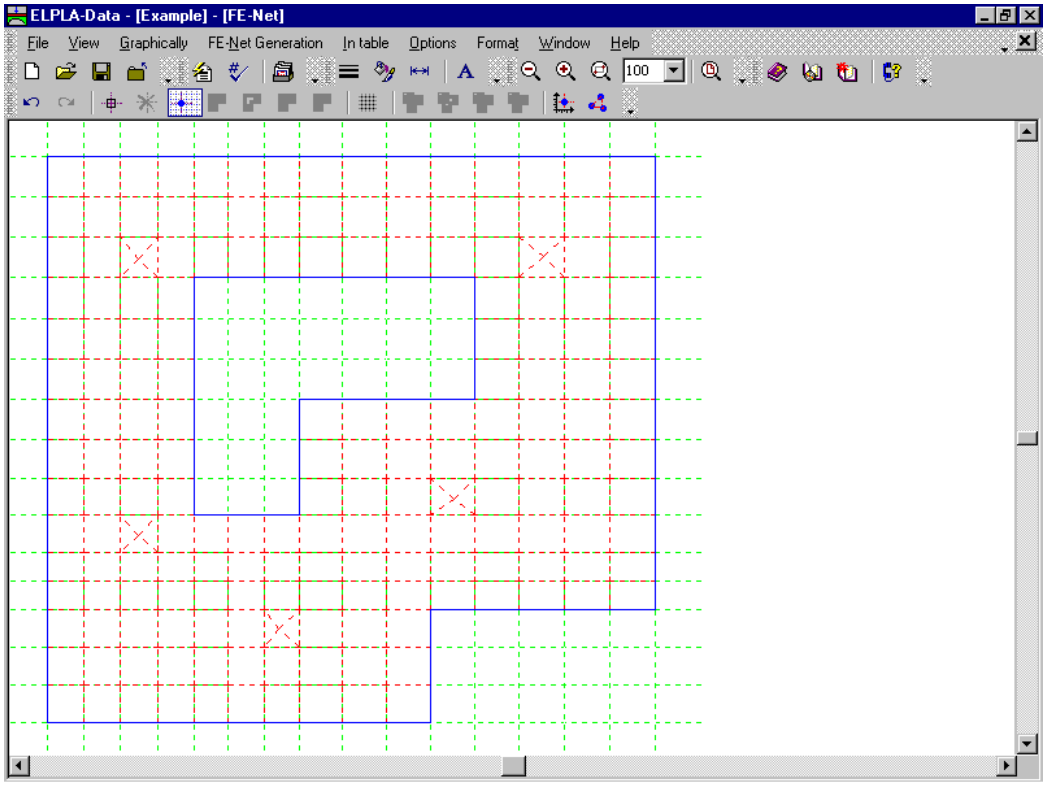


Figure C-23 Selected nodes were added

Graphically Menu–"Slab corners by Mouse" command

In ELPLA there are two alternative possibilities to define slab corners either by Mouse or by editing in a table. "Slab Corners by Mouse" Command is used to define corners of irregular slab by Mouse. The coordinates of corner positions (x, y) are related to the left bottom corner of the corresponding slab. Corners are set as a polyline, which is a connected sequence of lines or arc segments created as a single object.

To draw a slab polyline containing a line and arc combination by Mouse (Figure C-27):

- Choose the "Slab Corners by Mouse" Command from the "Graphically" Menu. The cursor will change from an arrow to a cross hair

- Click the left mouse button on the position of the start point of the first line segment and drag the mouse until the position of the end point of this segment. Then click on the end point position to consider the first line segment (Figure C-24)

- Press "A" key to switch to arc segment mode. The first point of the arc is the endpoint of the previous segment. As you drag the mouse, ELPLA draws an arc connected to the previous line segment and extended to the mouse position, Figure C-25. For drawing the arc segment the rotation direction and the radius of the arc segment is required to define. There are four cases concerning the rotation direction and the radius as shown in Figure C-26. Two of these cases are related to the position of the arc center relative to the line connected the arc points (Figure C-26a). The other two cases are related to the direction of the arc rotation starting from the first point of the arc to the second point, Figure C--26b. The arc can be drawn clockwise or anticlockwise

- Press "R" key to switch between the two possibilities of the arc center position

- Press "C" key to switch between the arc clockwise and anticlockwise directions

- Click the left mouse button on the position of the end point of the arc to consider the arc segment

- Press "A" key again to switch to line segment mode

- Repeat the previous steps for drawing arc or line segment whenever is applicable until finishing the polyline

- Double click the left mouse button on the position of the endpoint of the last segment to finish the "Slab Corners by Mouse", Figure C-27

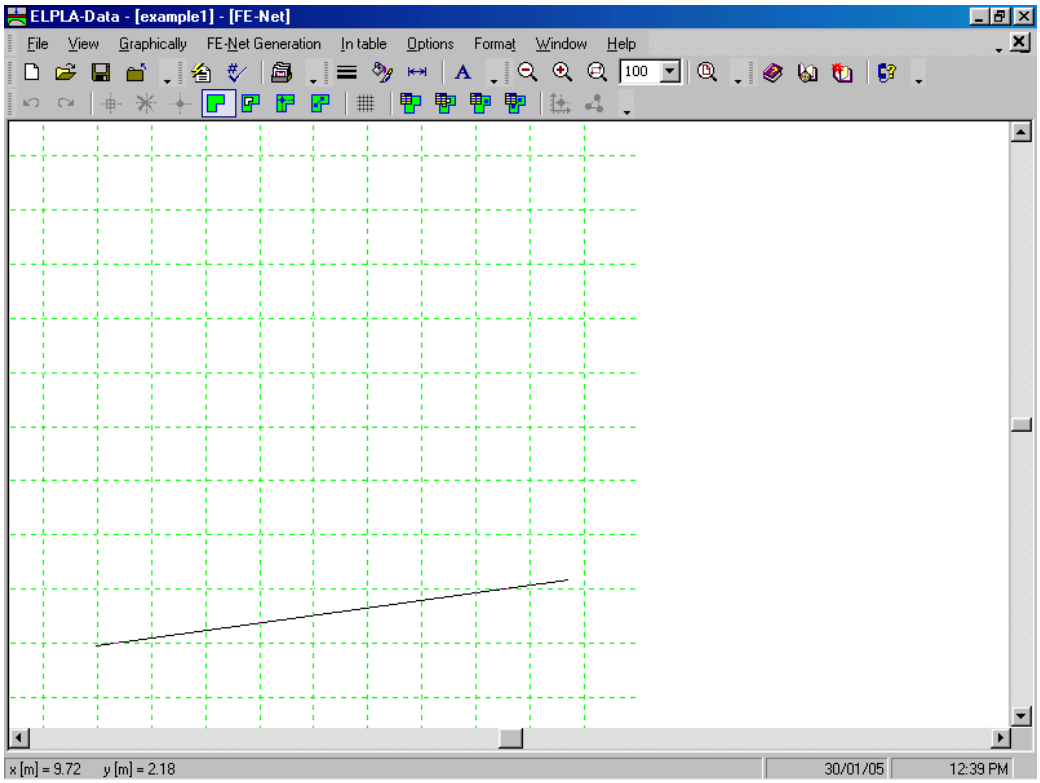


Figure C-24 Drawing the first line segment

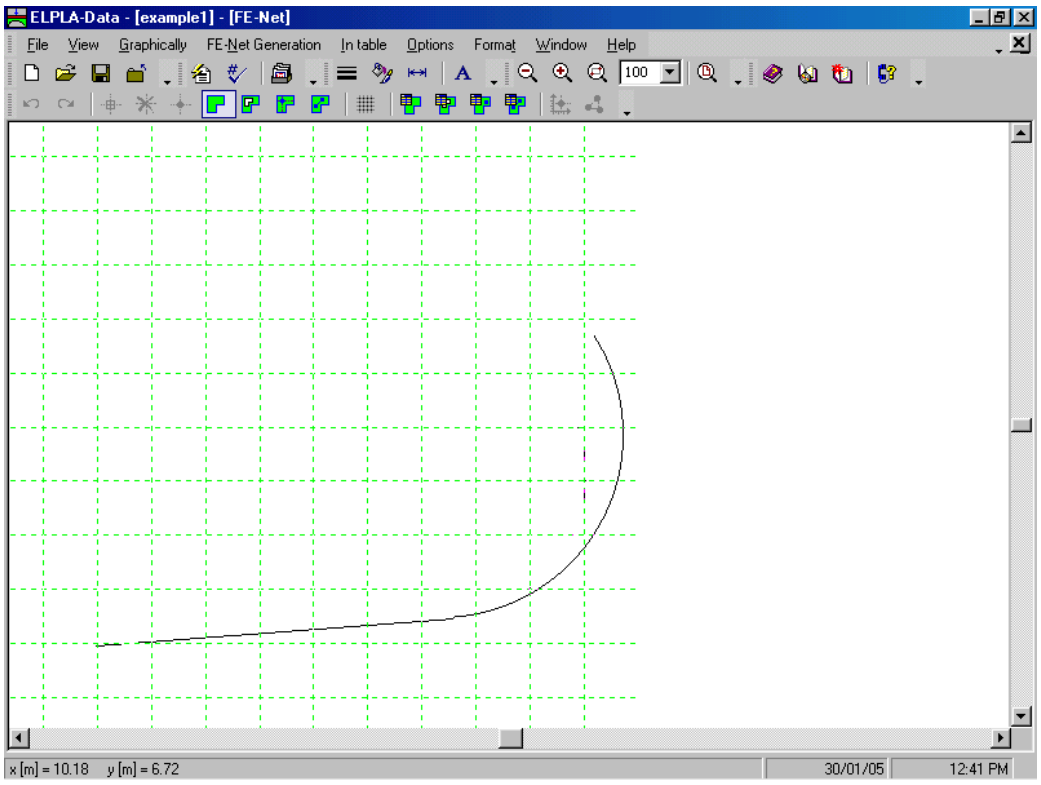
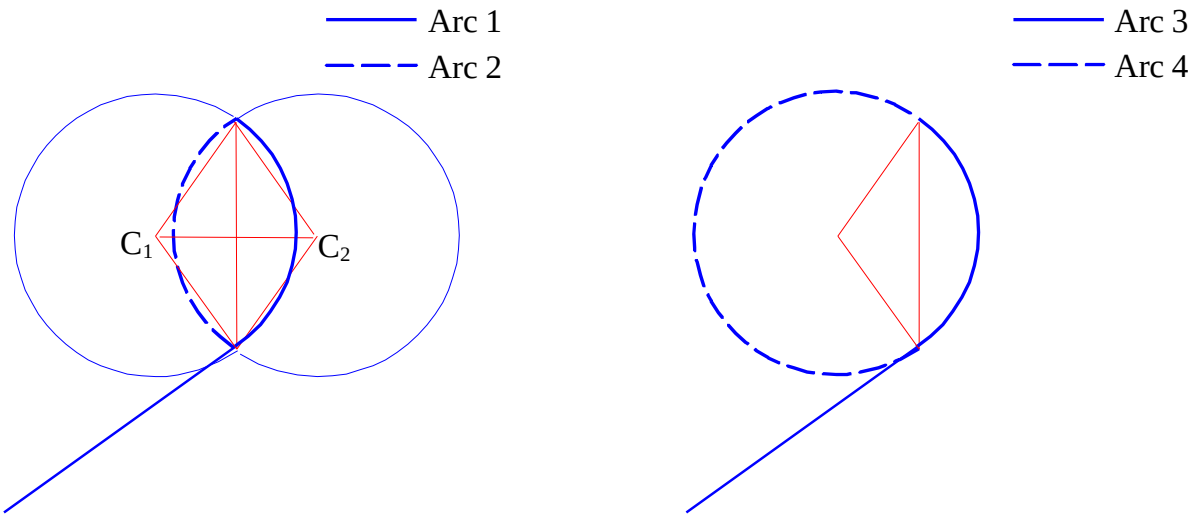


Figure C-25 Drawing the arc segment



a) Two center positions b) Two rotation directions

Figure C-26 Four cases for drawing arc segments

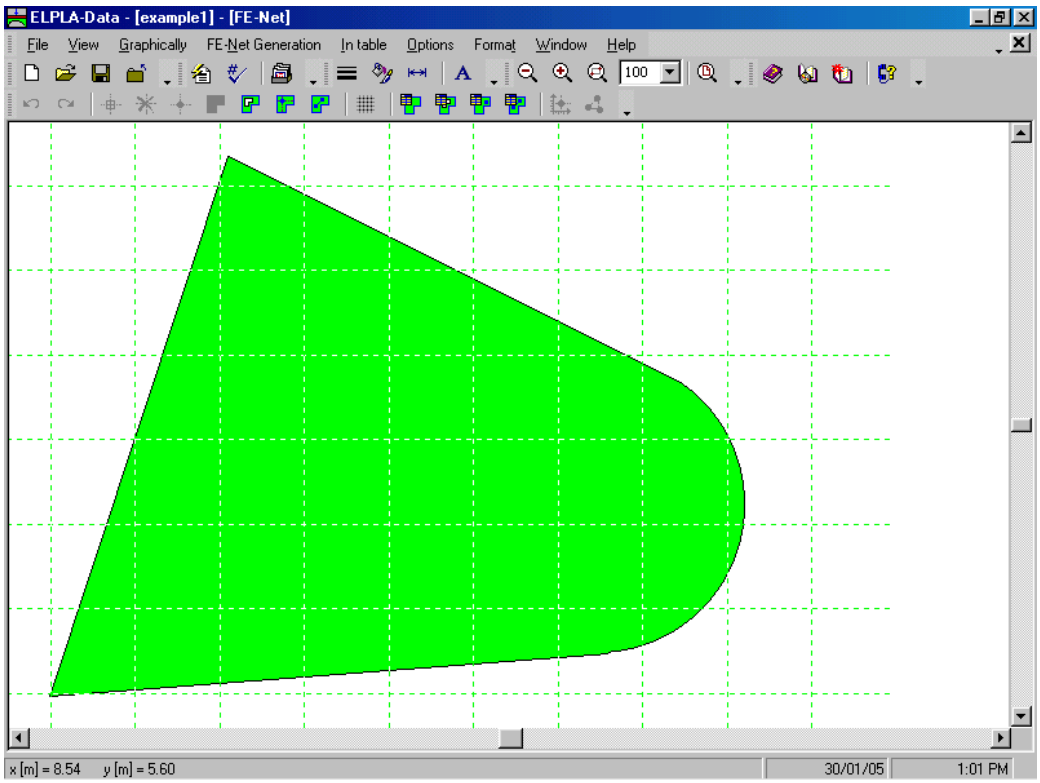


Figure C-27 Slab after defining corner points

Graphically Menu–"Add opening" command

To define an opening in the slab (Figure C-28), the position (x, y) of the opening corners must be entered by Mouse. The coordinates of corner positions (x, y) are related to the left bottom corner of the corresponding mesh. To draw an opening with arc segments, consider the same steps in drawing slab corners with arc segments.

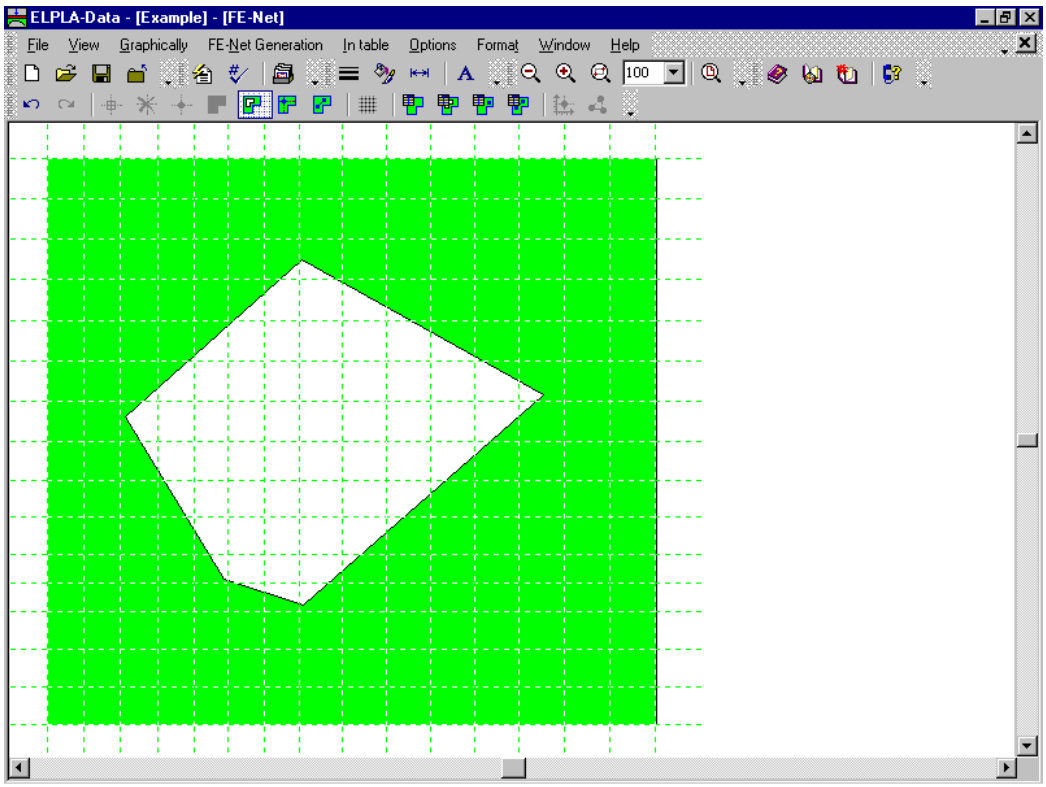


Figure C-28 Defining opening corners in the slab by Mouse

Graphically Menu–"Add reference points" command

By this option it is possible to define reference points on the slab by Mouse, Figure C-29. Reference points are used to define the positions of boundary points such as supports, springs, piles on the slab. Each time that the user generates the mesh, nodes of the FE-Net are passed automatically through these references. This provides the flexibility to make changes in the finite element mesh without having to redefine the positions of the boundary points.

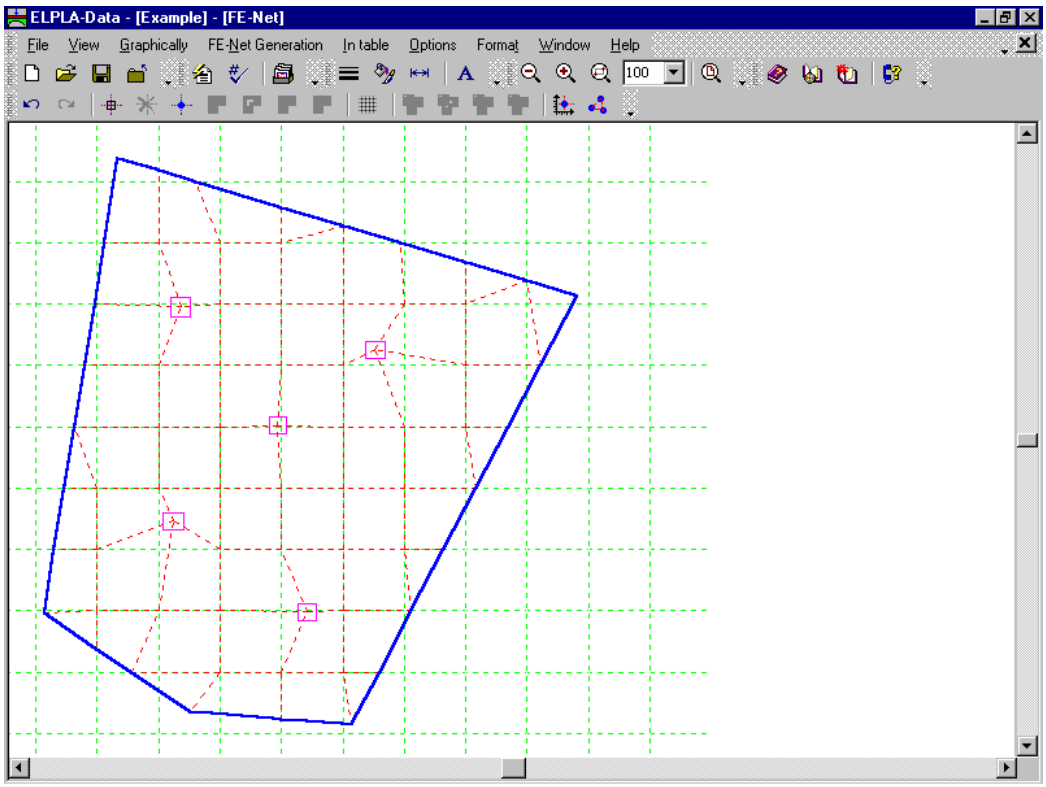


Figure C-29 Reference points on the FE-Net

Graphically Menu–"Add reference lines" command

By the option "Add reference lines" it is possible to define reference lines on the slab by Mouse, Figure C-30. Reference lines are used to define the positions of boundary lines such as girders on the slab. Each time the user generates the mesh, nodes of the FE-Net are passed automatically through these references. This provides the flexibility to make changes in the finite element mesh without having to redefine the positions of the boundary lines.

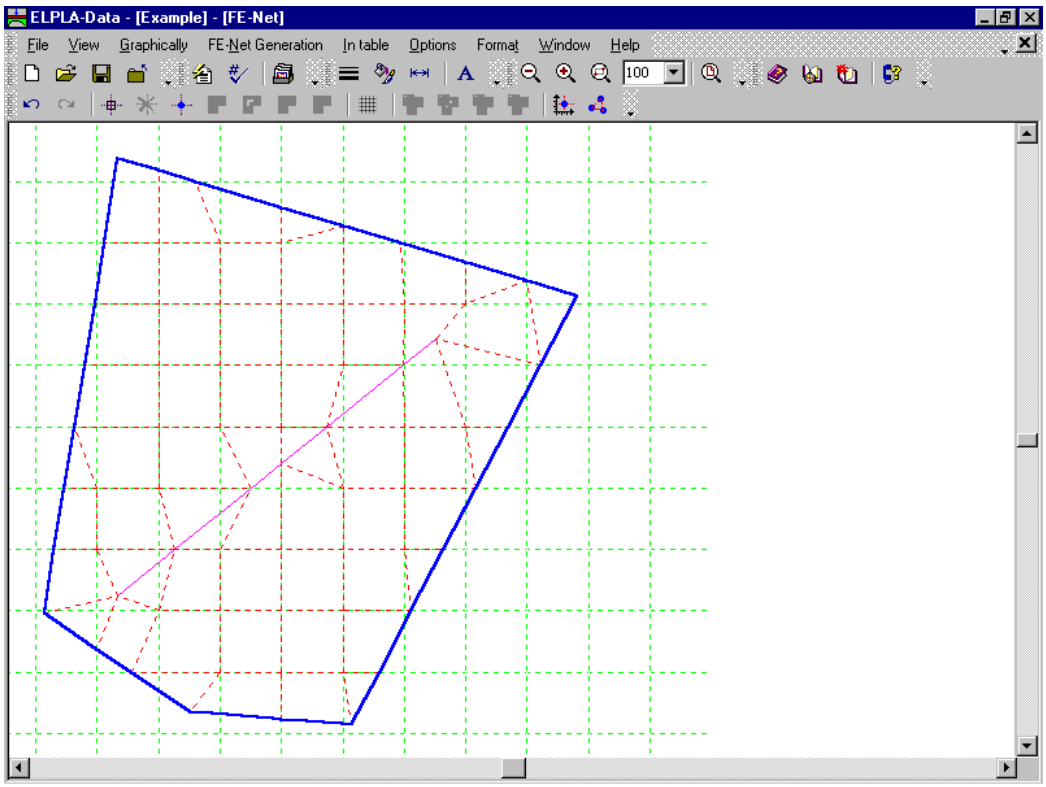


Figure C-30 Reference lines on the FE-Net

Graphically Menu–"Cartesian grid" command

The Dialog box in Figure C-31 appears to define a Cartesian grid, which is a background grid of points to assist in drawing the finite element mesh. These points can be 'Snapped to' when creating the mesh geometry in order to create nodes and elements with exact coordinates.

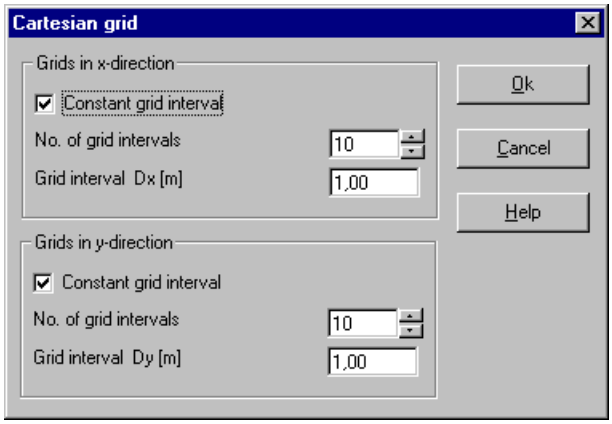


Figure C-31 "Cartesian grid" Dialog box

5.3.4 FE–Net Generation Menu

This menu contains the following commands:

- Generation type
- New generation
- Generating FE-Net
- Smoothing the mesh
- Directing border elements
- Refining the mesh

FE–Net Generation Menu–"Generation type" command
By the option "Generation type" the following Menu in Figure C-32 appears to define the type of FE-Net. There are 6 different element types to generate the FE-Net of the slab. The generation is carried out according to Grid-based approach for both triangular and rectangular elements and according to Delaunay's triangular generation for triangular elements.

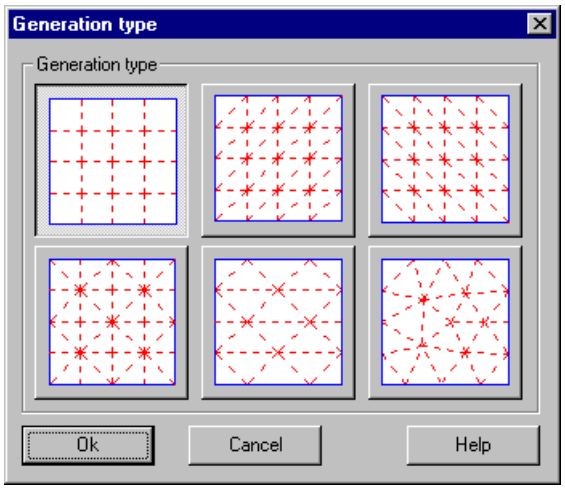


Figure C-32 Generation type for square, rectangular and irregular slabs

When generating circular and ring slabs for the first time, the Menu of Figure C-33 appears with 8 different types of nets.

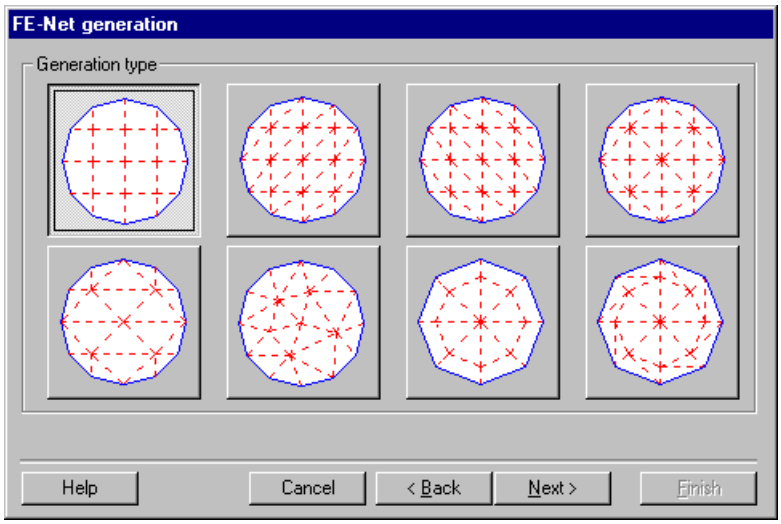


Figure C-33 Generation type for circular and ring slabs

FE–Net Generation Menu–"New generation" command
When FE-Net is defined, a new generation with new dimensions of elements or even slab can be carried out using "New generation" command.

FE–Net Generation Menu–"Generating FE-Net" command
After defining the slab corners, element dimensions and type of FE-Net the generation of the FE-Net can be created automatically by choosing the option "Generating FE-Net".

FE–Net Generation Menu–"Smoothing the mesh" command
By the option "Smoothing the mesh" it is possible to optimize the dimension of FE-Net by making all elements having nearly the same area as possible as.

FE–Net Generation Menu–"Directing border elements" command
It is possible to direct and arrange all elements on the slab borders (Figure C-34). This option is useful to present contact pressures at raft edges in good manner when analyzing the raft by Continuum model, in which the contact pressure at raft edges are higher than that at the middle.

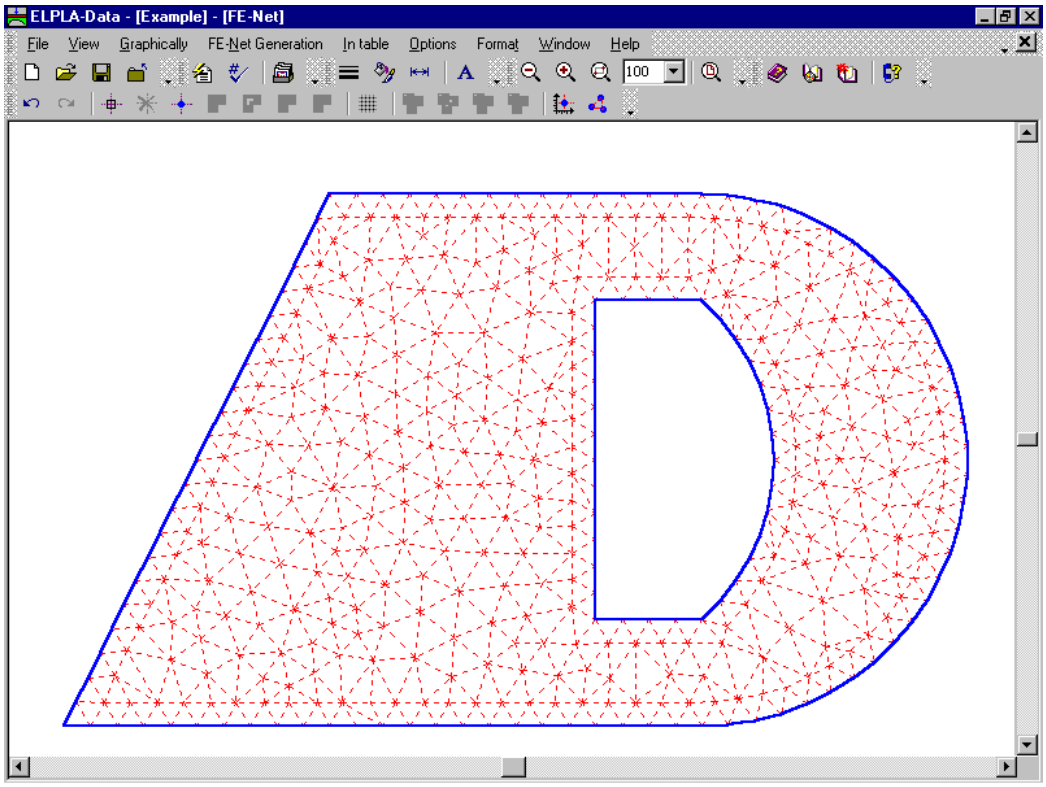


Figure C-34 Directing border elements

FE–Net Generation Menu–"Refining the mesh" command
It is possible to refine the mesh in a specified region such as around supports, springs and piles to present the concentration of stress, moment and settlement in this region (Figure C-35).

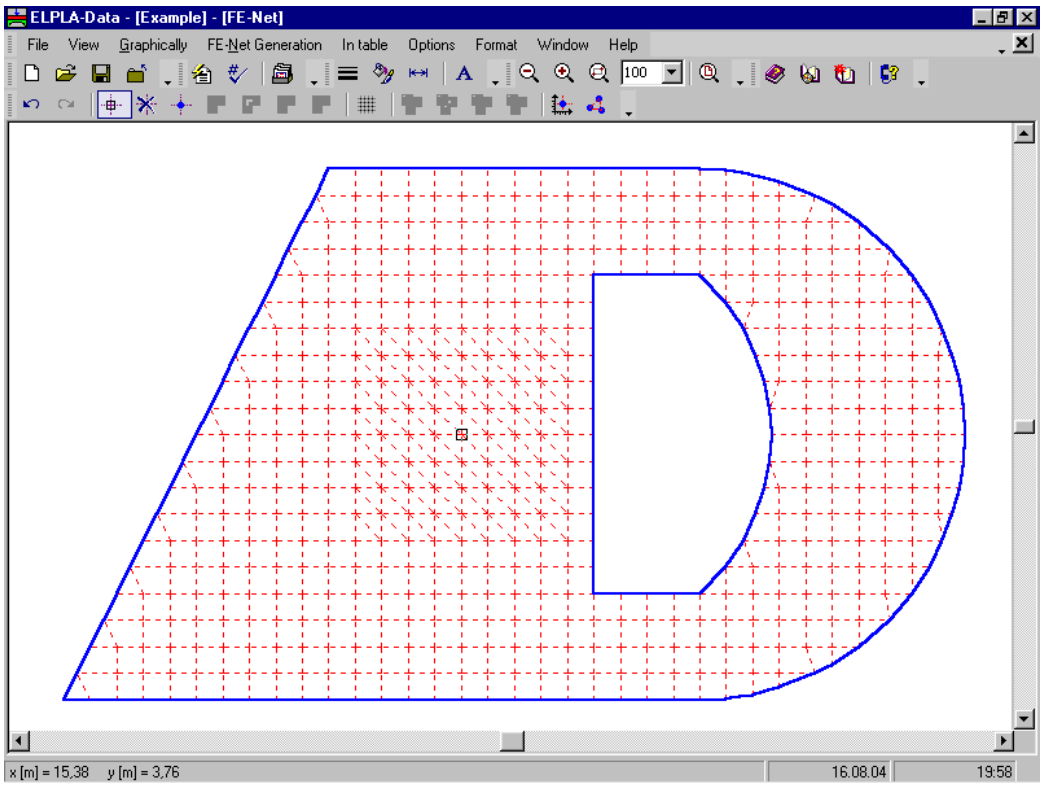


Figure C-35 Refining the mesh around a specified node

5.3.5 In table Menu

This menu contains the following commands:

- Slab corners
- Opening corners
- Reference points
- Reference lines
- Node coordinates
- Connectivity nodes

In table Menu –"Slab corners" command

By the option "Slab corners" the position (x, y) of the slab corners can be defined (Figure C-36). The coordinates of corner positions (x, y) are related to the left bottom corner of the corresponding slab.

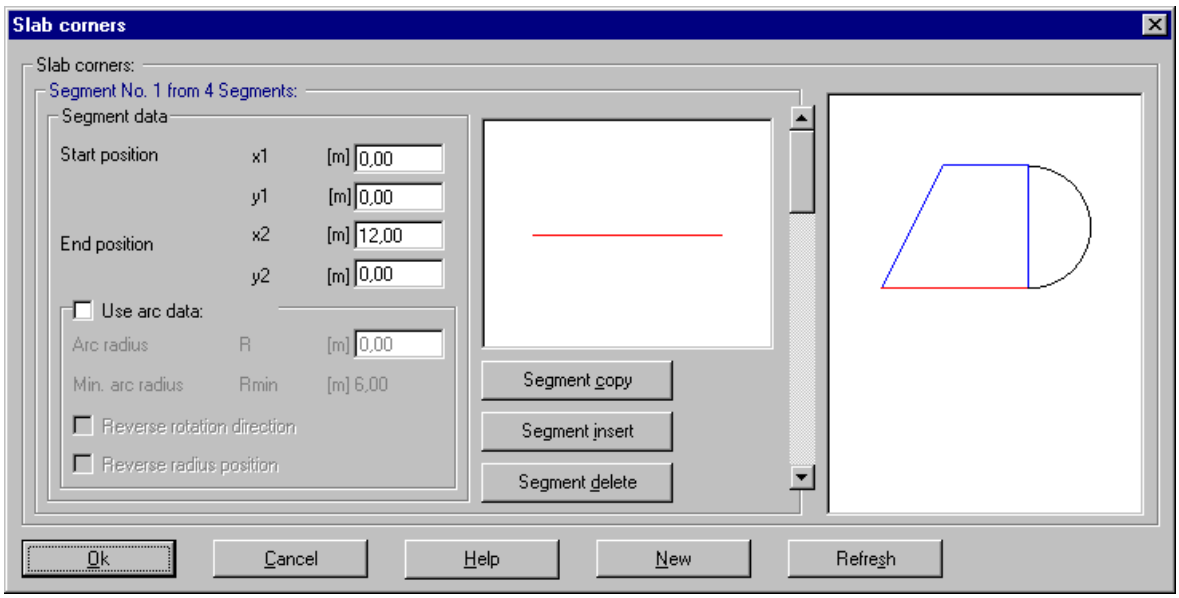


Figure C-36 Defining slab corners in a table

In table Menu–"Opening corners" command

Openings in the slab can be defined, Figure C-37. To define an opening in the slab, the position (x, y) of the opening corners must be entered in the Menu of Figure C-37. The coordinates of corner positions (x, y) are related to the left bottom corner of the corresponding mesh.

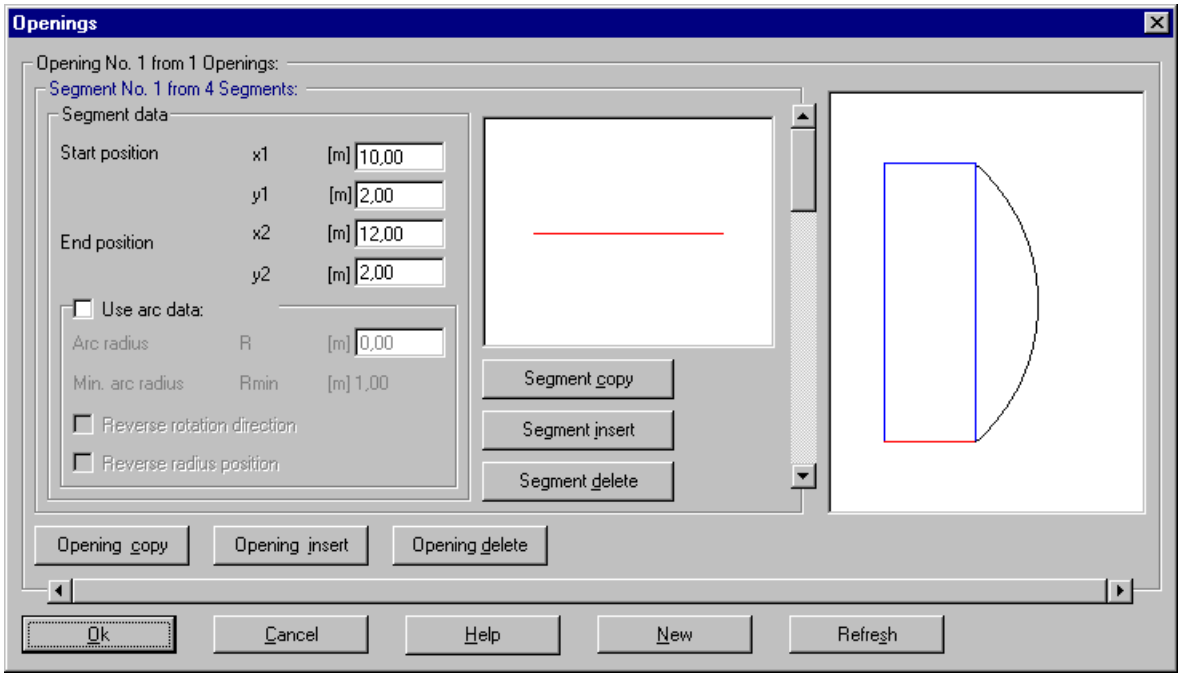


Figure C-37 Defining opening corners in a table

In table Menu–"Reference points" command

By this option it is possible to define reference points on the slab in Table, Figure C-38. Reference points are used to define the positions of boundary points such as supports, springs, piles on the slab. Each time the user generates the mesh, nodes of the FE-Net are passed automatically through these references. This provides the flexibility to make changes in the finite element mesh without having to redefine the positions of the boundary points.

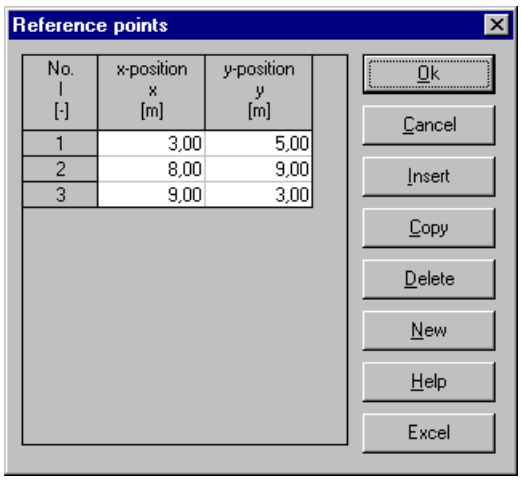


Figure C-38 Defining reference points in Table

In table Menu–"Reference lines" command

By the option "Reference lines" it is possible to define reference lines on the slab in Table, Figure C-39. Reference lines are used to define the positions of boundary lines such as girders on the slab. Each time that the user generates the mesh, nodes of the FE-Net are passed automatically through these references. This provides the flexibility to make changes in the finite element mesh without having to redefine the positions of the boundary lines.

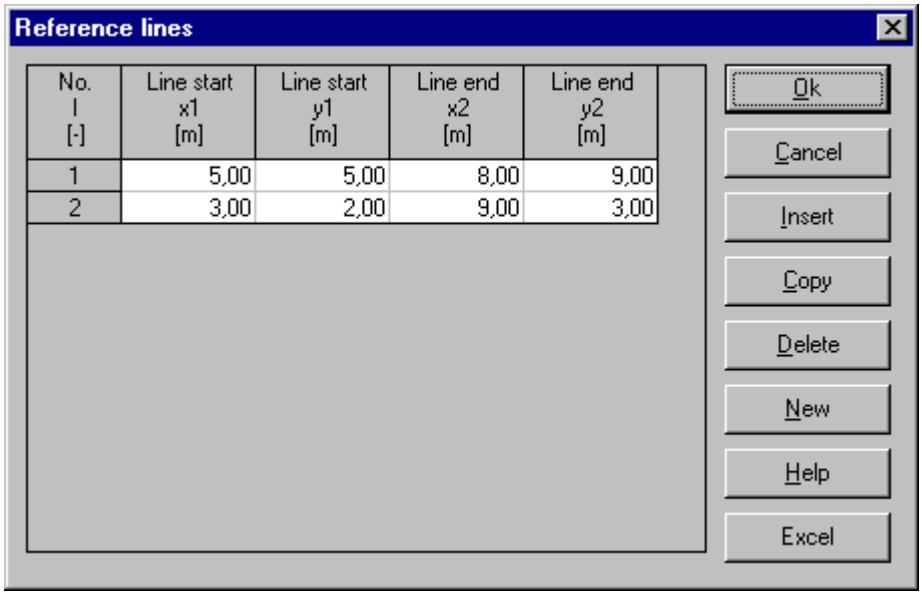


Figure C-39 Defining reference lines in a table

In table Menu–"Node coordinates" command

After choosing the option "Node coordinates" the following Table in Figure C-40 appears to define node coordinates of FE-Net.

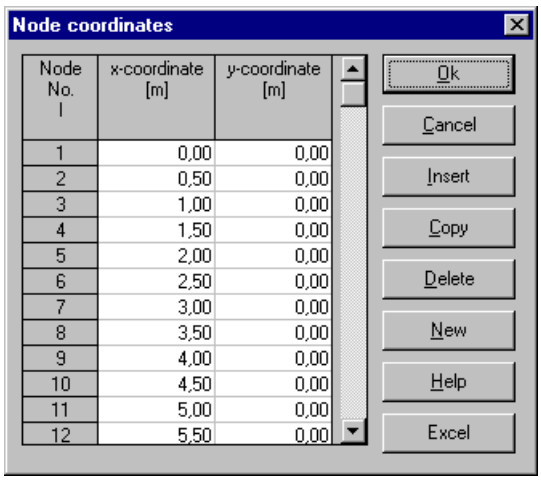


Figure C-40 Table "Node coordinate"

In table Menu–"Connectivity nodes" command

After choosing this option the following Table in Figure C-41 appears to define connectivity nodes of the elements.

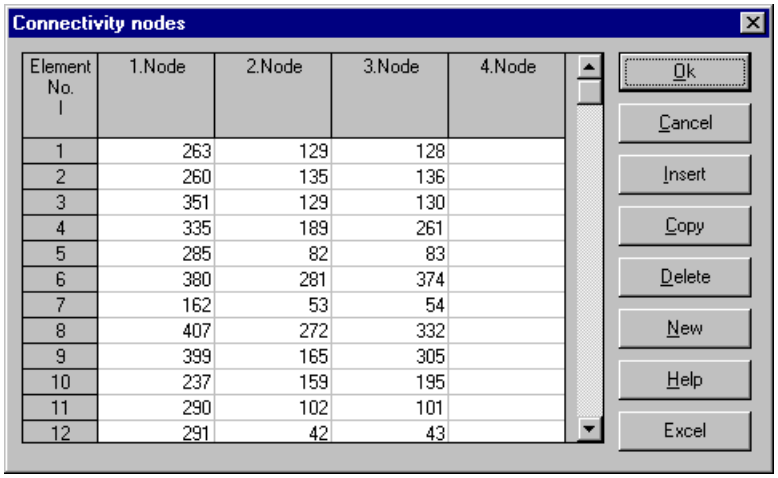


Figure C-41 Table "Net of finite elements"

5.3.6 Options Menu

The Options Menu has the following commands:

- Plot parameters
- Display values
- View grouping

Options Menu–"Plot parameters" command

Plot parameters may be set as default values by the program, or may be fully specified by the user. By this command the following plot parameters can be specified, Figure C-42:

- Display node numbering
- Display coordinates x, y
- Display element numbering
- Display Column types
- Display FE-Net in separated elements

- Color girders
- Draw girder thickness
- Display Cartesian grid
- Grid over entire area
- Color element groups and slab thickness
- Display boring subareas boundary
- Color boring subareas
- Mark boring subareas
- Snap to grid or node

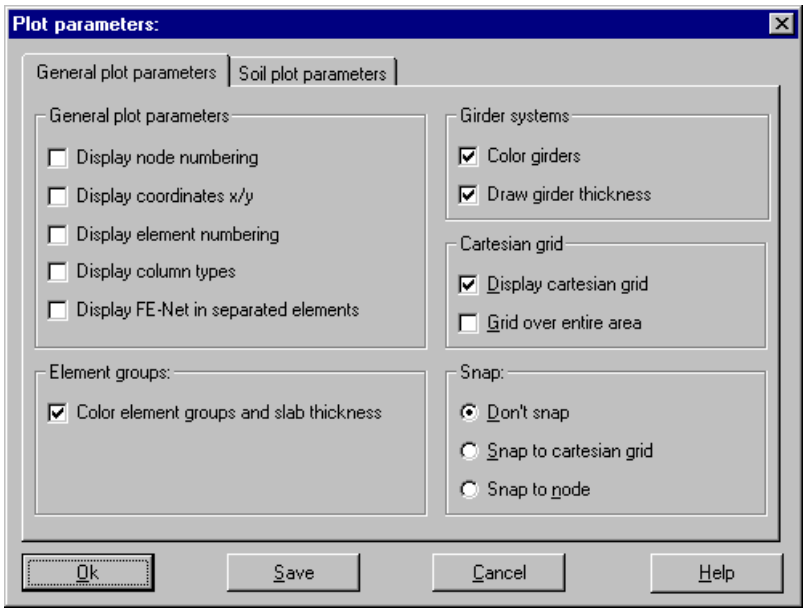


Figure C-42 "Plot parameters" Dialog box

Display Item
Turns on and off the display of the item on the drawing

Snap to grid
This command turns on and off the capability to snap to the grid when creating the mesh geometry.

Options Menu–"Display values" command
The values of the following items can be displayed, if desired, on the drawing (Figure C-43):

- Loads
- Boundary conditions
- Spring supports
- Element groups
- Girder system
- Piles
- Additional settlements

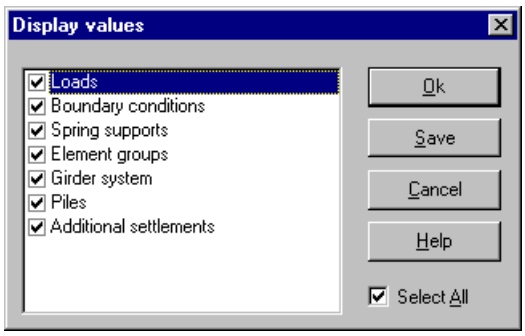


Figure C-43 "Display values" Dialog box

Options Menu–"View grouping" command

By the "View grouping" command you can draw a group of data together in one presentation (for example supports with loads and girders), Figure C-44.

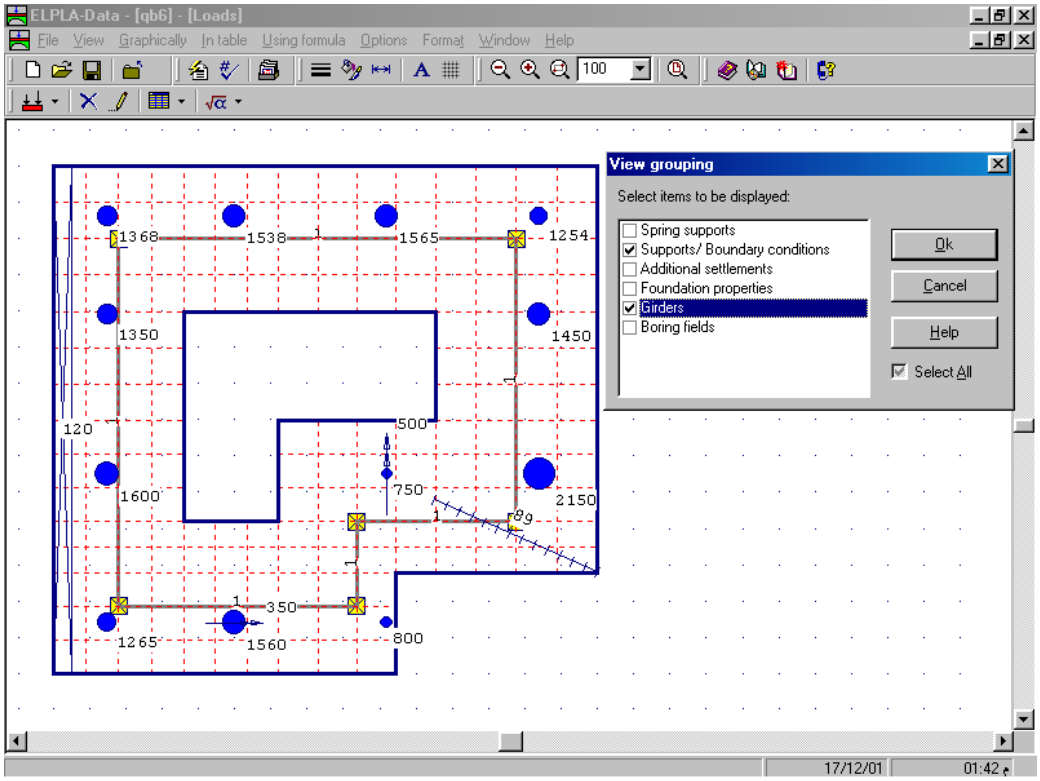


Figure C-44 Group of data together

5.3.7 Format Menu

The Format Menu has the following commands:

- Line formats
- Fill color
- Max. ordinate
- Font
- Grid

Format Menu–"Line formats" command

By the "Line formats" command the color, style and thickness of drawing lines can be defined, Figure C-45. The way a line is drawn depends on the setting of the color and style properties. There are 15 different colors and 5 styles available for line formats. The following list shows the available lines, which can be formatted:

- Slab boundary
- Elements
- Isometric view
- Distribution drawing
- Contour lines
- Legends
- Circular diagrams
- Arrows of support reactions
- Page boundary
- Identification box
- Streaks of principal moments (+ve)
- Streaks of principal moments (-ve)
- Slab dimensions
- Loads
- Boundary conditions
- Boring logs
- Slab thickness
- Axes of symmetry
- Beam elements
- Spring supports
- Axes of symmetry
- Symbol of symmetry
- Vectors of soil deformation
- Deformed mesh of soil deformation
- Axis arrows
- Girder axes
- Piles in plan
- Pile length
- Pile hatching
- Additional settlements
- Reference points and lines
- Columns
- Cartesian grid
- Punching shear
- Borders of block elements

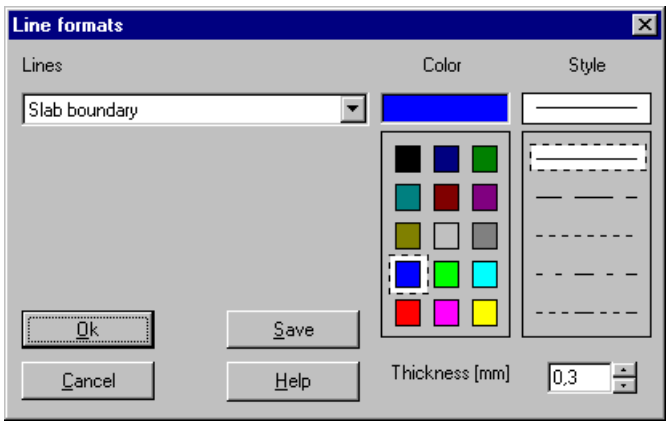


Figure C-45 "Line formats" Dialog box

Format Menu–"Fill color" command
By the "Fill color" command the fill color of drawing can be defined (Figure C-46).

The following list shows the available items, which can be filled with a specified color:

- Loads
- Boundary conditions
- Springs
- Zone type I: Bilinear interpolation among three boring logs
- Zone type II: Linear interpolation between two boring logs
- Zone type III: Node corresponds to boring
- Circle of boring
- Additional settlements
- (+ve) Circular diagrams
- (-ve) Circular diagrams
- Punching shear
- Material No.
- Girder group No.
- Sub area of boring No.
- Pile group No.
- Column group No.

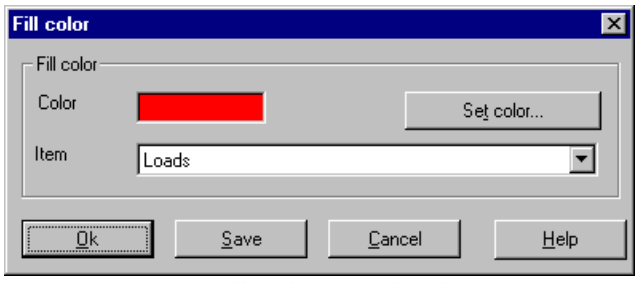


Figure C-46 "Fill color" Dialog box

Format Menu–"Max. ordinate" command
By "Max. ordinate" command the maximum ordinate, maximum diameter, maximum side, maximum width and maximum length for the drawing can be defined, Figure C-47.

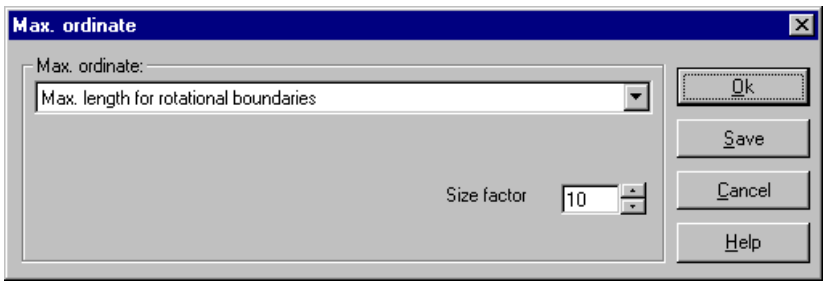


Figure C-47 "Max. ordinate" Dialog box

Format Menu–"Font" command

Here the font size (Figure C-48) and font type (Figure C-49) for the drawing can be defined.

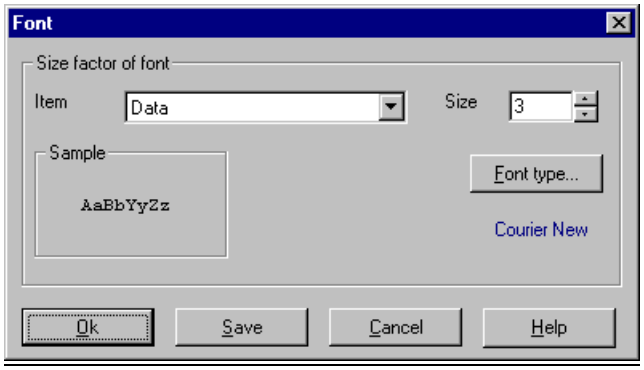


Figure C-48 "Font size" Dialog box

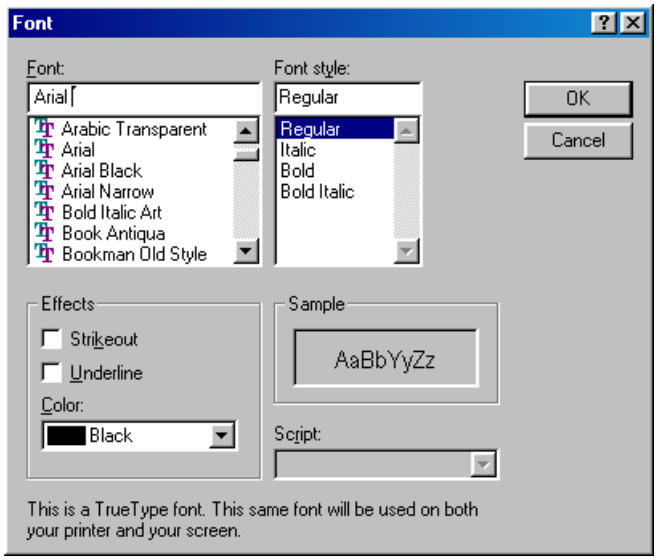


Figure C-49 "Font type" Dialog box

5.3.8 Window Menu

The Window menu has the following commands:

- Zoom in
- Zoom out
- Zoom window
- Zoom %
- Original size

Window Menu–"Zoom in" command
By the "Zoom in" command the size of the drawing on the screen can be reduced.

Window Menu–"Zoom out" command
By the "Zoom out" command the size of the drawing on the screen can be increased.

Window Menu–"Zoom window" command
By the "Zoom window" command the size of the drawing on a specified area can be increased.

Window Menu–"Zoom %" command
When you choose "Zoom %" command the following Dialog box appears, Figure C-50.

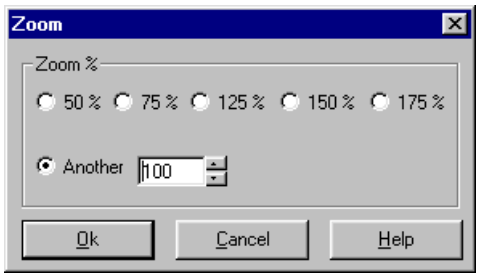


Figure C-50 "Zoom %" Dialog box

By the "Zoom %" command the size of drawing on the screen can be specified. Choosing "Zoom %" allows you to increase or decrease the size at which the drawing is displayed. Choosing "100%" displays the drawing at its original size. Clicking on the percentage changes the drawing size to the specified percentage. The drawing can be displayed at any size by typing the desired percentage in the specified Edit box.

Window Menu–"Original size" command
The commands "Zoom in", "Zoom out" and "Zoom %" can change the size of drawing on the screen. The drawing can be displayed in its original size again using the "Original size" command.

5.3.9 Help Menu

The Help Menu commands are:

- Contents
- Short description of ELPLA
- New in ELPLA
- About ELPLA-Data

Help Menu–"Contents" command
The "Contents" command displays a help file in HTML-Format containing the complete ELPLA User's Guide (Figure C-51).

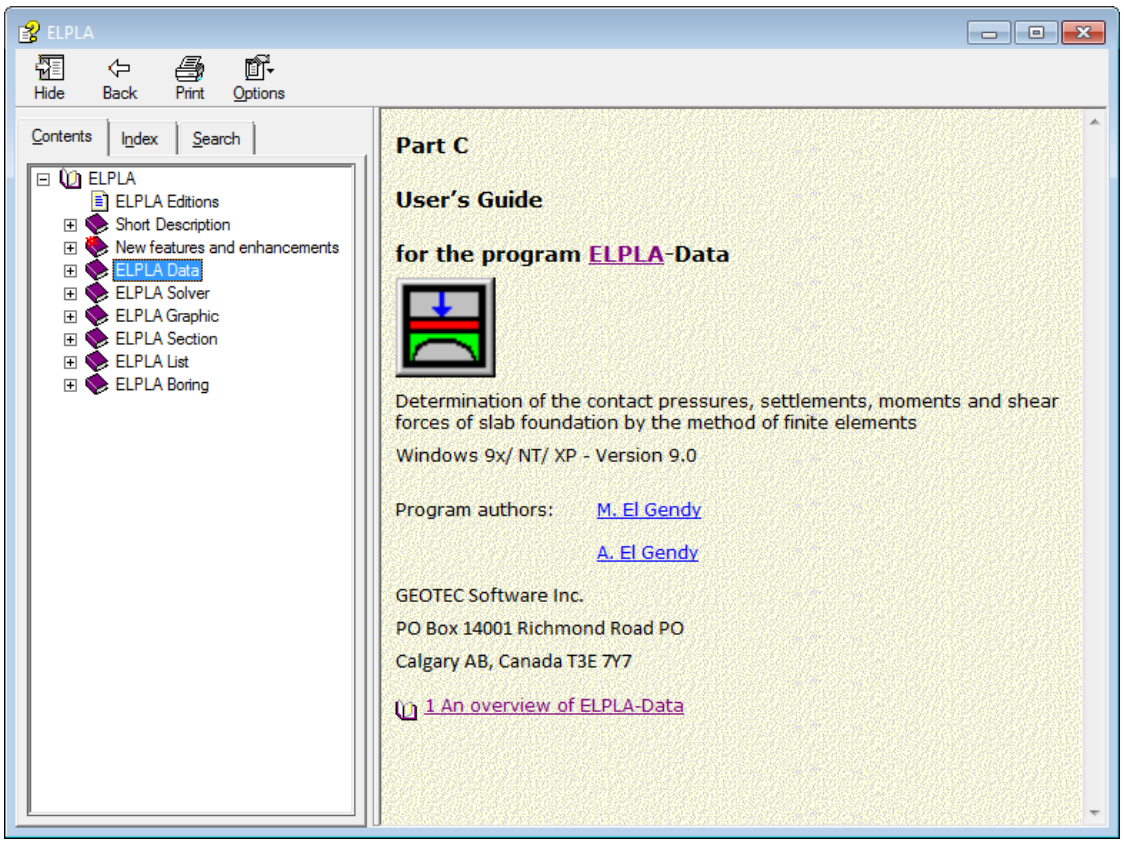


Figure C-51 Menu "Contents"

Help Menu–"Short description of ELPLA" command
The "Short description of ELPLA" command gives a short description of ELPLA package.

Help Menu–"New in ELPLA" command
The "New in ELPLA" command summarizes the new features and enhancements in ELPLA.

Help Menu–"About ELPLA-Data" command
Clicking the command "About ELPLA-Data" displays the information form of ELPLA-Data as shown in Figure C-52, which gives information about ELPLA-Data and the calculation method of the loaded project.

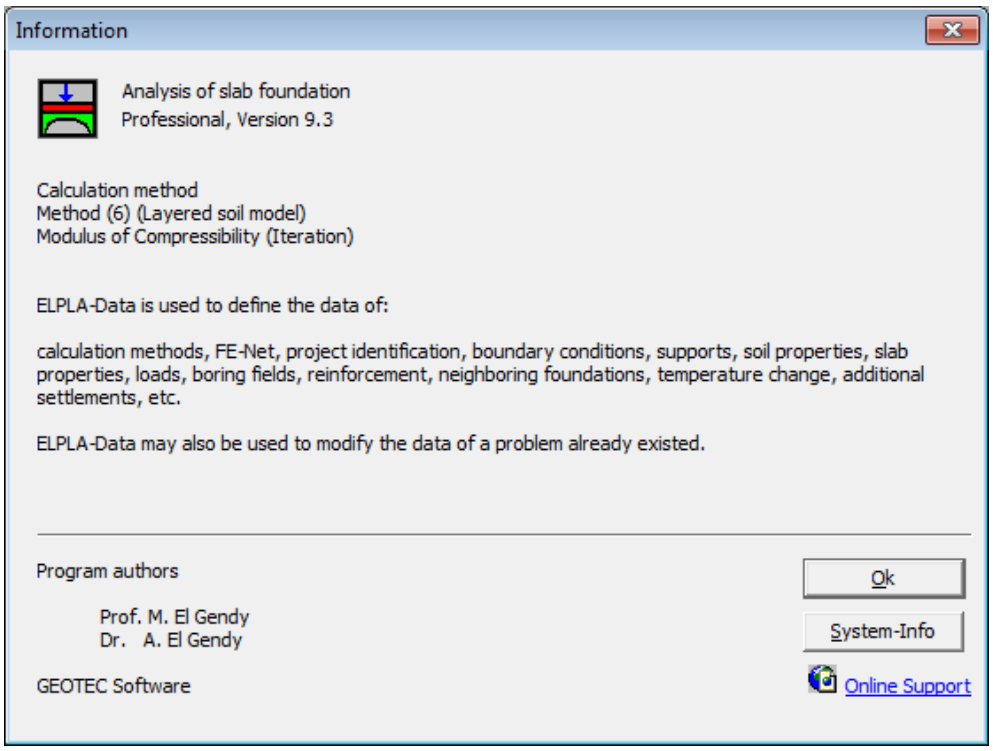


Figure C-52 Information form of program ELPLA-Data

5.3.10 Important notes

Element size

Sometimes edge moments M ≠ 0 appear on the slab, this means the element sizes are not optimal for the analysis. Therefore, if the moments of the slab are required, the length or width of the elements must be not longer than three times of slab thickness. Further more, the size of the middle element must be not more than three times the size of neighboring elements.

Change or modification of FE-Net

If the FE-Net of the slab is changed or modified for a new analysis, the input data of girders, boundary conditions, spring supports, slab properties, boring fields, etc. must be renewed. When closing a modified FE-Net, a Dialog box will appear (Figure C-53). This Dialog box shows the data that are set outside the FE-Net. To select the data to fix, check the boxes of these data.

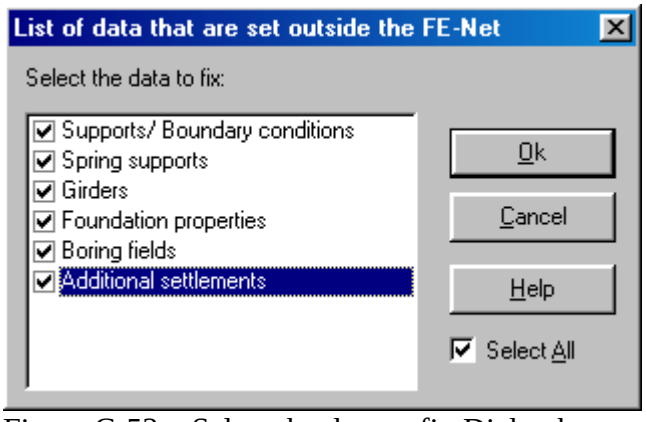


Figure C-53 Select the data to fix Dialog box

5.4 Data Menu–"Girders" command

In ELPLA beam elements are considered to represent the girder action in the slab. When the "Girders" command is chosen, the following embedded program appears, Figure C-54.

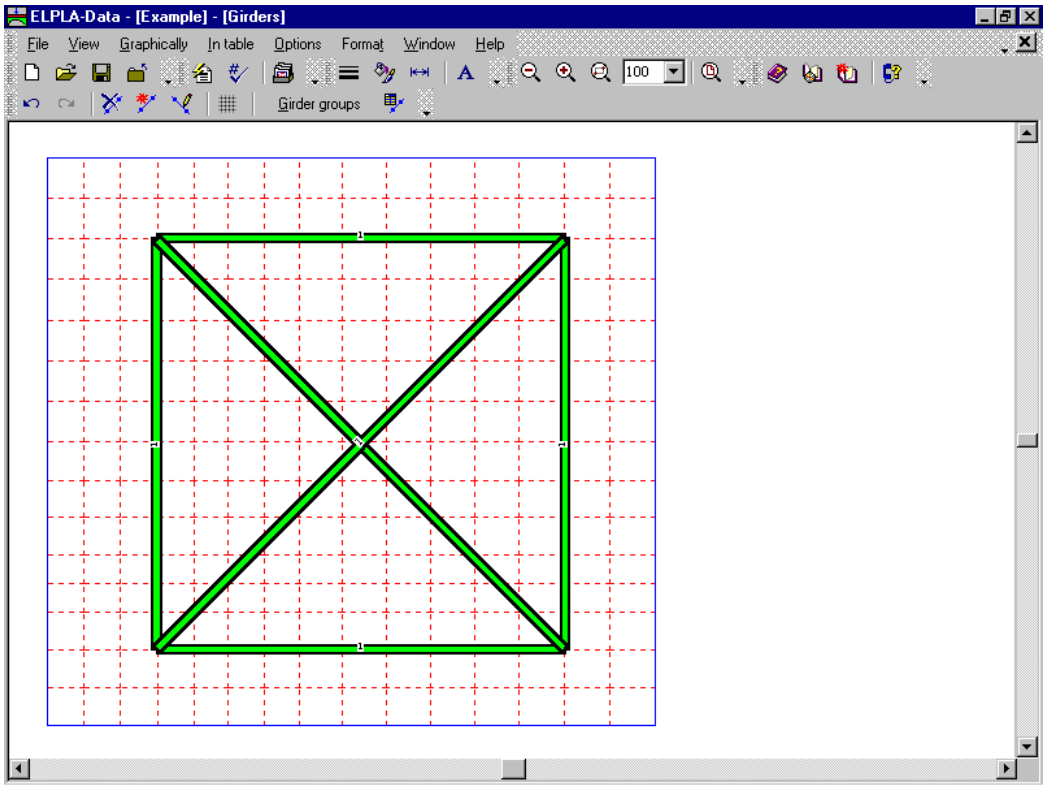


Figure C-54 "Girders" embedded program

The menu head of Figure C-54 contains the following commands:

- File
- View
- Graphically
- In table
- Options
- Format
- Window
- Help

The following paragraph presents and describes the menu commands and their sub-commands.

5.4.1 File Menu

This menu contains five commands:

- New girders
- Open girders
- Save girders
- Save girders as
- Close girders

File Menu–"New girders" command
New girders are defined.

File Menu–"Open girders" command
Existing girders-file are opened again. Then the girders, if desired, can be redefined.

File Menu–"Save girders" command
Saves the active girders under the available name

File Menu–"Save girders as" command
Saves the active girders under a new name

File Menu–"Close girders" command
Closes the Girders-embedded program and returns to ELPLA-Data.

5.4.2 View Menu (See paragraph 5.3.2)

5.4.3 Graphically Menu

The menu "Graphically" contains the following commands:

- Undo
- Redo
- Remove girders
- Add girders
- Edit girders
- Cartesian grid

Graphically Menu–"Undo" command
This command is used to undo the effects of a selected command and return to a previous state.

Graphically Menu–"Redo" command
This command is used to redo the last action of "Undo" command.

Graphically Menu–"Remove girders" command
When "Remove girders" command is chosen, the cursor is changed from an arrow to a cross hair. The desired girder is deleted by double clicking on it.

Graphically Menu–"Add girders" command
"Add girders" command is used to define and add a new girder. To add a girder, click the left mouse button on the start node of the girder and drag the mouse until the end node of the girder, then click on the end node.

Graphically Menu–"Edit girders" command
The main function of "Edit girders" command is to provide a method for redefining girders. When "Edit girders" command is chosen, the cursor is changed from an arrow to a cross hair. The desired girder is redefined by double clicking on it. Then, the following Dialog box of Figure C-55 appears.

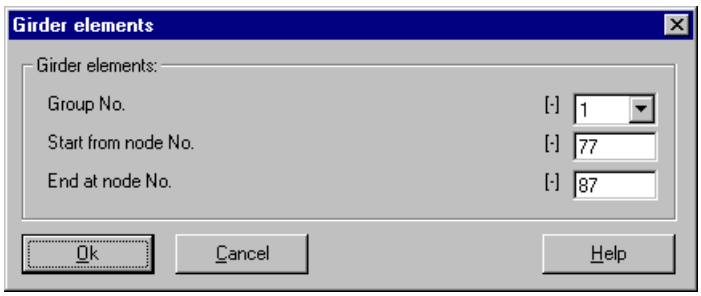


Figure C-55 "Girder elements" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.4.4 In table Menu

In ELPLA there are two alternative possibilities to define girders on the slab, graphically or numerically (in a table). The menu "In table" contains the following commands:

- Girder groups
- Girders

In table Menu–"Girder groups" command
When "Girder groups" command is chosen, the following Dialog box in Figure C-56 appears first to chose the option of cross section definition.

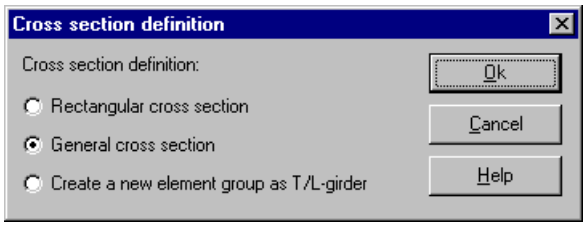


Figure C-56 "Cross section definition" Dialog box

In the "Cross section definition" Dialog box (Figure C-56) choose the option to define the girder cross section. The three alternative possibilities to define the girder groups are:

- Rectangular cross sections (Figure C-57)
- General cross sections (Figure C-58)
- T/L-Girders (Figure C-59)

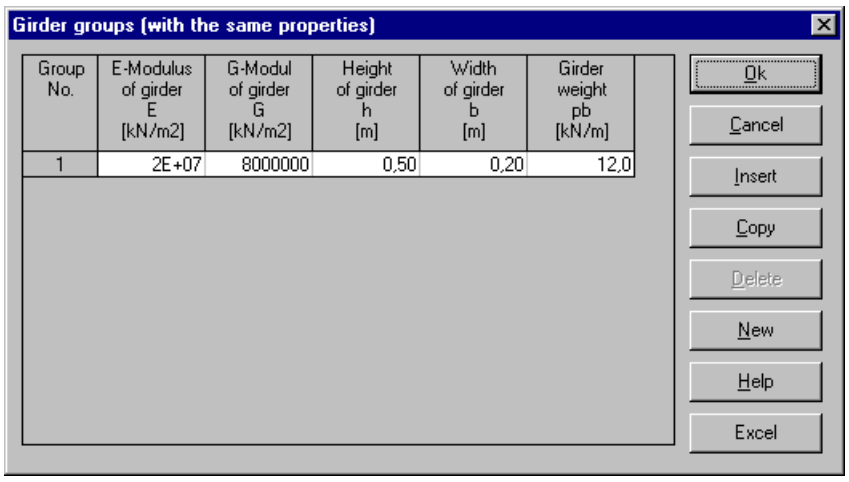


Figure C-57 "Defining girder groups – Rectangular cross section" Dialog box

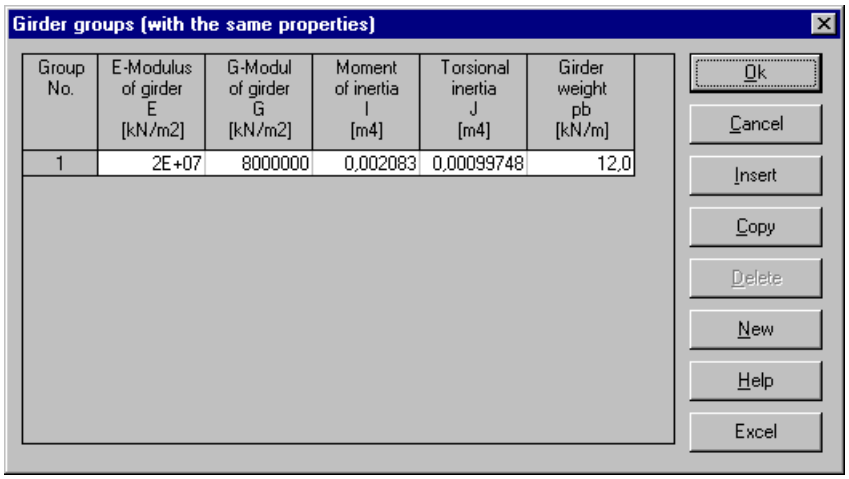


Figure C-58 "Defining girder groups – General cross section" Dialog box

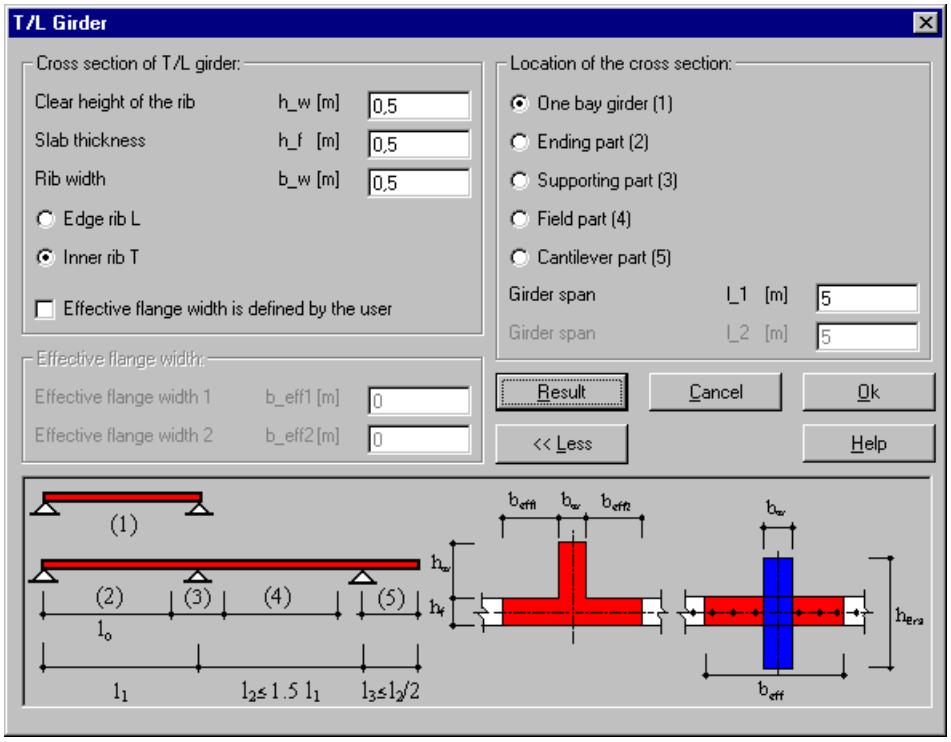


Figure C-59 "Defining girder groups – T/L-Girder" Dialog box

In the Dialog boxes of Figure C-57 and Figure C-58 E-Modulus, G-Modulus, girder section and girder weight for each group are required to define. Girder group is defined as a group of beam elements that have the same properties.

In the Dialog box of Figure C-59 the stiffness of girders that have T/L-section can be simulated in the slab by using additional beam elements. The stiffness of the girder can be obtained through a replacement beam arranged in the center plan of the slab. The dimensions of the replacement beam can be taken as in DIN 1075 or EC 2.

In table Menu–"Girders" command
Girder as beam element is defined through start and end nodes that at them the girder starts and ends (Figure C-60).

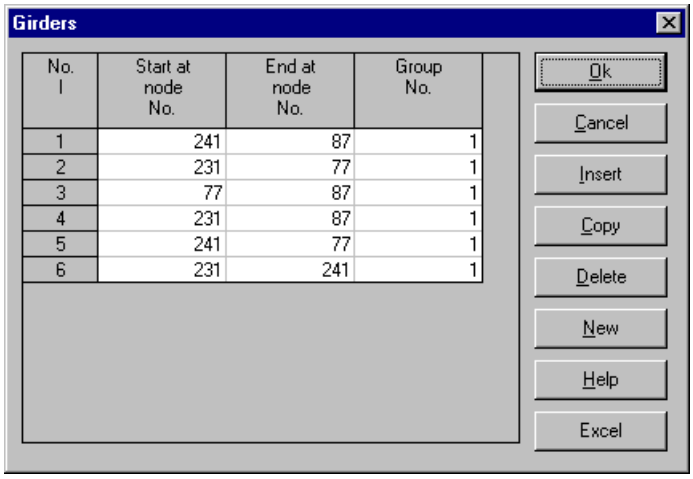


Figure C-60 "Girders in x-direction" Dialog box

5.4.5 Options Menu (See paragraph 5.3.6)

5.4.6 Format Menu (See paragraph 5.3.7)

5.4.7 Window Menu (See paragraph 5.3.8)

5.4.8 Help Menu (See paragraph 5.3.9)

5.5 Data Menu–"Spring supports" command

By the "Spring supports" command elastic support or node stiffness on the slab is defined. When the command is chosen, the following embedded program appears (Figure C-61).

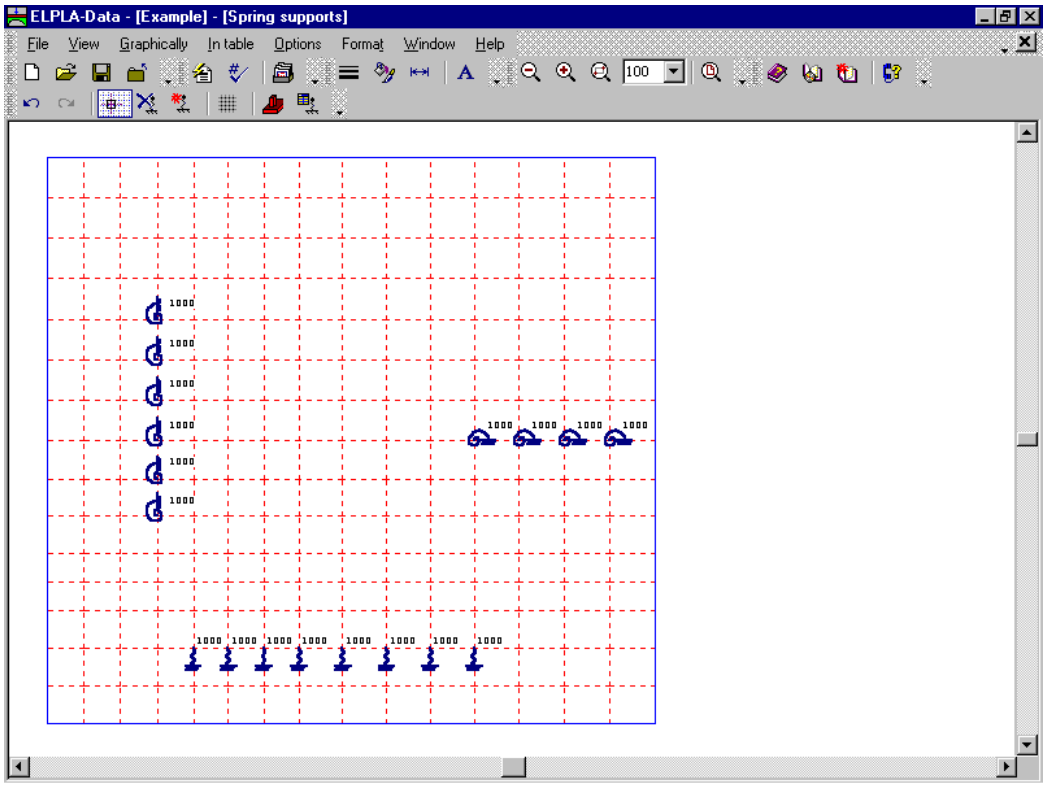


Figure C-61 "Spring supports" embedded program

Treatment of spring supports
The following types of spring supports are possible:

- Vertical spring k_z
- Rotational spring about x-direction k_{tx}
- Rotational spring about y-direction k_{ty}

Figure C-61 and the Dialog box of Figure C-63 show some samples of spring supports that may be used in ELPLA.

The menu head of Figure C-61 contains the following eight commands:

- File
- View
- Graphically
- In table
- Options
- Format
- Window
- Help

After clicking one of the eight commands (options) other sub-commands or options become available. The following paragraph presents and describes the eight menu commands and their sub-commands.

5.5.1 File Menu

This menu contains five commands:

- New spring supports
- Open spring supports
- Save spring supports
- Save spring supports as
- Close spring supports

File Menu–"New spring supports" command
Defines new spring supports.

File Menu–"Open spring supports" command
Opens existing spring supports-file again. Then the spring supports, if desired, can be redefined.

File Menu–"Save spring supports" command
Saves the active spring supports under the available name.

File Menu–"Save spring supports as" command
Saves the active spring supports under a new name.

File Menu–"Close spring supports" command
Closes the spring supports-embedded program and returns to ELPLA-Data

5.5.2 View Menu (See paragraph 5.3.2)

5.5.3 Graphically Menu

This menu contains the following commands:

- Undo
- Redo
- Select nodes
- Add spring supports
- Remove spring supports
- Cartesian grid

Graphically Menu–"Undo" command
This command is used to undo the effects of a selected command and return to a previous state.

Graphically Menu–"Redo" command
This command is used to redo the last action of "Undo" command.

Graphically Menu–"Select nodes" command
The main function of "Select nodes" command is to provide a method for removing or adding elastic nodes. When "Select nodes" command is chosen, the cursor is changed from an arrow to a cross hair. In this case, "Add spring supports" and also "Remove spring supports" will be enabled, indicating the modes in which are being operated. The desired nodes are selected by clicking on each node individually or selecting a group of nodes. A group of nodes can be selected by holding the left mouse button down at the corner of the region and dragging the mouse until a rectangle encompasses the desired group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

Graphically Menu–"Remove spring supports" command
This command is used to make the selected nodes free from spring supports.

Graphically Menu–"Add spring supports" command
It is used to define the stiffness for the selected nodes. Any old spring supports of the selected nodes will be replaced by the new editing. When "Add spring supports" command is chosen, the following Dialog box of Figure C-62 appears to define springs and column type.

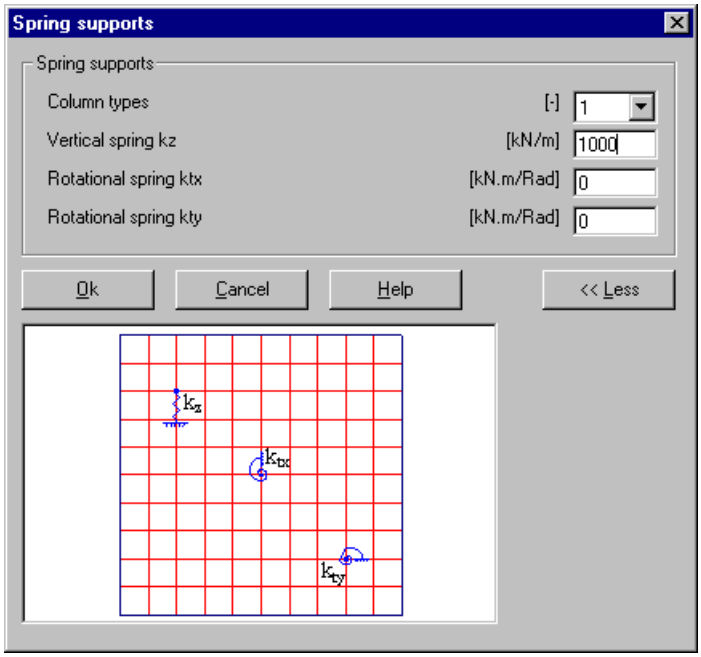


Figure C-62 "Add spring supports" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.5.4 In table Menu

This menu contains the following commands:

- Column types
- Spring supports

In table Menu–"Column types" command
When the "Column types" command is chosen, the following Table in Figure C-63 appears to define the column dimensions. Column dimensions are required for design of the slab for punching shear.

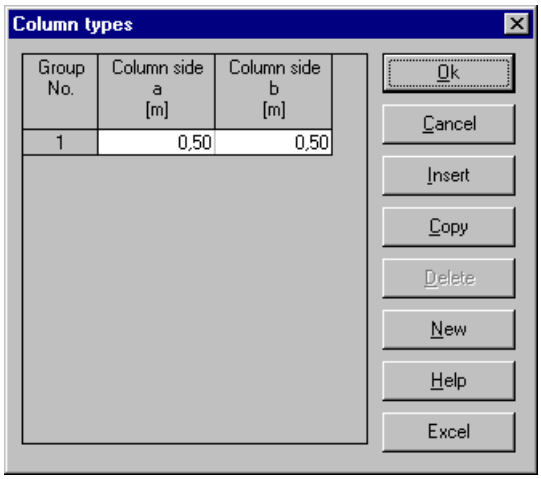


Figure C-63 Defining column dimensions

In table Menu–"Spring supports" command
When this command is chosen, the following Table in Figure C-64 appears. In this Table the elastic support is described by spring stiffness.

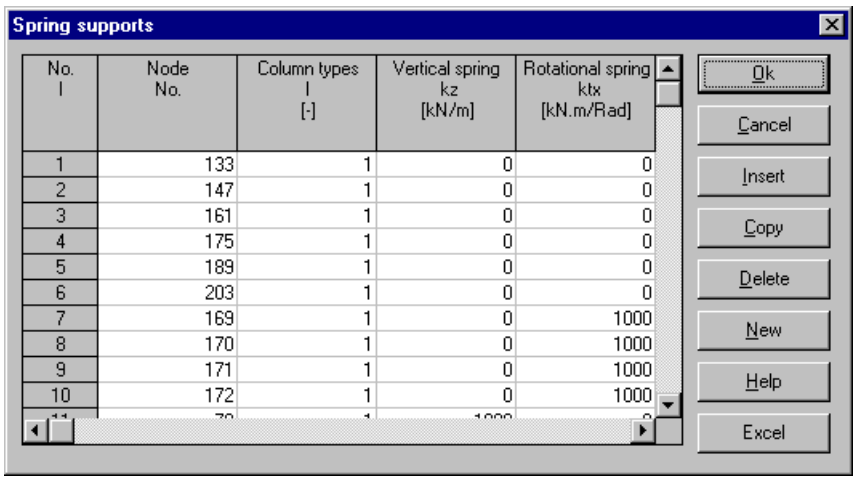


Figure C-64 Defining spring stiffness in a table

5.5.5 Options Menu (See paragraph 5.3.6)

5.5.6 Format Menu (See paragraph 5.3.7)

5.5.7 Window Menu (See paragraph 5.3.8)

5.5.8 Help Menu (See paragraph 5.3.9)

5.6 Data Menu–"Supports/ Boundary conditions" command

By this command supports or boundary conditions on the slab are defined. When this command is chosen, the following embedded program appears (Figure C-65).

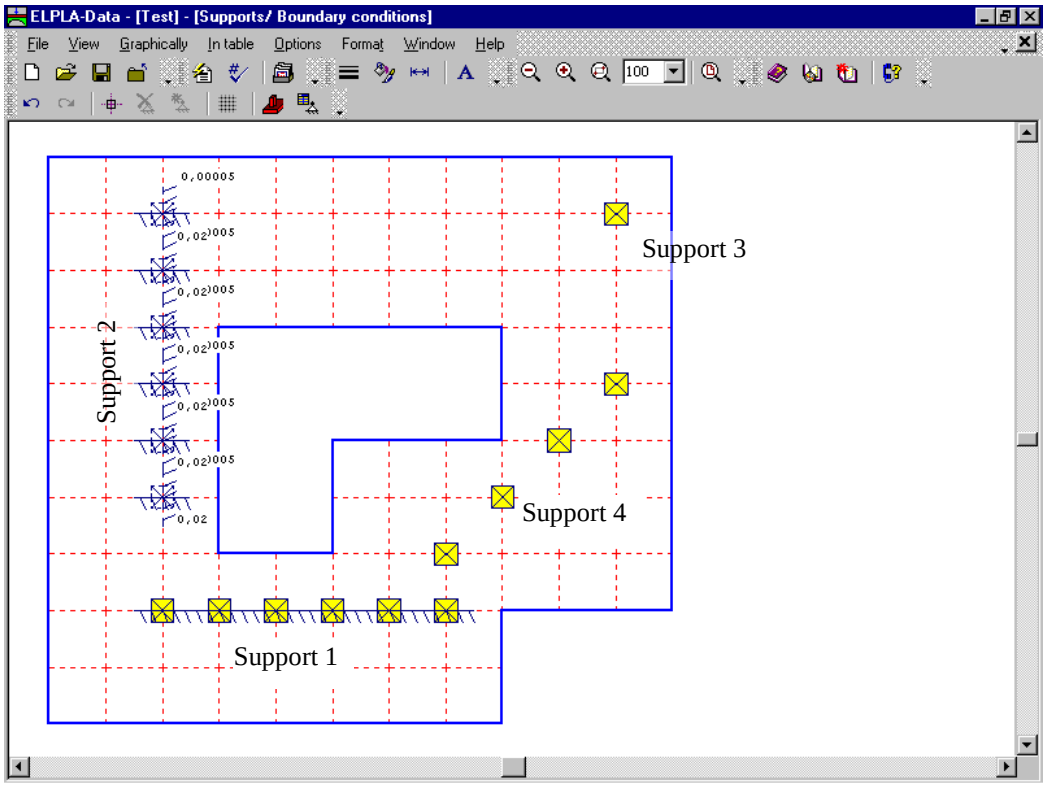


Figure C-65 "Supports/ Boundary conditions" embedded program

Treatment of rigid or elastic supports

Rigid or elastic nodes may represent the point and line supports. The following types of supports or boundary conditions are possible:

- Elastic or rigid displacements w
- Elastic or rigid rotations about x-direction θ_x
- Elastic or rigid rotations about y-direction θ_y

Figure C-65 and the Dialog box of Figure C-68 show some samples of supports and boundary conditions that may be used in ELPLA.

Illustrative example for point and line supports

The following example is used to describe point and line supports. The problem is an arbitrary slab foundation with variable type of supports as shown in Figure C-65 and the Dialog box of Figure C-68. The supports are:

- Support 1 is a line support in x-direction. All nodes are fixed against vertical displacement. No rotation is permissible about y-axis and the rotation about x-axis is free

- Support 2 is a line support in y-direction. All nodes have w =2 [cm] vertical displacement. No rotation is permissible about y-axis and the rotation about x-axis is θ_x = 0.00005

- Support 3 is a point support. This point is fixed against vertical displacement and the rotations about x- and y-axis are free

- Support 4 is a group of point supports (inclined line support). All nodes are fixed against vertical displacement and the rotations about x- and y-axis are free

The menu head of Figure C-65 contains the following eight commands:

- File
- View
- Graphically
- In table
- Options
- Format
- Window
- Help

After clicking one of the eight commands (options) other sub-commands or options become available. The following paragraph presents and describes the eight menu commands and their sub-commands.

5.6.1 File Menu

This menu contains five commands:

- New supports/ boundary conditions
- Open supports/ boundary conditions
- Save supports/ boundary conditions
- Save supports/ boundary conditions as
- Close supports/ boundary conditions

File Menu–"New supports/ boundary conditions" command
Defines new supports/ boundary conditions

File Menu–"Open supports/ boundary conditions" command
Opens existing supports/ boundary conditions-file again on the screen. Then the supports/ boundary conditions, if desired, can be redefined.

File Menu–"Save supports/ boundary conditions" command
Saves the active supports/ boundary conditions under the available name

File Menu–"Save supports/ boundary conditions as" command
Saves the active supports/ boundary conditions under a new name

File Menu–"Close supports/ boundary conditions" command
Closes the supports/ boundary conditions embedded program and returns to ELPLA-Data

5.6.2 View Menu (See paragraph 5.3.2)

5.6.3 Graphically Menu

This menu contains the following commands:

- Undo
- Redo
- Select nodes
- Add boundaries
- Remove boundaries
- Cartesian grid

Graphically Menu–"Undo" command
This command is used to undo the effects of a selected command and return to a previous state.

Graphically Menu–"Redo" command
This command is used to redo the last action of "Undo" command.

Graphically Menu–"Select nodes" command
The main function of "Select nodes" command is to provide a method for removing or adding restrains to nodes. When "Select nodes" command is chosen, the cursor is changed from an arrow to a cross hair. In this case "Add boundaries" and also "Remove boundaries" will be enabled, indicating the modes in which are being operated. The desired nodes are selected by clicking on each node individually or selecting a group of nodes. A group of nodes can be selected by holding the left mouse button down at the corner of the region and dragging the mouse until a rectangle encompasses the desired group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

Graphically Menu–"Remove boundaries" command
"Remove boundaries" command is used to make the selected nodes free from restraints.

Graphically Menu–"Add boundary" command
"Add boundary" command is used to define the restraints for the selected nodes. Any old restraints of the selected nodes will be replaced by the new editing. When "Add boundary" command is chosen, the following Dialog box of Figure C-66 appears to define supports, boundary conditions and column types.

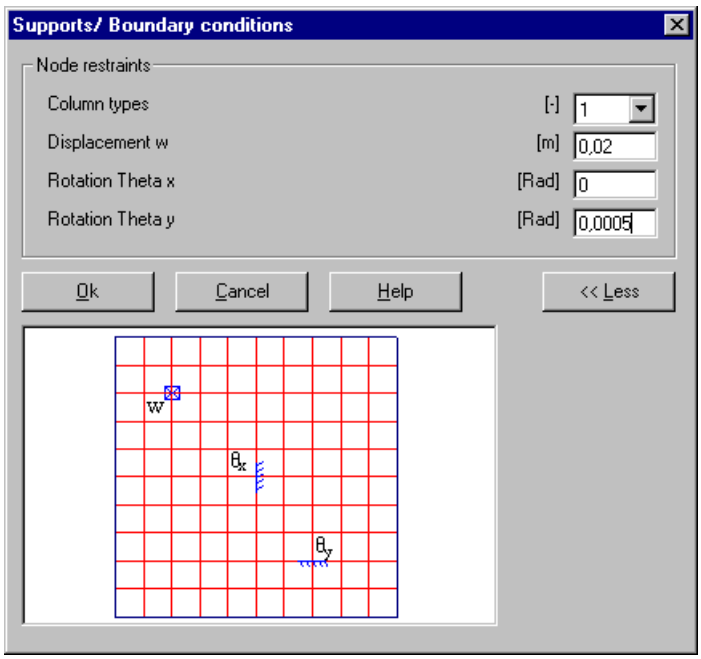


Figure C-66 Defining node restraints

The following input characters in the Dialog box of Figure C-66 mean:

- "0" means rigid support
- "F" means free displacement or rotation
- "0<or>0" means the value of elastic support

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.6.4 In table Menu

This menu contains the following commands:

- Column types
- Node restraints

In table Menu–"Column types" command
When this command is chosen, the following Table in Figure C-67 appears to define the column dimensions. Column dimensions are required for design of the slab for punching shear.

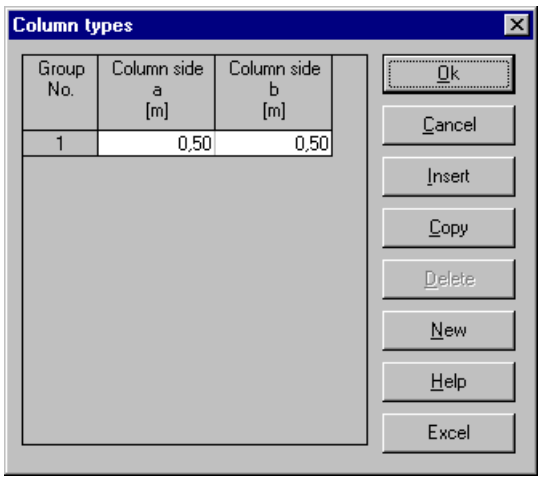


Figure C-67 Defining column dimensions

In table Menu–"Node restraints" command

When the "Node restraints" command is chosen, the following Table in Figure C-68 appears. In this Table define the node restraints and column types.

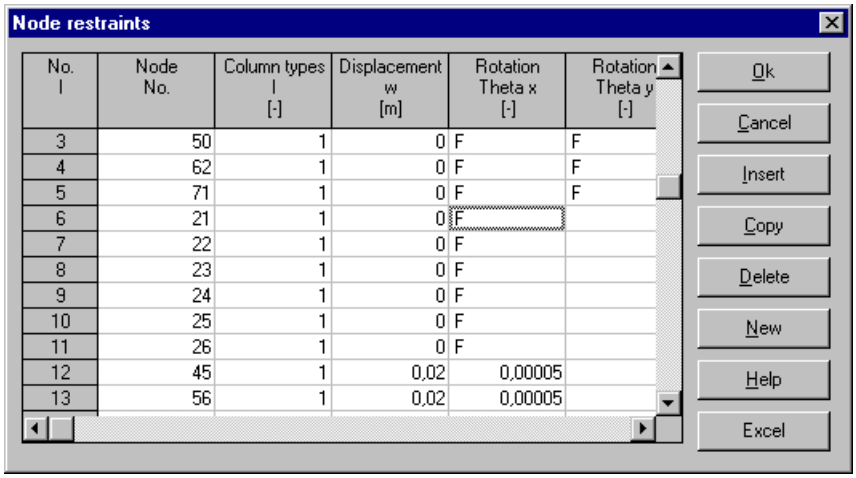


Figure C-68 Defining node restraints in a table

5.6.5 Options Menu (See paragraph 5.3.6)

5.6.6 Format Menu (See paragraph 5.3.7)

5.6.7 Window Menu (See paragraph 5.3.8)

5.6.8 Help Menu (See paragraph 5.3.9)

5.7 Data Menu–"Piles" command

By the "Piles" command piles on the slab are defined. When the "Piles" command is chosen, the following embedded program appears (Figure C-69).

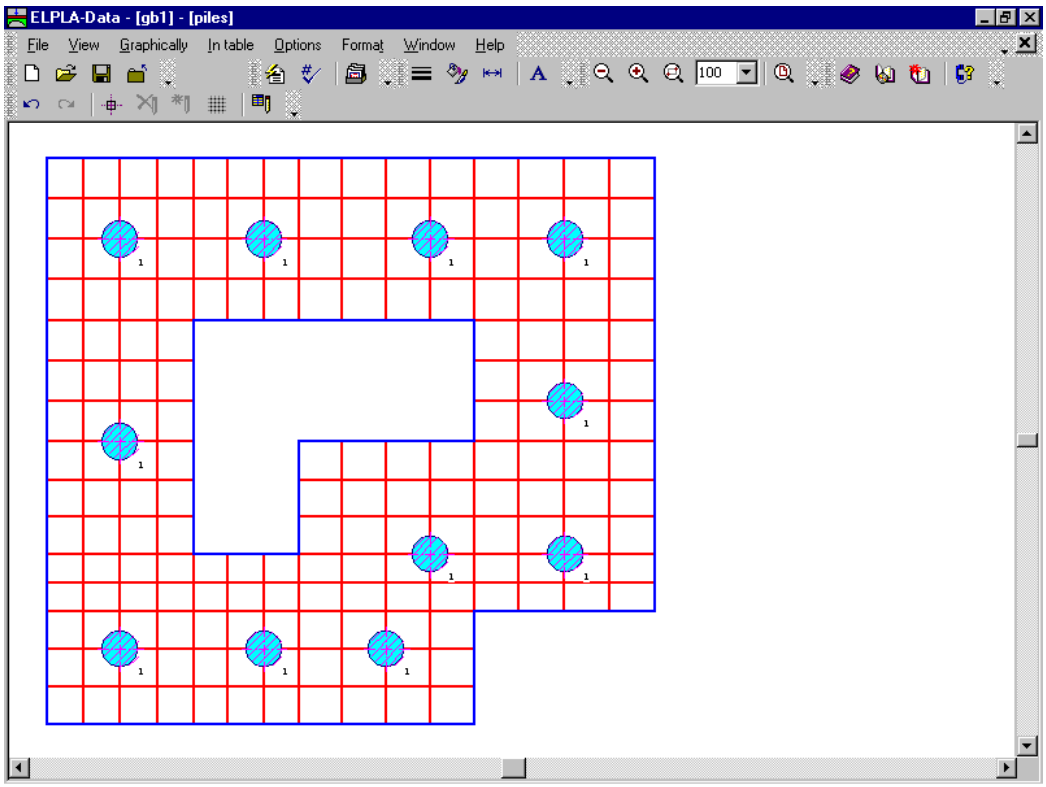


Figure C-69 "Piles" embedded program

The menu head of Figure C-69 contains the following eight commands:

- File
- View
- Graphically
- In table
- Options
- Format
- Window
- Help

The following paragraph presents and describes the menu commands and their sub-commands.

5.7.1 File Menu

This menu contains five commands:

- New piles
- Open piles
- Save piles
- Save piles
- Close piles

File Menu–"New piles" command
Defines new piles

File Menu–"Open piles" command
Opens existing piles-file again. Then the piles, if desired, can be redefined.

File Menu–"Save piles" command
Saves the active piles under the available name

File Menu–"Save piles as" command
Saves the active piles under a new name

File Menu–"Close piles" command
Closes the piles-embedded program and returns to ELPLA-Data

5.7.2 View Menu (See paragraph 5.3.2)

5.7.3 Graphically Menu

This menu contains the following commands:

- Undo
- Redo
- Select nodes
- Add spring supports
- Remove spring supports
- Cartesian grid

Graphically Menu–"Undo" command
This command is used to undo the effects of a selected command and return to a previous state.

Graphically Menu–"Redo" command
This command is used to redo the last action of "Undo" command.

Graphically Menu–"Select nodes" command
The main function of this command is to provide a method for removing or adding piles. When "Select nodes" command is chosen, the cursor is changed from an arrow to a cross hair. In this case "Add piles" and also "Remove piles" will be enabled, indicating the modes in which are being operated. The desired nodes are selected by clicking on each node individually or selecting a group of nodes. A group of nodes can be selected by holding the left mouse button down at the corner of the region and dragging the mouse until a rectangle encompasses the desired group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

Graphically Menu–"Remove piles" command
"Remove piles" command is used to make the selected nodes free from piles.

Graphically Menu–"Add piles" command
"Add piles" command is used to add piles at the chosen nodes and to define the pile group No. for these piles. Any old piles of the selected nodes will be replaced by the new editing. When "Add piles" command is chosen, the following Dialog box of Figure C-70 appears.

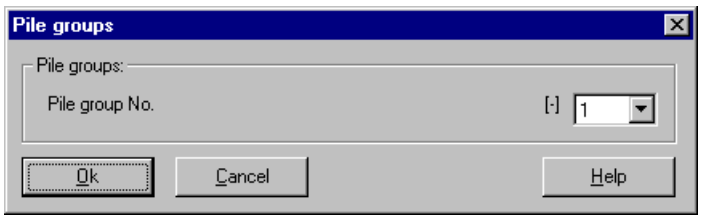


Figure C-70 "Add piles" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.7.4 In table Menu

This menu contains the following commands:

- Pile groups
- Pile location and groups
- Pile material

In table Menu–"Pile groups" command
In ELPLA there are different calculation methods to analyze the raft on piles. Therefore the pile groups for each method are required to define according to the used soil model as described in the next paragraphs.

Pile groups for Simple Assumption Model
In this model all forces acting on the raft will be transmit linearly on the piles. When the "Pile groups" command is chosen for this model, the following Table in Figure C-71 appears to define the pile diameter. Pile diameter is required for design of the slab for punching shear.

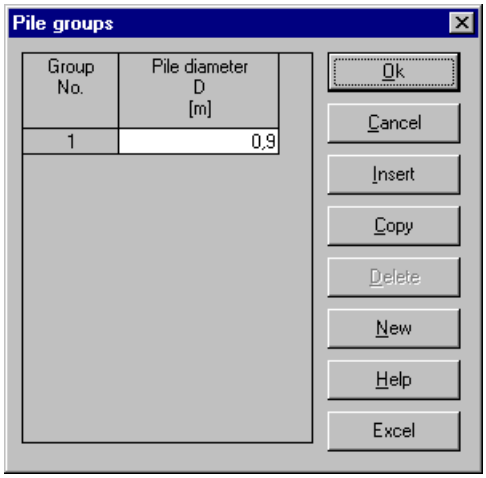


Figure C-71 Defining pile groups for Simple Assumption Model

Pile groups for *Winkler*'s Model
For the two methods of Constant and Variable Modulus of Subgrade Reaction (methods 2 and 3), when the modulus of subgrade reaction is required to define by the user, pile groups in this case will be the pile diameter and the pile stiffness (Figure C-72).

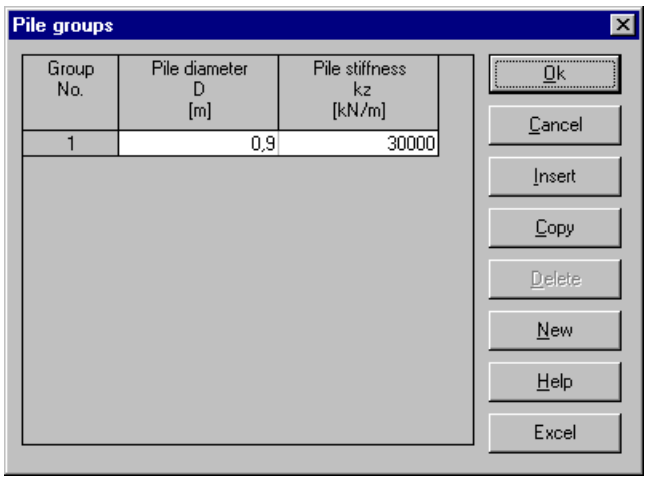


Figure C-72 Defining pile groups for *Winkler*'s Model

Pile groups for Isotropic Elastic Half-Space and Layered soil Models

When pile groups are required to define for one of these two soil models, the following Dialog box of Figure C-73 appears. The soil data around and under the pile are required to define. Soil data are used to determine the pile stiffness due to the soil type by ELPLA.

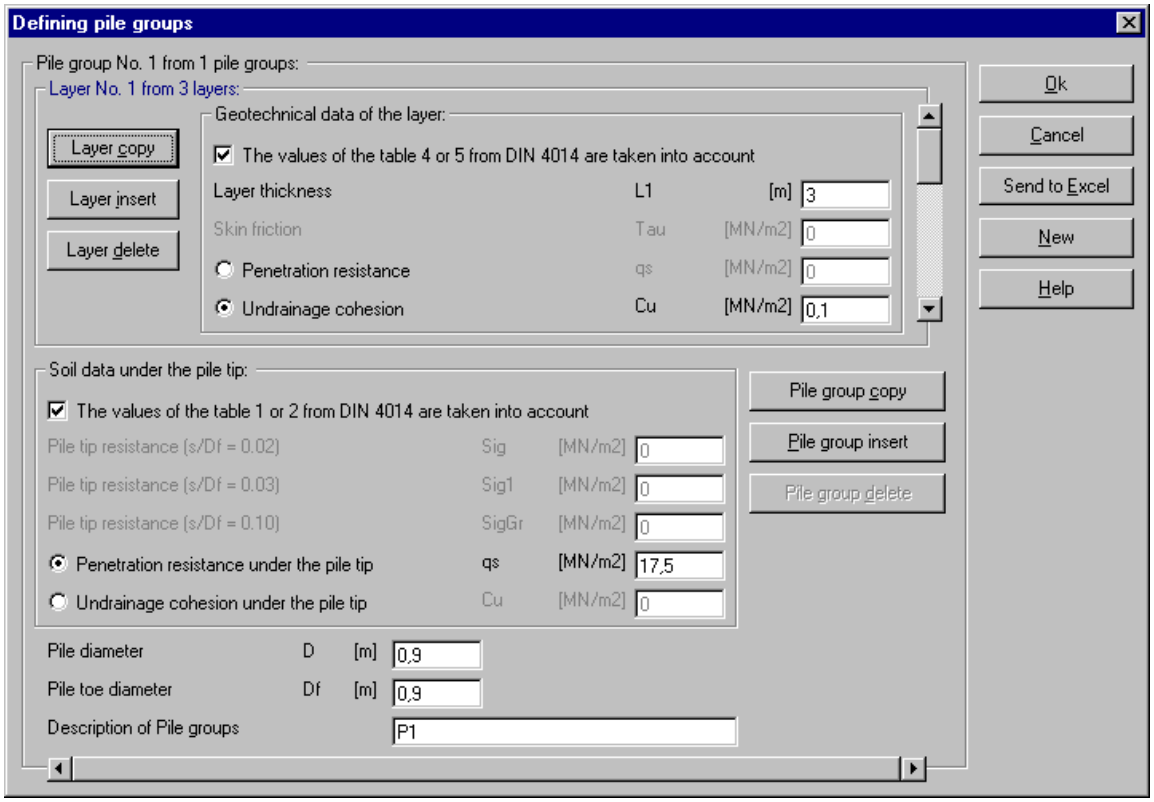


Figure C-73 Defining pile groups for Half-Space and Layered soil Models

In table Menu–"Pile locations and groups" command
When this command is chosen, the following Table in Figure C-74 appears. In this Table define the pile locations and their groups. Pile group is a group that has the same dimension and soil properties.

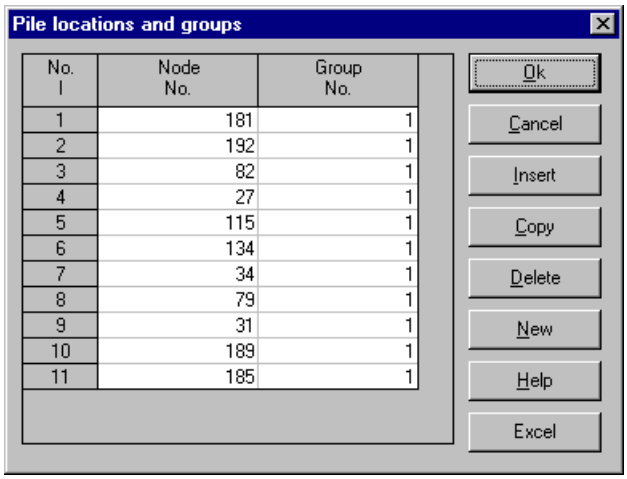


Figure C-74 Defining pile locations and groups in a table

In table Menu–"Pile material" command
When "Pile material" command is chosen, the following Dialog box in Figure C-75 appears. In this dialog box E-Modulus of the pile and the unit weight of the pile material are defined. Pile material are used to determine the pile stiffness due to its material type.

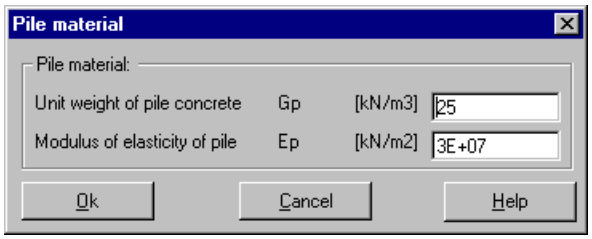


Figure C-75 "Defining element groups" Dialog box

5.7.5 Options Menu (See paragraph 5.3.6)

5.7.6 Format Menu (See paragraph 5.3.7)

5.7.7 Window Menu (See paragraph 5.3.8)

5.7.8 Help Menu (See paragraph 5.3.9)

5.8 Data Menu–"Soil properties" command

In ELPLA there are nine different calculation methods with different subsoil models. Therefore the soil properties for each method are required to define according to the used soil model as described in the next paragraphs.

Soil properties for Simple Assumption Model
There is no interaction between the subsoil and the foundation for Simple Assumption model (Linear Contact Pressure method - method 1). Therefore the soil data are not required in this method (only groundwater G_w and foundation level T_f are required). When soil properties are required to define for the calculation methods 1 (Linear Contact Pressure method), the following Dialog box of Figure C-76 appears.

If the water table is located above the foundation, the foundation will be exposed to an additional negative pressure. In the Dialog box of Figure C-76 define the groundwater depth under the ground surface G_w in order to take the effect of groundwater pressure in the analysis.

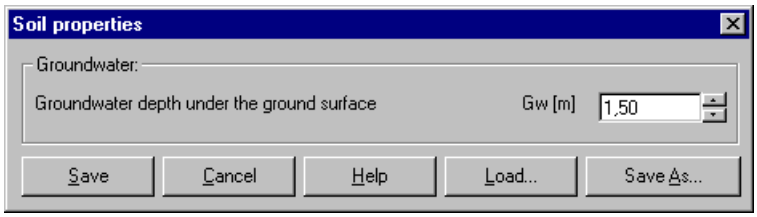


Figure C-76 "Soil properties" Dialog box (method 1)

Soil properties for *Winkler*'s Model
For the two methods of Constant and Variable Modulus of Subgrade Reaction (methods 2 and 3), when the modulus of subgrade reaction is required to define by the user, soil properties in this case will be the modulus of subgrade reaction k_s besides its coordinates (x, y) in the global system and groundwater depth under the ground surface G_w. If the nonlinear analysis is required, the ultimate bearing capacity of the soil q_{ult} must be defined (Figure C-77).

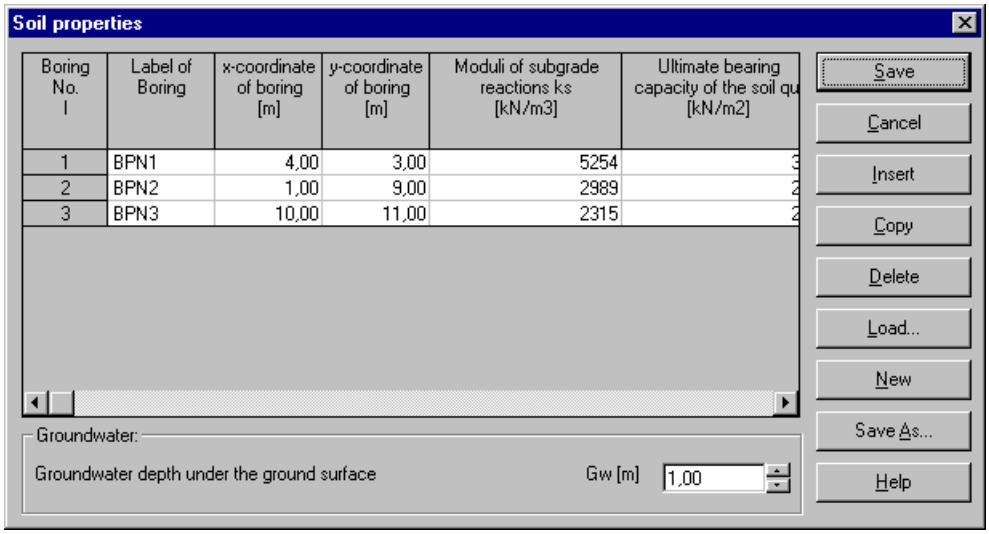


Figure C-77 "Soil properties" Dialog box (methods 2 and 3)

Soil properties for Isotropic Elastic Half-Space Model

When soil properties are required to define for the calculation method 2 (Modulus of subgrade reaction is determined from Half-Space) and calculation method 5 (Isotropic Elastic Half-Space), the following Dialog box of Figure C-78 appears.

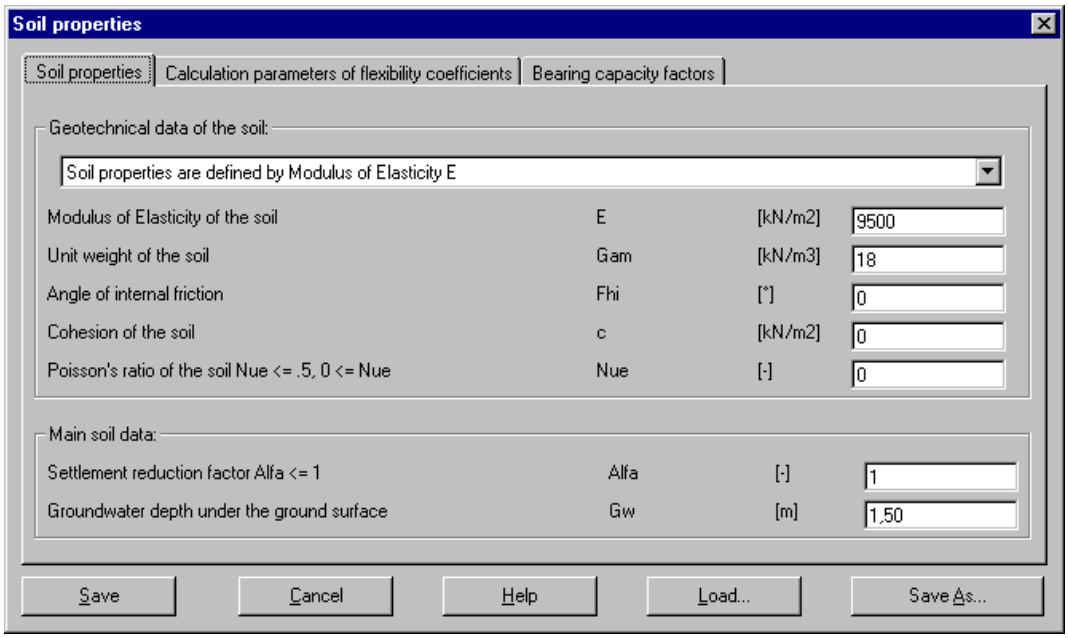


Figure C-78 "Soil properties" Dialog box (methods 2 and 5)

In the Dialog box of Figure C-78 define the settlement reduction factor α, *Poisson*'s ratio of the soil ν_s, groundwater depth under the ground surface G_w and the modulus of compressibility of the soil E_s. If the nonlinear analysis is required, the angle of internal friction φ and the cohesion c of the soil must be defined.

Soil properties for Layered soil Model

Layered soil model is used in the analysis of the calculation methods shown in Table C-4. When soil properties are required to define for one of the calculation methods shown in this Table, the following embedded program (Figure C-79) appears with default-boring logs.

Table C-4 Numerical calculation methods (Layered soil model)

Method No.	Method
2	Constant modulus of subgrade reaction *Winkler*'s model, Modulus of subgrade reaction is determined from soil layers
3	Variable modulus of subgrade reaction *Winkler*'s model, Modulus of subgrade reaction is determined from soil layers
4	Modification of modulus of subgrade reaction by iteration *Winkler*'s model/ Continuum model
6	Modulus of compressibility method for elastic raft on layered soil medium Solving system of linear equations by iteration, Layered soil medium - Continuum model
7	Modulus of compressibility method for elastic raft on layered soil medium Solving system of linear equations by elimination, Layered soil medium - Continuum model
8	Modulus of compressibility method for rigid raft on layered soil medium, Layered soil medium - Continuum model
9	Modulus of compressibility method for flexible foundation on layered soil medium, Layered soil medium - Continuum model

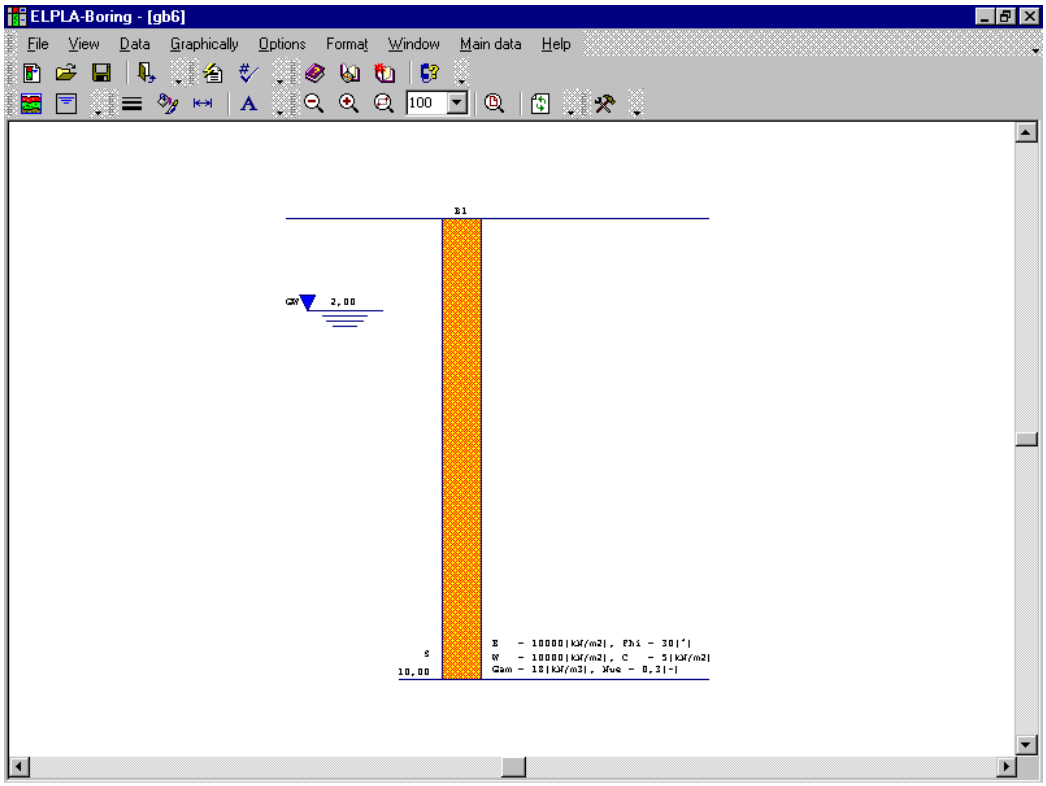


Figure C-79 "Soil properties" embedded program

The menu head of Figure C-79 contains the following nine commands:

- File
- View
- Data
- Graphically
- Options
- Formats
- Main data
- Window
- Help

After clicking one of the nine menus other sub-menus or commands become available. The following paragraph presents and describes the nine menu commands and their sub-commands.

5.8.1 File Menu

This menu contains five commands:

- New boring logs
- Open boring logs
- Save boring logs
- Save boring logs as
- Close boring logs

File–"New boring logs" command
Defines new boring logs

File–"Open boring logs" command
Opens existing boring log-files again on the screen. Then they, if desired, can be redefined.

File–"Save boring logs" command
Saves the active boring logs under the available name

File–"Save boring logs as" command
Saves the active boring logs under a new name

File–"Close boring logs" command
Closes the Boring-embedded program and returns to ELPLA-Data.

5.8.2 View Menu (See paragraph 5.3.2)

5.8.3 Data Menu

The menu "Data" contains the following commands:

- Soil data
- Main soil data

Data Menu–"Soil data" command
When "Soil data" command is chosen, the following Dialog box in Figure C-80 appears.

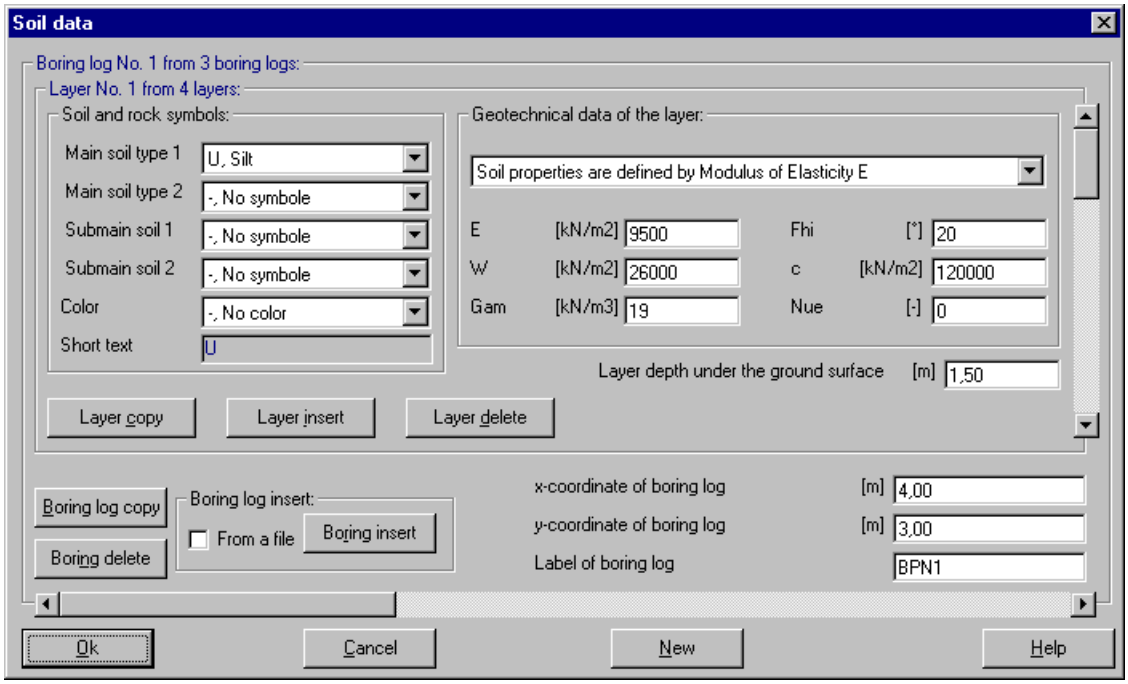


Figure C-80 "Soil data" Dialog box

Elastic settlement and consolidation settlement can be determined using their actual properties, where the soil properties of the individual layers are defined by:

- Modulus of Compressibility Es (1/mv)
- Modulus of Elasticity E
- Compression Index Cc

This option enables ELPLA to analyze rafts on consolidated clay deposits by the different calculation methods that are available in ELPLA. Also the user doesn't need to convert a soil parameter to other. When defining soil properties by the Modulus of Elasticity E, the *Poisson's* ratio ν_s can be different for every layer.

In the Dialog box of Figure C-80 the soil under the foundation may be defined by a number of boring logs. Each boring log has multi-layers with different soil materials. The geotechnical data for each layer are unit weight of the soil γ_s, modulus of compressibility for loading Es (or Modulus of Elasticity E or Compression Index Cc) and for reloading Ws and *Poisson's* ratio of the soil ν_s. If the nonlinear analysis is required, the angle of internal friction φ and the cohesion c of the soil must be defined; besides the boring coordinates x, y in the global system and its label are required to define. In order to draw the soil layers by different symbols according to the German specification code DIN 4023, define soil art and color in the Dialog box of Figure C-80.

Copy, Insert and Delete commands for both soil layer and boring log are available in the Dialog box of Figure C-80. Further more, boring logs can be inserted from a file by checking the option "From file" in the "Boring insert" Frame box (Figure C-81).

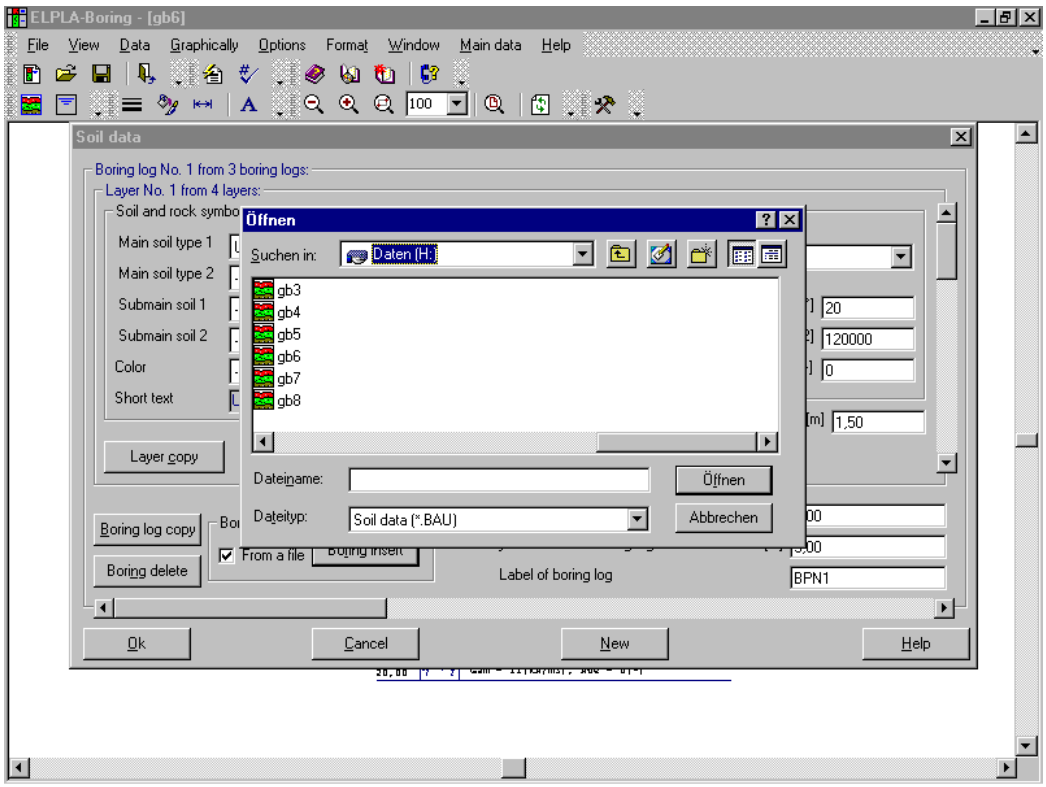


Figure C-81 Boring log can be inserted from a file

Data Menu–"Main soil data" command

Main soil data is the general data for all soil layers and boring logs. When "Main soil data" command is chosen, the following Dialog box in Figure C-82 appears.

Settlement reduction factor

According to experience the real consolidation settlements are different from those calculated. Therefore the settlement s may be multiplied by a factor α according to the German standard DIN 4019. According to the German standard DIN 4019 the following reduction factors α can be applied:

Sand and silt	α = 0.66
Normally and slightly over consolidated clay	α = 1.0
Heavily over consolidated clay	α = 0.5-1.0

In the Dialog box of Figure C-82 define the settlement reduction factor α and the groundwater depth under the ground surface G_w.

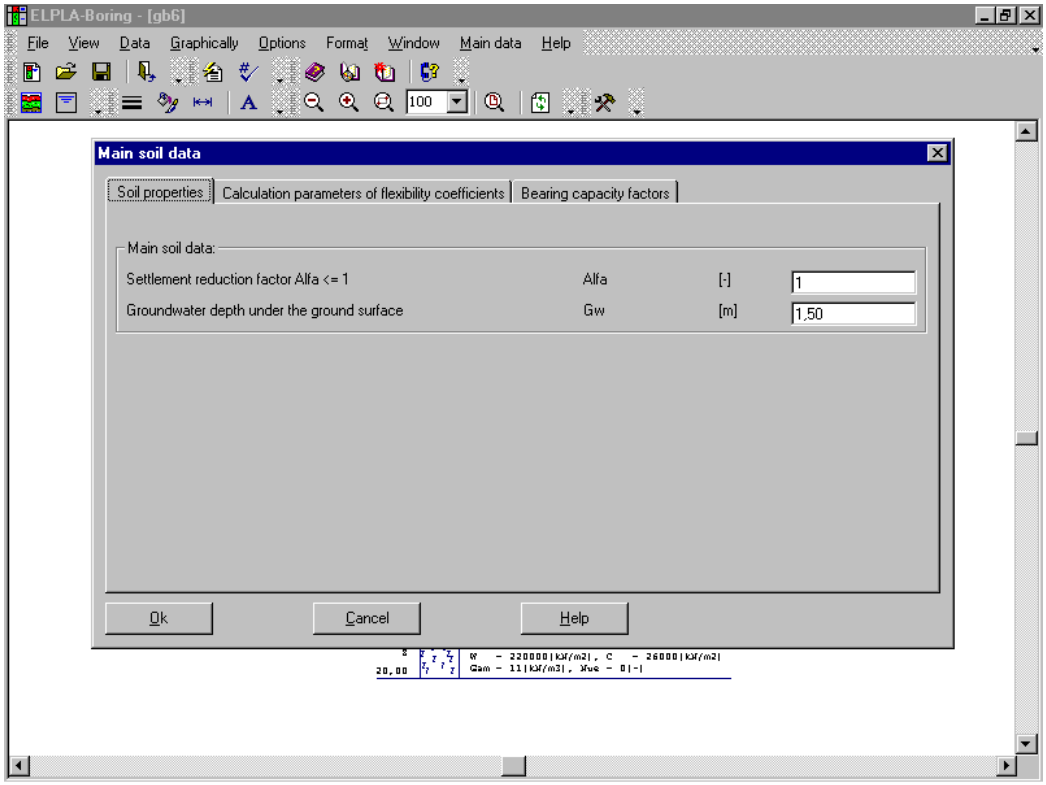


Figure C-82 "Main soil data" dialog box

Bearing capacity factors

The bearing capacity factors used to determine the ultimate bearing capacity can optionally be defined according to different codes and authors. These factors are required to carry out the nonlinear analysis of the soil. The bearing capacity factors are defined according to Figure C-83:

- German Standard DIN 1054
- Euro Code EC 7
- Egyptian code ECP
- Terzaghi
- Meyerhof

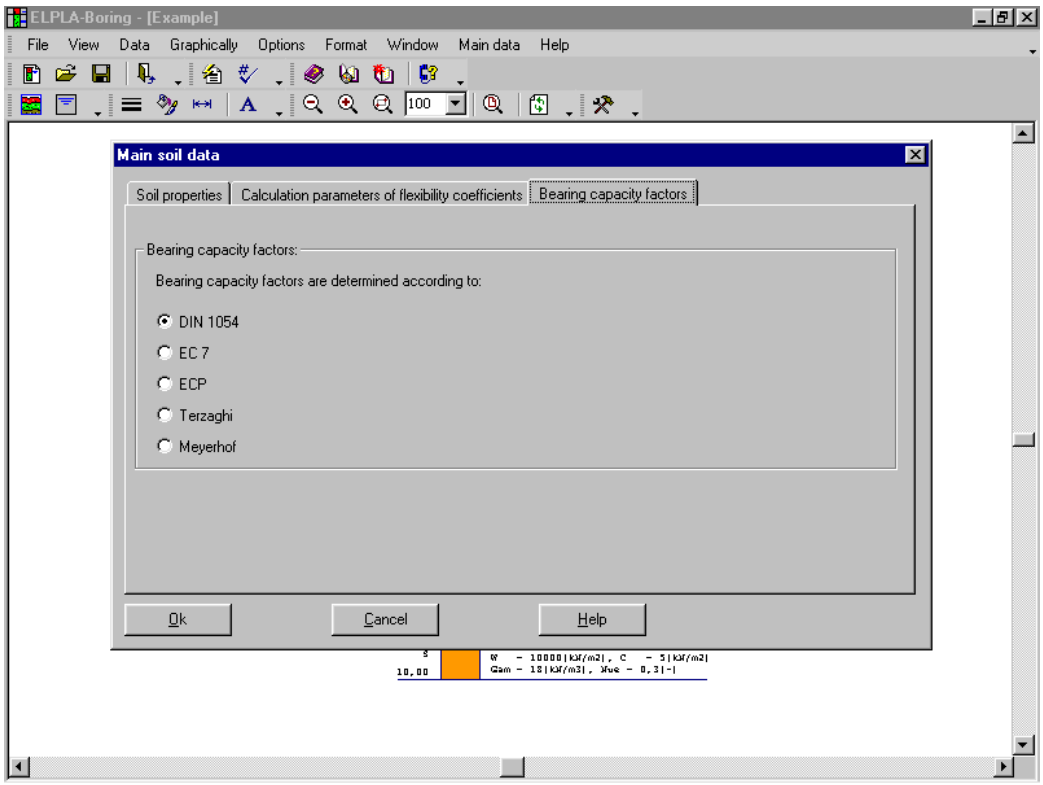


Figure C-83 Menu "Bearing capacity factors"

Flexibility coefficients for interior nodes

For rigid and elastic rafts it is convenient to determine the flexibility coefficient of interior node at the characteristic point of the loaded area on that node. For flexible foundation it is real to determine the flexibility coefficient of interior node at that node.

Now it is possible to determine the flexibility coefficient of the interior node due to a uniform load at that node (0):

- at the characteristic point of the loaded area, where rigid settlement is equal to flexible settlement
- at the midpoint of the loaded area, where maximum settlement occurs
- at the interior node on the loaded area

Flexibility coefficients for exterior nodes

Earlier versions of ELPLA determine flexibility coefficients for both interior and exterior nodes by assuming uniform loaded areas on these nodes. This assumption needs to use the principle of superposition for determining the flexibility coefficients. Now it is possible, optionally to convert the loaded areas on exterior nodes to point loads, Figure C-84. By this way the program doesn't need to use the principle of superposition in the analysis, making it much faster than the old analysis. The new way of analysis is consequently faster and more efficient for problems that contain a large finite element mesh.

Limit distance
If the distance between two nodes is too large, the settlement of a node due to a load on the other will be small enough to be neglected. To reduce the time required for determining the flexibility coefficients for great rafts, a limit distance between node i and j for determining the flexibility coefficient c(i, j) may be defined, Figure C-84.

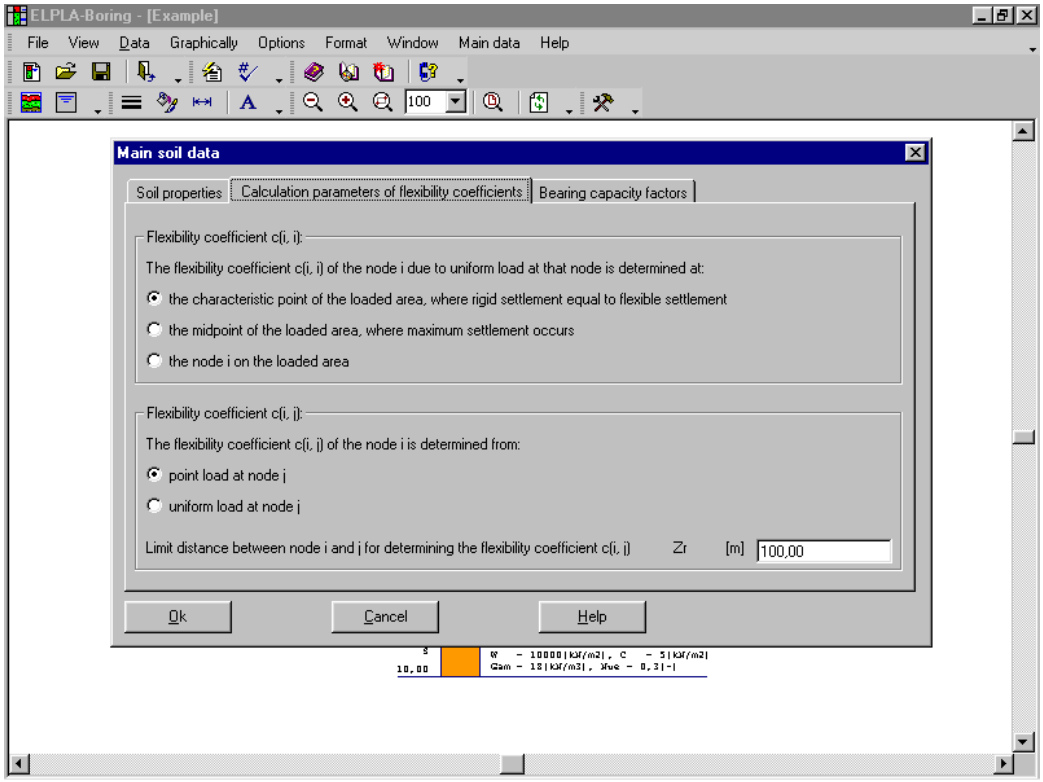


Figure C-84 Menu "Flexibility coefficients"

5.8.4 Graphically Menu

In the program ELPLA it is also possible to define the boring logs graphically. This makes the definition of the boring logs very easy. This option is used also for drawing the defined boring logs to make a control on the input soil data and parameters.

The menu "Graphically" contains one command:

- Drawing boring logs

Graphically Menu–"Drawing boring logs" command
When "Drawing boring logs" command is chosen, the following Dialog box in Figure C-85 appears. In this Dialog box chose the boring logs you want to draw, then click on Button "OK". Then the menu of Figure C-86 appears to control input soil data and parameters or to redefine the boring data.

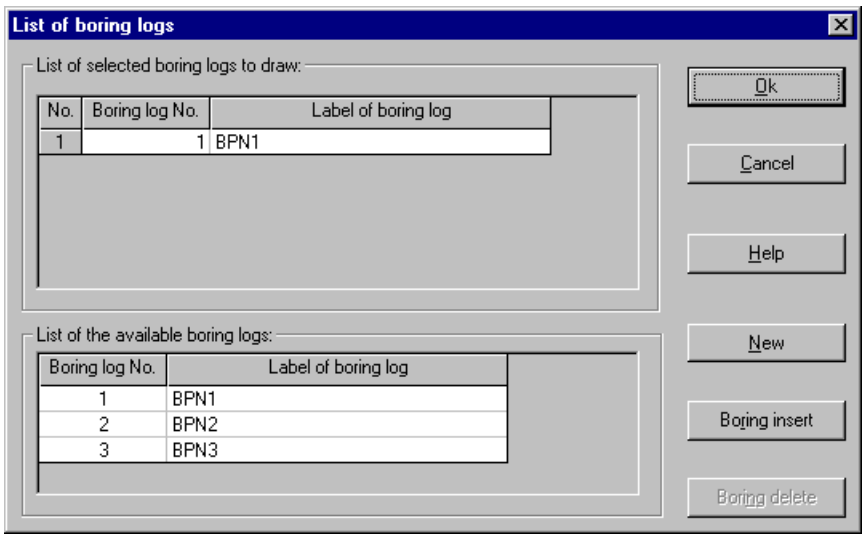


Figure C-85 "List of boring logs" Dialog box

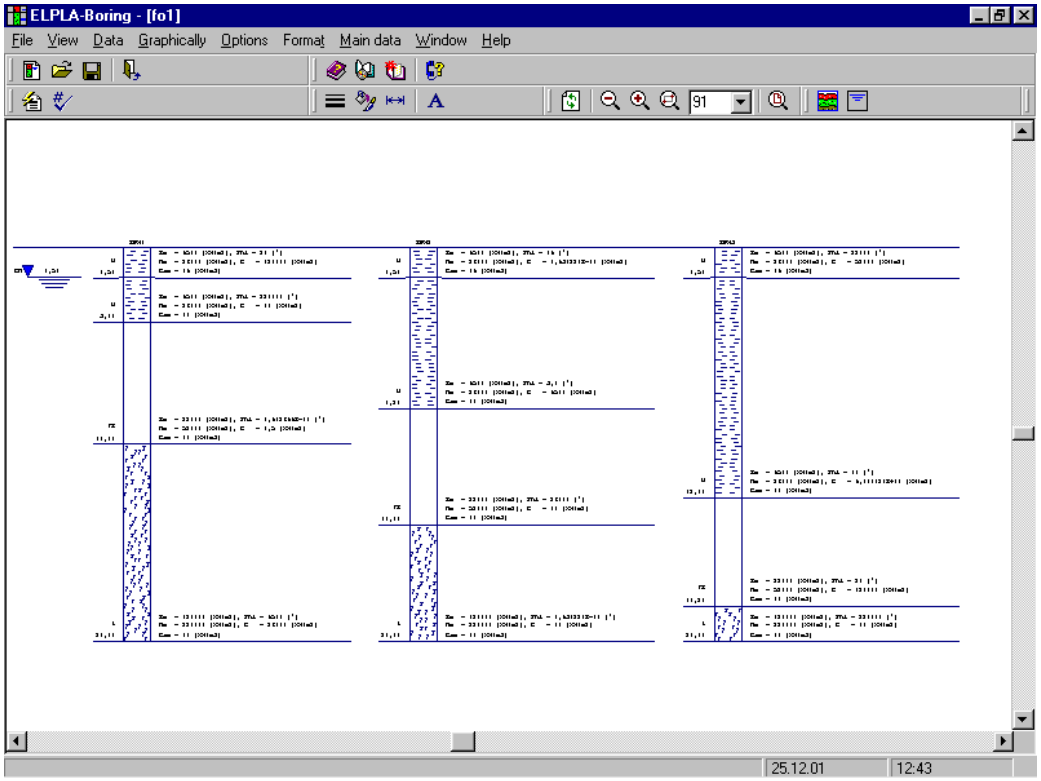


Figure C-86 Boring logs on the screen

Definition of boring logs graphically

By double-clicking the left mouse Button on a specified screen position, the user can also define soil data and input parameters.

- By double-clicking on the geotechnical data of a soil layer the corresponding Dialog box to define the geotechnical data of that layer appears, Figure C-87

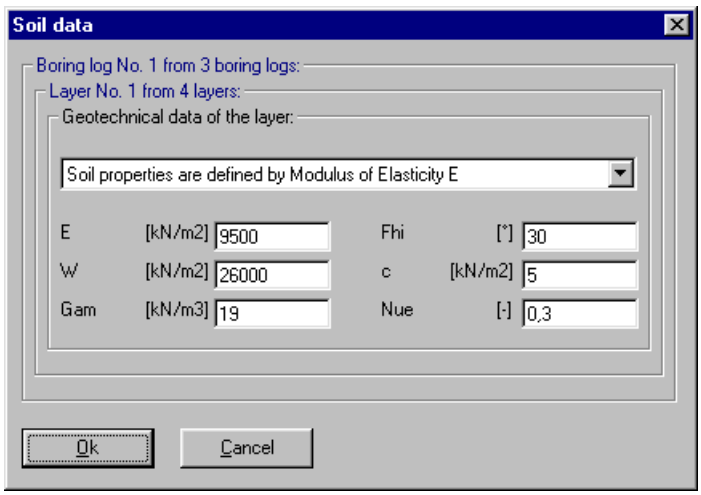


Figure C-87 "Geotechnical data of the layer" Dialog box

- By double-clicking on the layer level the corresponding Dialog box to define the layer depth under the ground surface appears, Figure C-88

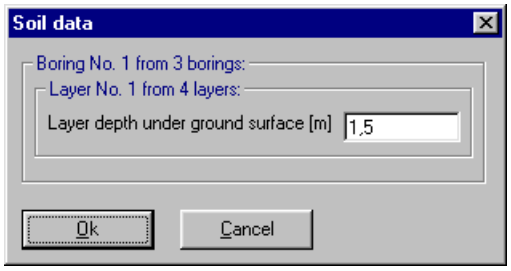


Figure C-88 "Layer depth under the ground surface" Dialog box

- By double-clicking on the soil symbol of a soil layer the corresponding Dialog box to define the soil symbols of that layer appears, Figure C-89

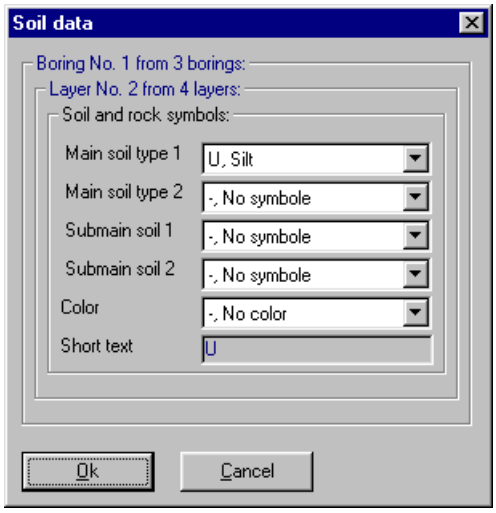


Figure C-89 "Sand and rock symbols" Dialog box

- By double-clicking on the groundwater level the corresponding Dialog box to define the groundwater depth under the ground surface appears, Figure C-90

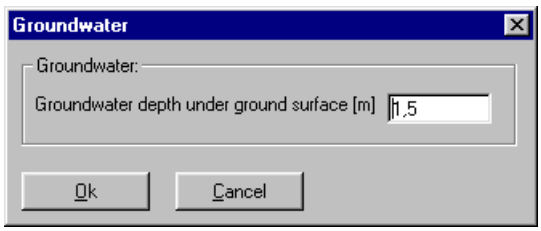


Figure C-90 "Groundwater" Dialog box

- By double-clicking on the label of a boring log the corresponding Text box to define the label of that boring log appears, Figure C-91

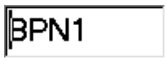


Figure C-91 "Label of the boring log" Text box

5.8.5 Options Menu

The Options Menu has the following commands:

- Plot parameters
- Display values

Options Menu–"Plot parameters" command
Plot parameters may be set as default values by the program, or may be fully specified by the user.

By the "Plot parameters" command the following plot parameters can be specified, Figure C-92:

- Color soil layers
- Draw water table
- Simple drawing of boring logs
- Setting soil colors according to DIN 4023
- Display soil properties c, Phi and Nue

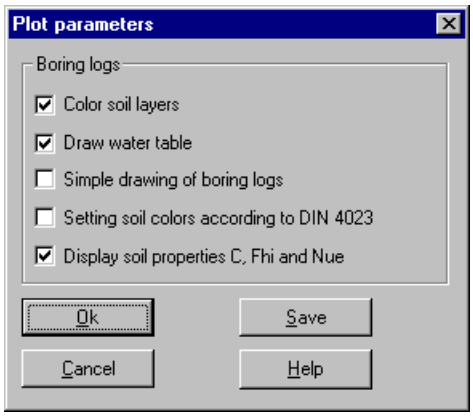


Figure C-92 "Plot parameters" Dialog box

Options Menu–"Display values" command
By this command the values of the following items can be displayed on the drawing, Figure C-93:

- Label of boring
- Layer description
- Layer depth
- Display text of soil symbols
- Measurement bar (available in ELPLA-Boring)
- Foundation (available in ELPLA-Boring)
- Water level
- Limit depth (available in ELPLA-Boring)
- Stress value (available in ELPLA-Boring)

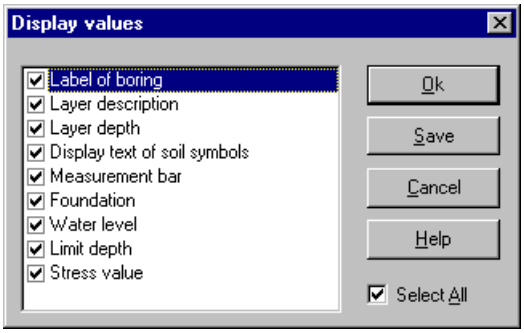


Figure C-93 "Display values" Dialog box

5.8.6 Format Menu

The Format Menu has the following commands:

- Line formats
- Fill color
- Max. width
- Font

Format Menu–"Line formats" command
By the "Line formats" command the color, style and thickness of drawing lines can be defined, Figure C-94. The way a line is drawn depends on the setting of the color and style properties. There are available 15 different colors and 5 styles for line formats.

The following list shows the available lines, which can be formatted:

- Identification box (available in ELPLA-Boring)
- Page Boundary (available in ELPLA-Boring)
- Boring boundary
- Soil layer levels
- Soil symbols
- Groundwater
- Foundation (available in ELPLA-Boring)
- Measurement bar (available in ELPLA-Boring)
- Limit depth (available in ELPLA-Boring)

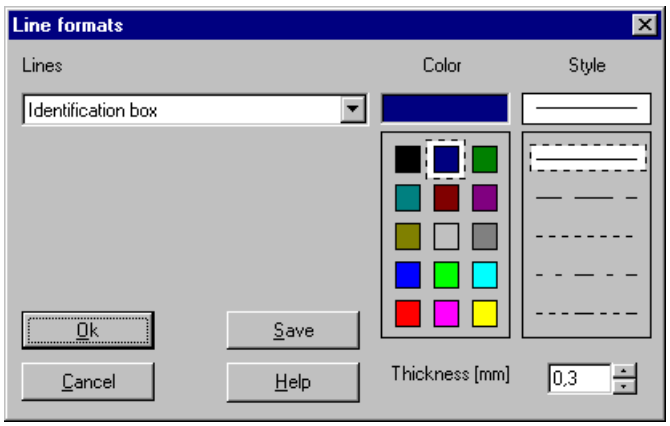


Figure C-94 "Line formats" Dialog box

Format Menu–"Fill color" command

By the "Fill color" command the fill color of drawing can be defined, Figure C-95. The following list shows the available items, which can be filled with a specified color:

- Groundwater
- Foundation (available in ELPLA-Boring)
- Measurement bar (available in ELPLA-Boring)
- Stress due to foundation (available in ELPLA-Boring)
- Stress from neighboring foundations (available in ELPLA-Boring)
- Stress from soil weight (available in ELPLA-Boring)

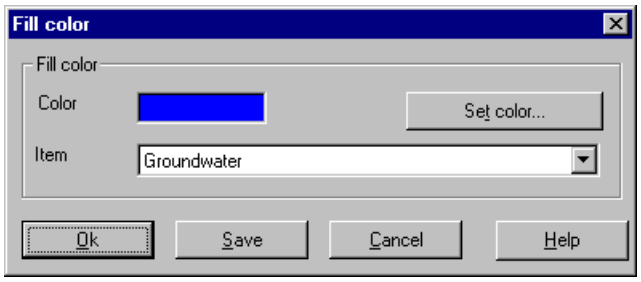


Figure C-95 "Fill color" Dialog box

Format Menu–"Max. width" command

Here the maximum width for the drawing can be defined, Figure C-96.

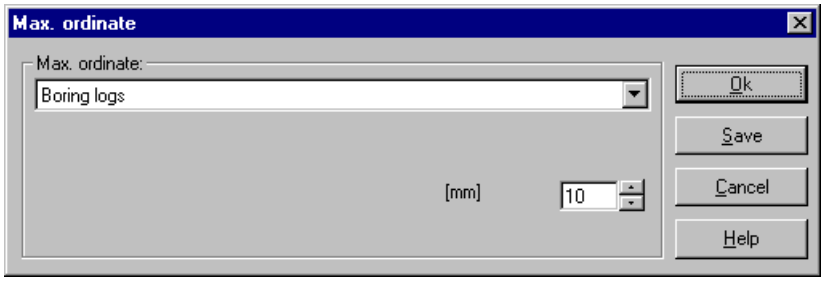


Figure C-96 "Max. width" Dialog box

Format Menu–"Font" command
By this command Font size (Figure C-97) and Font type (Figure C-98) can be defined.

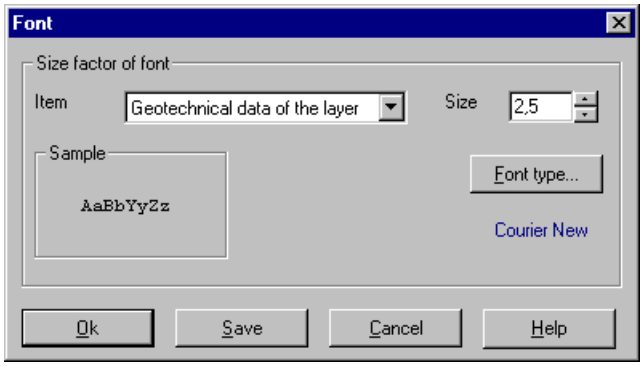


Figure C-97 "Font size" Dialog box

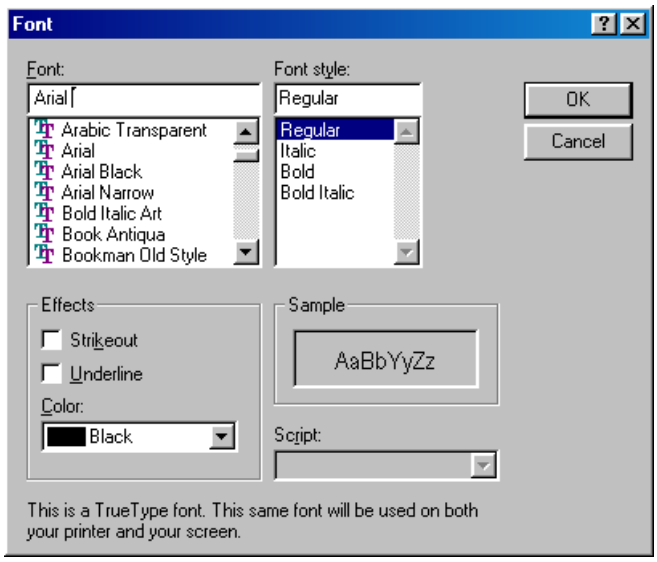


Figure C-98 "Font type" Dialog box

5.8.7 Main data Menu

The Main data Menu has the following command:

- Preferences

Main data Menu–"Preferences" command
When "Preferences" command is chosen, the following Dialog box in Figure C-99 appears.

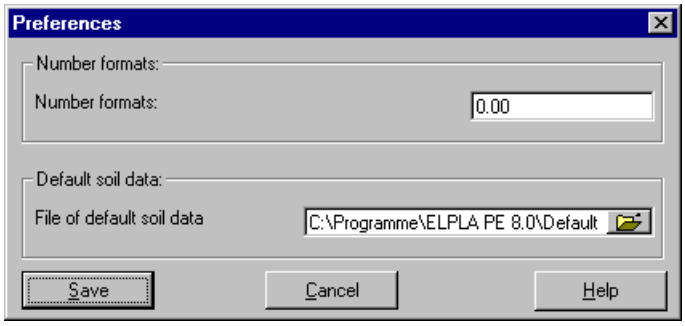


Figure C-99 "Preferences" Dialog box

In the Dialog box of Figure C-99 the user can specify the following items:

- In the Dialog box "Default soil data" the user can specify the file of default soil data. Default soil data are considered when creating new boring logs

- In the Dialog box "Number formats" the user can specify how the numbers of soil parameter values (levels, depths, dimensions, etc.) are displayed or printed

  The following examples describe the number formats:

  Number = 5459.3472
  Format "0.000" gives 5459.347
  Format "0.00" gives 5459.35
  Format "0.0" gives 5459.4
  Format "0" gives 5459
  Format "00E+00" gives 55E+02 (Exponential format)

5.8.8 Window Menu (See paragraph 5.3.8)

5.8.9 Help Menu (See paragraph 5.3.9)

5.9 Data Menu–"Net of soil elements in z-direction" command

The number of elements and element sizes of the net of soil elements in z-direction are defined. This net is required for determining displacements, stresses and strains in soil. When the command is chosen, the following Dialog box of Figure C-100 appears. In order to activate the option of variable element sizes in z-direction, the corresponding item must be unchecked.

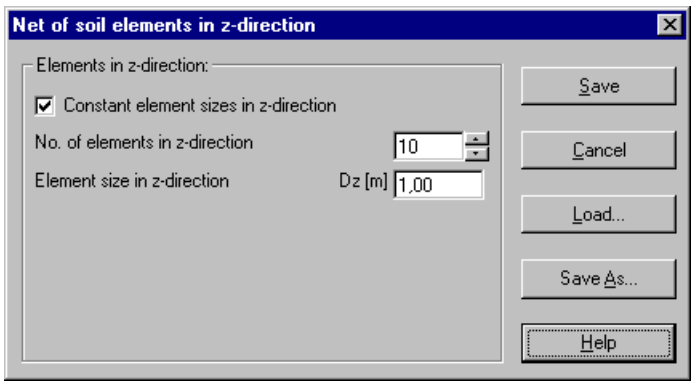


Figure C-100 "Net of soil elements in z-direction" Dialog box

5.10 Data Menu–"Limit depth" command

It is found from experience that the number of layers under foundation depends on the limit depth Zg, where no settlement occurs. The limit depth Zg is defined as the level of which the stress due to the foundation loads reaches a standard ratio Cs of the initial vertical stress due to the self-weight of the soil layers. According to the German Standard DIN 4019 part 1 the recommended standard value of Cs is 0.2.

By the "Limit depth" command limit depth of the soil layers in a boring is defined. When the "Limit depth" command is chosen, the following Dialog box of Figure C-101 appears.

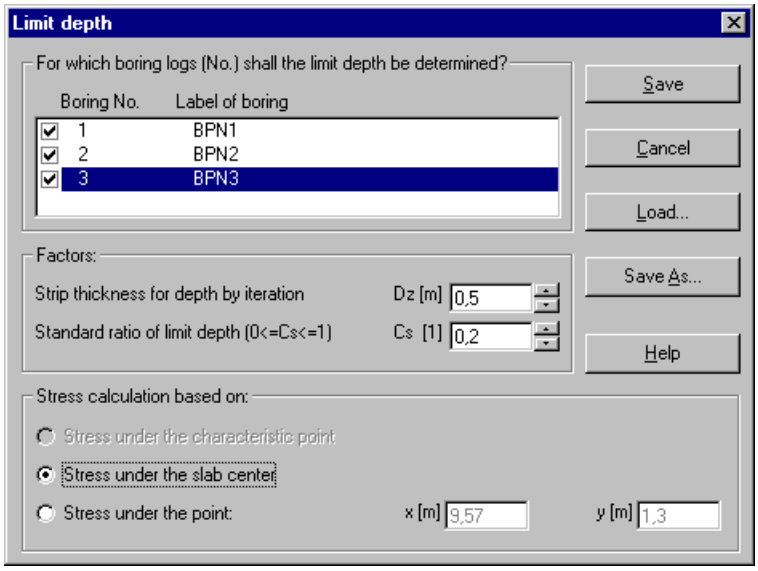


Figure C-101 "Limit depth" Dialog box

The Dialog box of Figure C-101 shows the available boring logs in the project. To determine the limit depth of a boring log, check the box of that boring log.

5.11 Data Menu–"Foundation (Slab) properties" command

When the "Foundation properties" command is chosen, the following embedded program appears, Figure C-102.

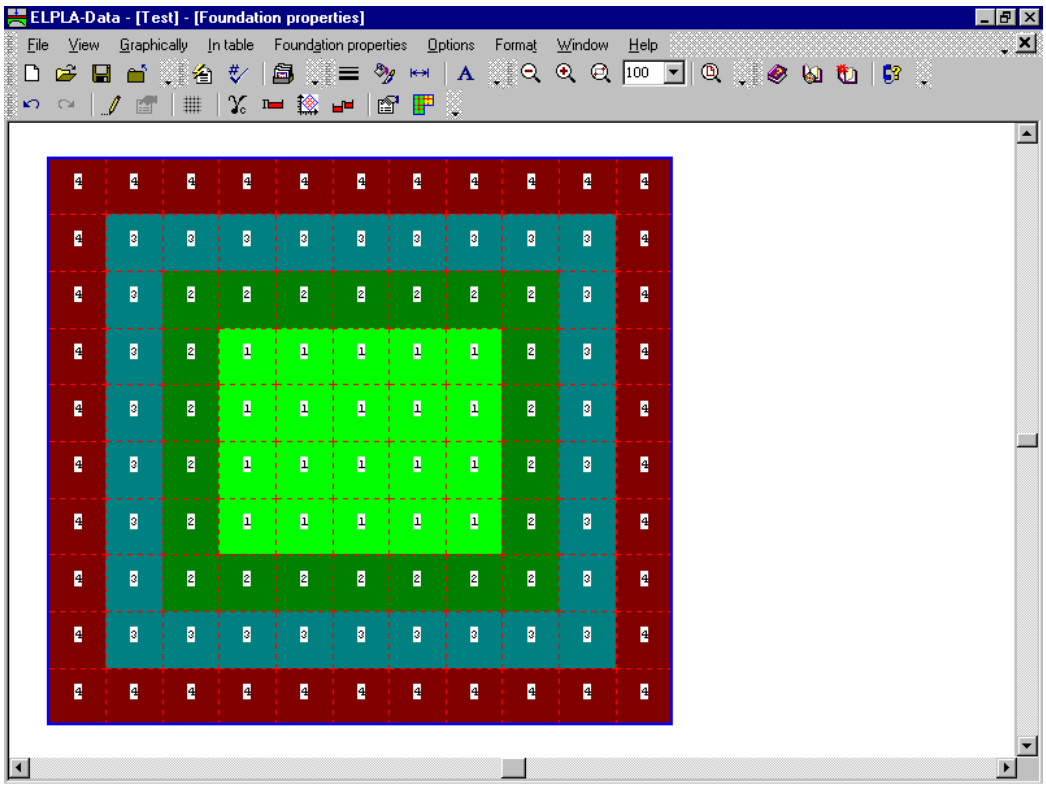


Figure C-102 "Foundation properties" embedded program

The menu head of Figure C-102 contains the following nine commands:

- File
- View
- Graphically
- In table
- Foundation (Slab) properties
- Options
- Format
- Window
- Help

After clicking one of the nine menus other sub-menus or commands become available. The following paragraph presents and describes the nine menu commands and their sub-commands.

5.11.1 File Menu

This menu contains five commands:

- New foundation properties
- Open foundation properties
- Save foundation properties
- Save foundation properties as
- Close foundation properties

File Menu–"New foundation properties" command
Defines new foundation properties

File Menu–"Open foundation properties" command
Opens existing foundation properties-files again on the screen. Then the foundation properties, if desired, can be redefined.

File Menu–"Save foundation properties" command
Saves the active foundation properties under the available name

File Menu–"Save foundation properties as" command
Saves the active foundation properties under a new name

File Menu–"Close foundation properties" command
Closes the Foundation properties-embedded program and returns to ELPLA-Data

5.11.2 View Menu (See paragraph 5.3.2)

5.11.3 Graphically Menu

The menu "Graphically" contains the following commands:

- Undo
- Redo
- Select elements
- Element groups
- Cartesian grid

Graphically Menu–"Undo" command
This command is used to undo the effects of a selected command and return to a previous state.

Graphically Menu–"Redo" command
This command is used to redo the last action of "Undo" command.

Graphically Menu–"Select elements" command
The main function of the command is to provide a method to define the element properties. When the command is chosen, the cursor is changed from an arrow to a cross hair. The desired elements are selected by selecting a group of elements or click by Mouse on each element individually. A group of elements can be selected by holding the left mouse button down at the corner of the region and dragging the mouse until a rectangle encompasses the desired group of elements. When the left mouse button is released, all elements in the rectangle are selected.

Graphically Menu–"Element groups" command
When "Element groups" command is chosen, the following Dialog box in Figure C-103 appears. In this Dialog box define the group No..

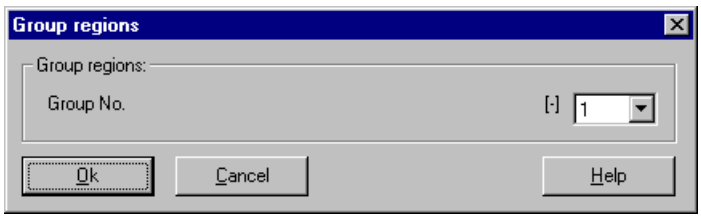


Figure C-103 Defining group No.

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.11.4 In table Menu

The menu "In table" contains the following commands:

- Element groups
- Group regions

In table Menu–"Element groups" command
When "Element groups" command is chosen, the following Dialog box in Figure C-104 appears. In this Dialog box E-Modulus of the slab, *Poisson's* ratio of the slab and slab thickness are defined. Element group is a group that has the same thickness and material.

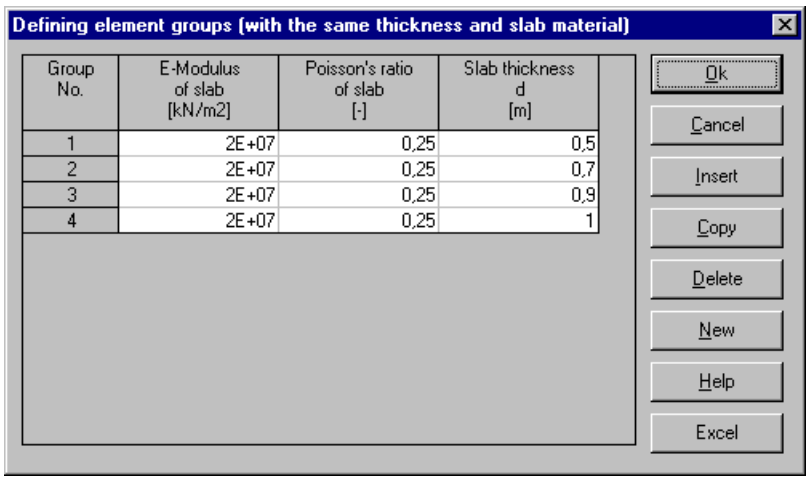


Figure C-104 "Defining element groups" Dialog box

In table Menu–"Group regions" command
By the program ELPLA there are two alternative possibilities to define group regions graphically or numerically (in a table). If it is required to define the group regions in a table, the user must choose "Group regions" command from In table Menu. When "Group regions" command is chosen, the following Dialog box in Figure C-105 appears. Here the element groups of the elements are defined.

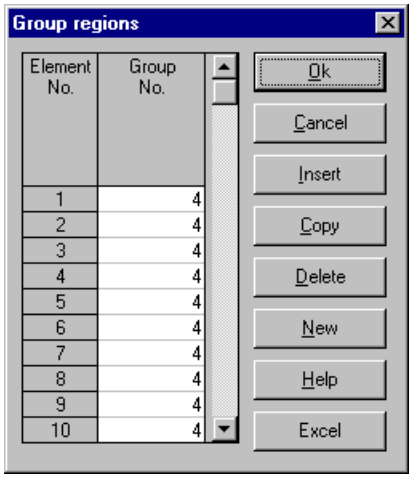


Figure C-105 "Group regions" Dialog box

5.11.5 Foundation (Slab) properties Menu

The menu "Foundation properties" contains the following commands:

- Unit weight of the foundation
- Foundation depth
- Origin coordinates
- Foundation level from a fixed datum

Foundation properties Menu–"Unit weight of the foundation" command
When the command is chosen, the following Dialog box in Figure C-106 appears. To consider the self-weight of the slab in the analysis, define the unit weight of the slab material.

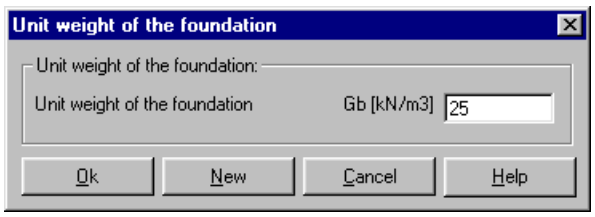


Figure C-106 "Unit weight of the foundation" Dialog box

Foundation properties Menu–"Foundation depth" command

When "Foundation depth" command is chosen, the following Dialog box in Figure C-107 appears.

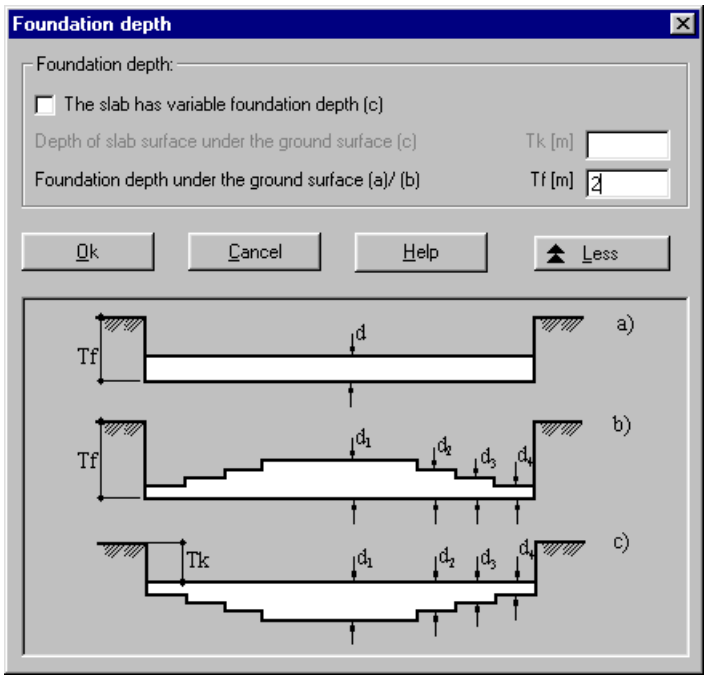


Figure C-107 "Foundation depth" Dialog box

In ELPLA, there are three different possibilities to define the slab thickness:

- Slab thickness is constant for the entire slab. In this case there is only one group, Figure C-108
- Variable slab thickness with constant foundation level, Figure C-109
- Variable slab thickness with variable foundation level, Figure C-110

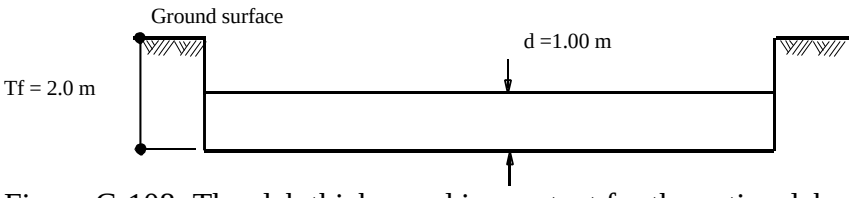


Figure C-108 The slab thickness d is constant for the entire slab

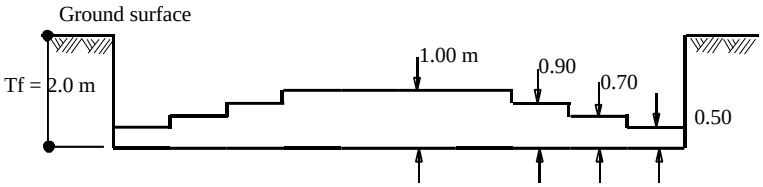


Figure C-109 Variable slab thickness with constant foundation level

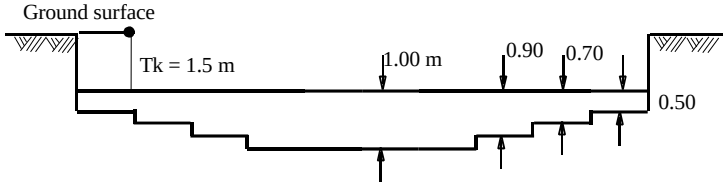


Figure C-110 Variable slab thickness with variable foundation level

In the Dialog box of Figure C-107 define the foundation depth under the ground surface Tf, if the foundation level is constant or define the depth of the slab surface under the ground surface Tk, if the foundation level is variable.

Foundation properties Menu–"Origin coordinates" command

By analysis of a system of slab foundations or study the effect of neighboring foundations, every slab is defined in a global system through the origin coordinates x_o, y_o and angle $\exists_o$ between the x-axes of global and local systems.

When "Origin coordinates" command is chosen, the following Dialog box in Figure C-112 appears. In this Dialog box define the origin coordinates x_o, y_o and angle β_o between the x-axes of global and local systems.

Note

In the analysis of an isolated slab without consideration of neighboring foundations, the origin coordinates play no roles in the analysis.

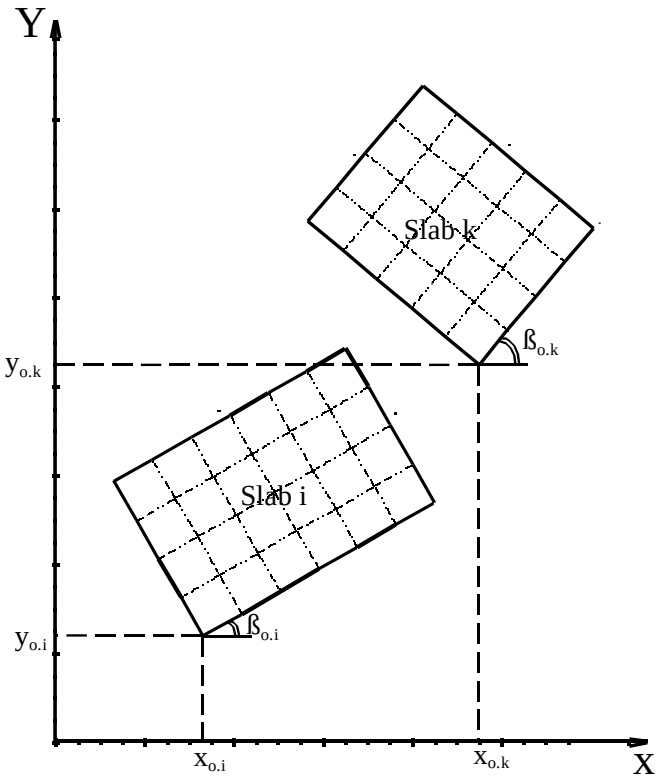


Figure C-111 Geometrical plan by studding the influence of slab k on the slab i

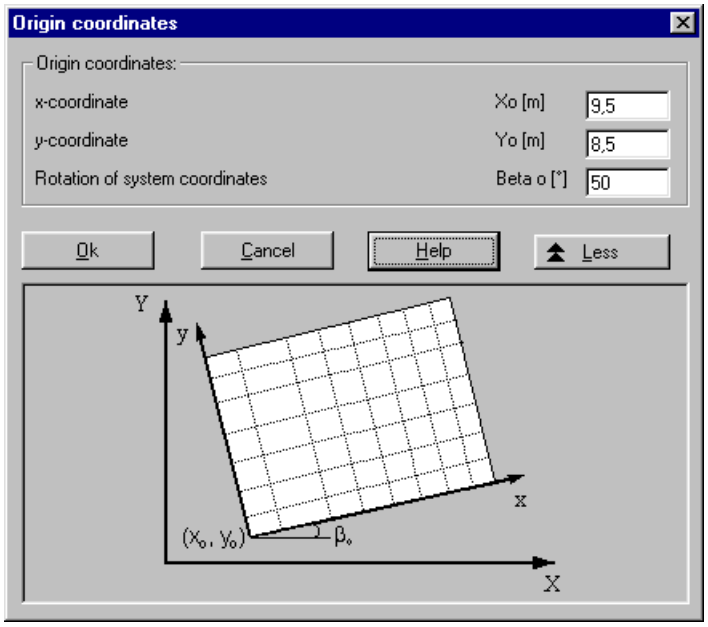


Figure C-112 "Origin coordinates" Dialog box

Foundation properties Menu–"Foundation level from a fixed datum" command

Sometimes, when determining the influence of the neighboring slabs or the interaction of a system of slabs, the slabs are constructed with variable foundation levels, Figure C-113. This can be considered through the command "Foundation level from a fixed datum" as shown in Figure C-114. In this case, the foundation levels of the slabs must be related to a specified datum H_m.

Note
In the analysis of an isolated slab without consideration of neighboring foundations, the levels H_m play no roles in the analysis.

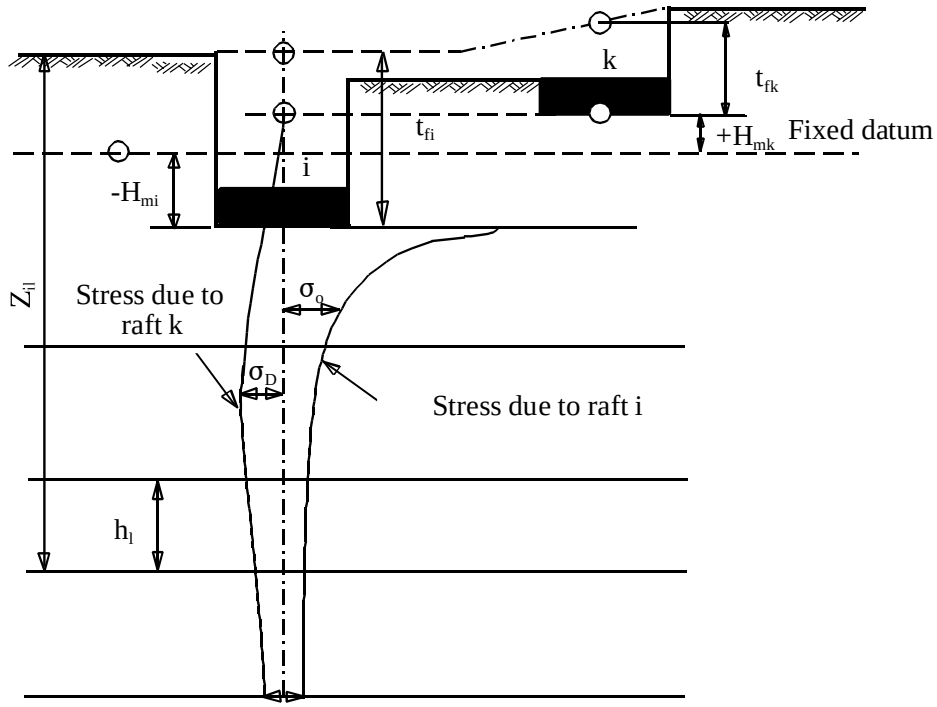


Figure C-113 Influence of slab k on the slab i

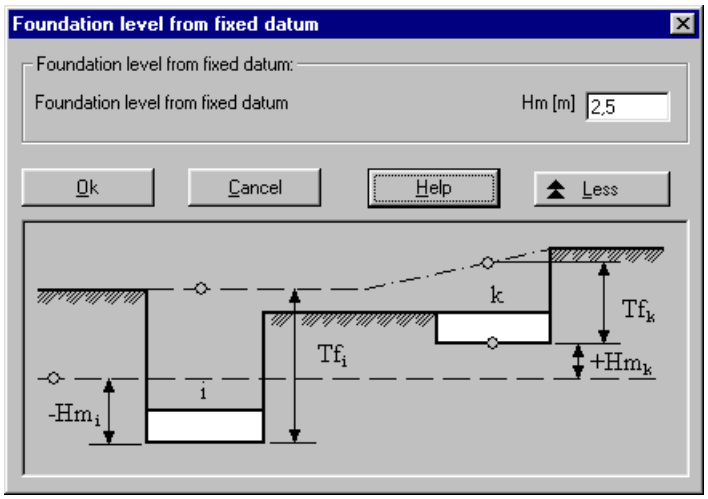


Figure C-114 "Foundation level from a fixed datum" Dialog box

5.11.6 Options Menu (See paragraph 5.3.6)

5.11.7 Format Menu (See paragraph 5.3.7)

5.11.8 Window Menu (See paragraph 5.3.8)

5.11.9 Help Menu (See paragraph 5.3.9)

5.12 Data Menu–"Reinforcement data" command

The design of the slab for flexure moment and punching shear can be carried out according to the design codes EC 2, DIN 1045, ACI and ECP (working stress and limit state design methods). When "Reinforcement data" command is chosen, the following Dialog box in Figure C-115 appears. In this Dialog box define design code, concrete grade, steel grade and concrete covers.

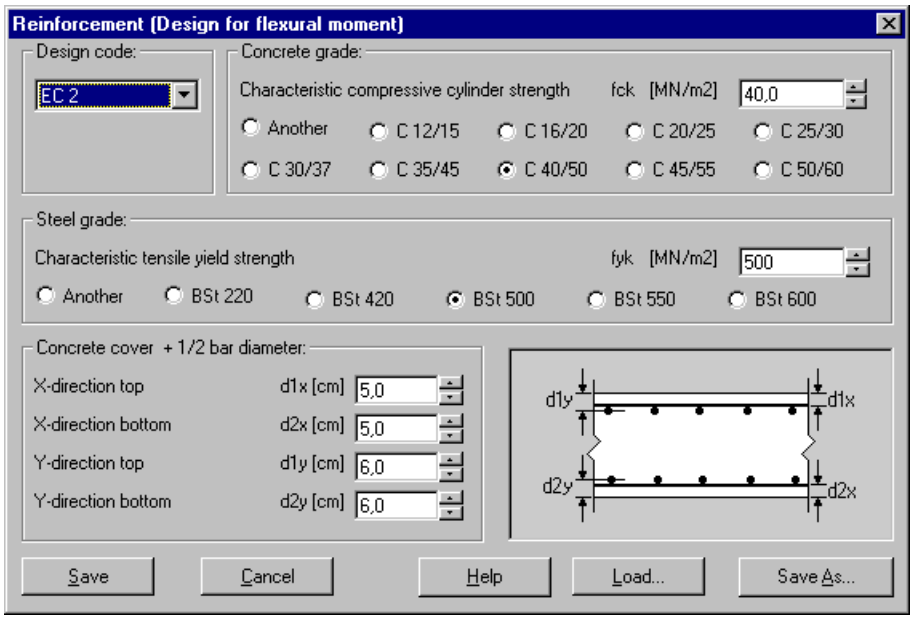


Figure C-115 "Reinforcement" dialog box

5.13 Data Menu–"Boring fields" command

If the subsoil under the slab foundation is characterized by more than one boring, the variation in the subsoil in the three directions must be taken into consideration according to ELPLA-Theory. By the "Boring fields" command the boring fields can be defined. It is also possible to define the boring fields graphically, which makes the definition of the boring fields very easy, or numerically (in a table). When the "Boring fields" command is chosen, the following embedded program appears, Figure C-116.

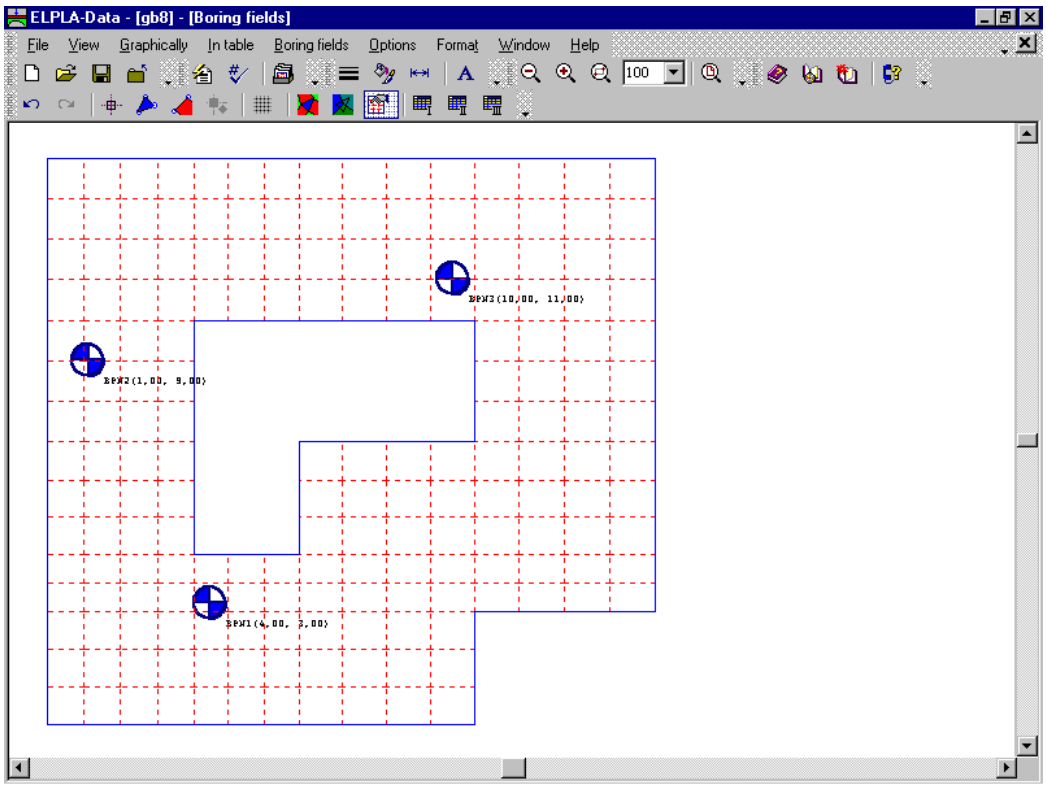


Figure C-116 "Boring fields" embedded program

The menu head of Figure C-116 contains the following nine commands:

- File
- View
- Graphically
- In table
- Boring fields
- Options
- Format
- Window
- Help

After clicking one of the nine commands (options) other sub-commands or options become available. The following paragraph presents and describes the nine menu commands and their sub-commands.

5.13.1 File Menu

This menu contains five commands:

- New boring fields
- Open boring fields
- Save boring fields
- Save boring fields as
- Close boring fields

File Menu–"New boring fields" command
Defines new boring fields

File Menu–"Open boring fields" command
Opens existing boring fields-file again. Then the boring fields, if desired, can be redefined.

File Menu–"Save boring fields" command
Saves the active boring fields under the available name

File Menu–"Save boring fields as" command
Saves the active boring fields under a new name

File Menu–"Close boring fields" command
Closes the boring fields-embedded program and returns to ELPLA-Data

5.13.2 View Menu (See paragraph 5.3.2)

5.13.3 Graphically Menu

This menu contains the following commands:

- Undo
- Redo
- Select nodes
- Zone type I
- Zone type II
- Zone type III
- Cartesian grid

Graphically Menu–"Undo" command
This command is used to undo the effects of a selected command and return to a previous state.

Graphically Menu–"Redo" command
This command is used to redo the last action of "Undo" command.

Graphically Menu–"Select nodes" command
The main function of "Select nodes" command is to provide a method for defining the nodes of Zone type III. Soil properties for nodes lying in this zone are defined according to a specified boring by the user. Zone type III contains also the nodes that are outside the zones I and II.

When "Select nodes" command is chosen, the cursor is changed from an arrow to a cross hair. In this case, "Zone type I" and also "Zone type II" will be disabled. The desired nodes are selected by clicking on each node individually or selecting a group of nodes. A group of nodes can be selected by holding the left mouse button down at the corner of the region and dragging the mouse until a rectangle encompasses the desired group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

Graphically Menu–"Zone type I" command

Zone type I is defined as a triangular region; such a region is confined by three boring logs. The flexibility coefficient or the modulus of subgrade reaction for a node that lies at a triangular region, can be obtained through interpolation among the three values of the parameters of these three boring logs.

When "Zone type I" command is chosen, the cursor is changed from an arrow to a cross hair. The desired triangular region of Zone type I is selected by clicking on the three borings that confine it, Figure C-117.

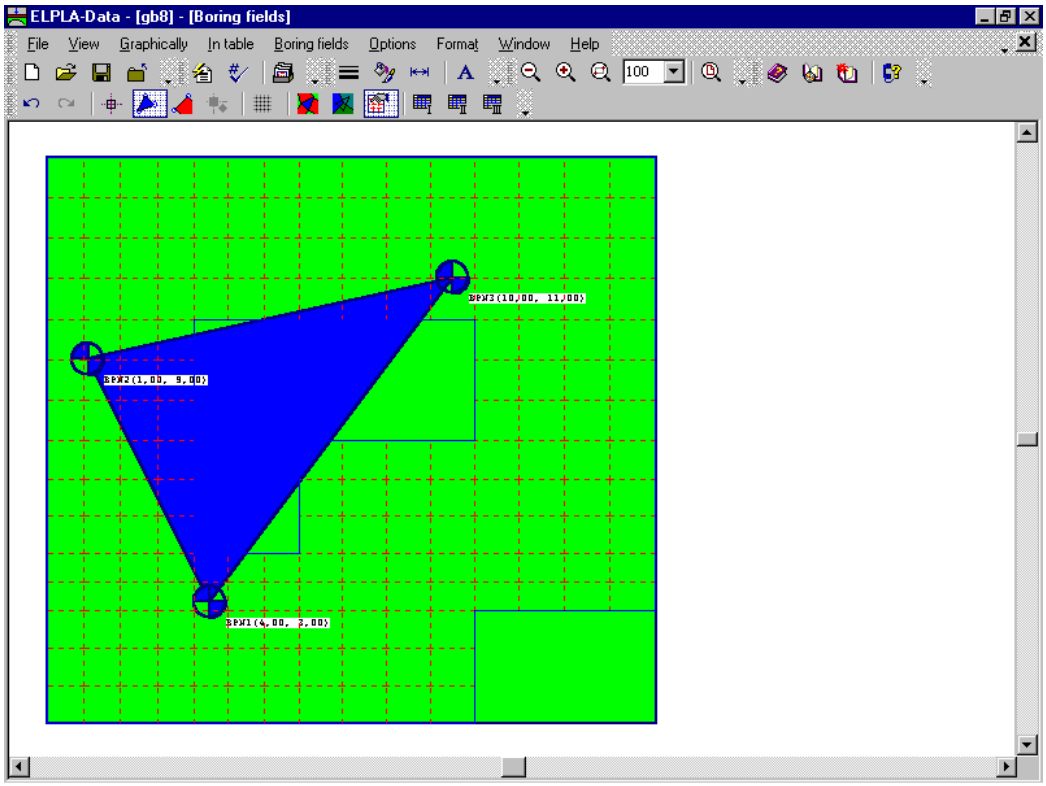


Figure C-117 Zone type I

Graphically Menu–"Zone type II" command

Zone type II is a region that is confined by one or more sides of the foundation and two borings. The flexibility coefficient or the modulus of subgrade reaction for a node in this region may be obtained by assuming a linear interpolation between the values of the parameters of these two boring logs.

When "Zone type II" command is chosen, the cursor is changed from an arrow to a cross hair. The desired region of Zone type II is selected by clicking on the two borings that confine it, then clicking on any point inside that region, Figure C-118.

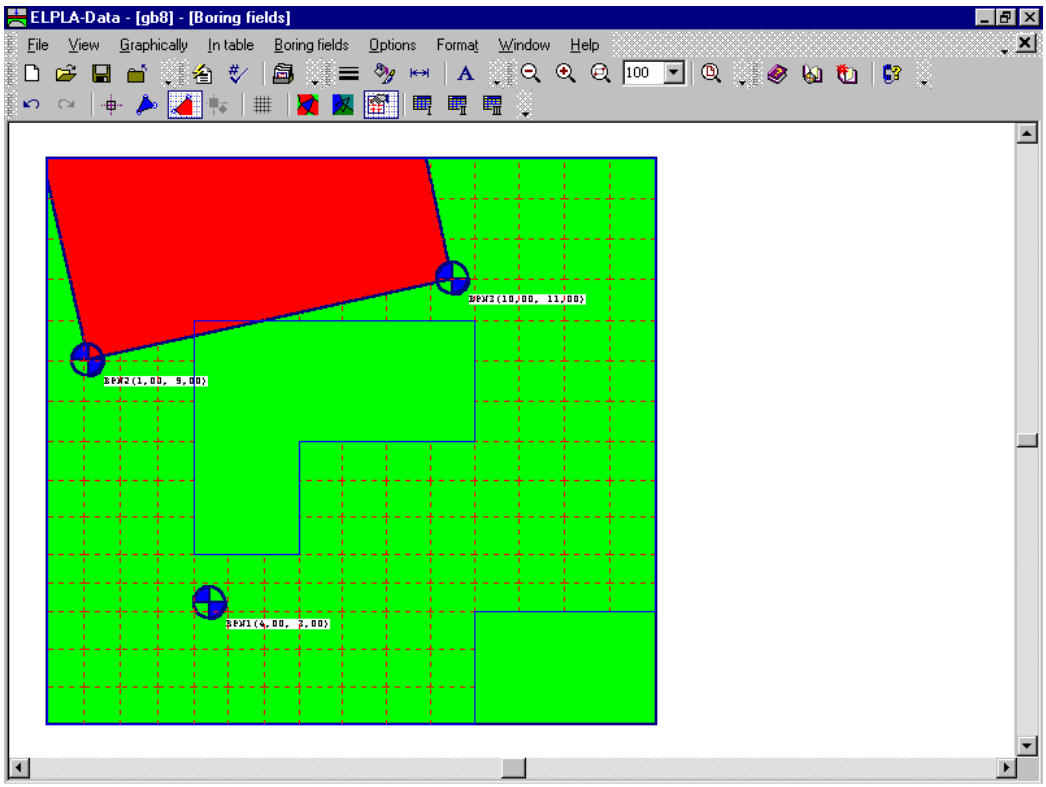


Figure C-118 Zone type II

Graphically Menu–"Zone type III" command
When "Zone type III" command is chosen, the following Dialog box of Figure C-119 appears to define the boring of the selected nodes.

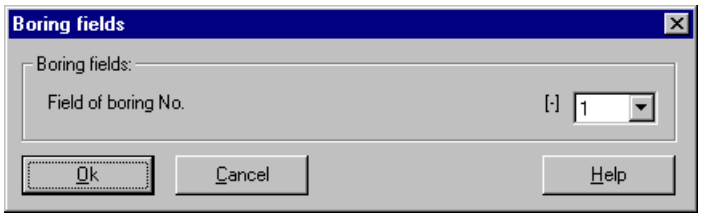


Figure C-119 "Definition of boring fields" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.13.4 In table Menu

This menu contains the following commands:

- Zone type I
- Zone type II
- Zone type III

In table Menu–"Zone type I" command
When the "Zone type I" command is chosen, the following Dialog box appears. In the Dialog box of Figure C-120 each region of Zone type I is defined by the three borings that confine it.

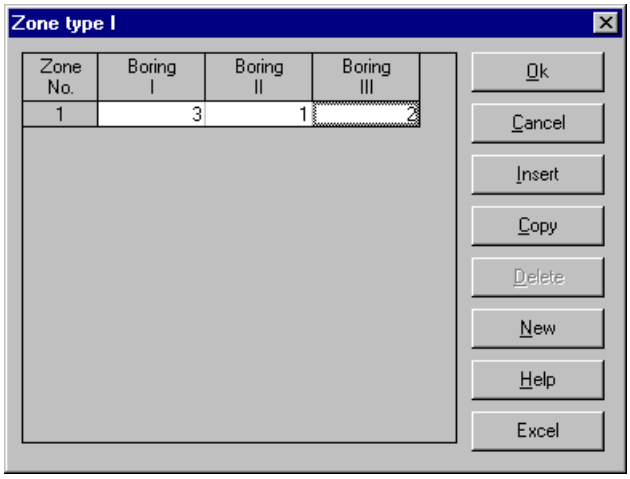


Figure C-120 "Zone type I" Dialog box

In table Menu–"Zone type II" command
When the "Zone type II" command is chosen, the following Dialog box appears, Figure C-121. In this Dialog box each region of Zone type II is defined by the two borings that confine it as well as a corner of the foundation lies inside it. The corner of the foundation can be described as follows:

- Corner No. 1: bottom left corner of the foundation
- Corner No. 2: bottom right corner of the foundation
- Corner No. 3: top left corner of the foundation
- Corner No. 4: top right corner of the foundation

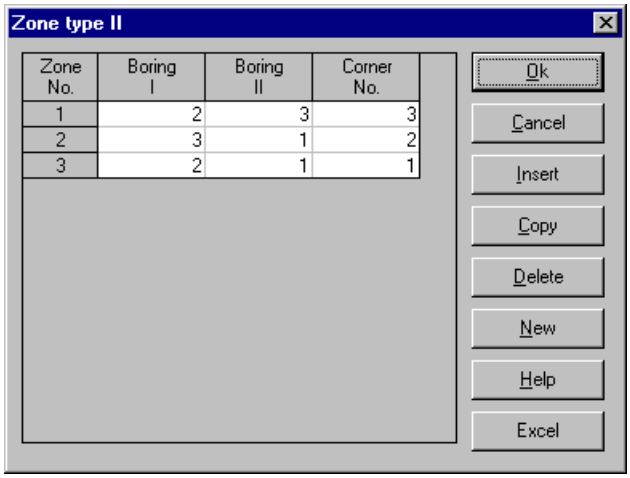


Figure C-121 "Zone type II" Dialog box

In table Menu–"Zone type III" command
When the "Zone type III" command is chosen, the following Dialog box appears. In the Dialog box of Figure C-122 define the borings of nodes that are not considered in Zone type I or II.

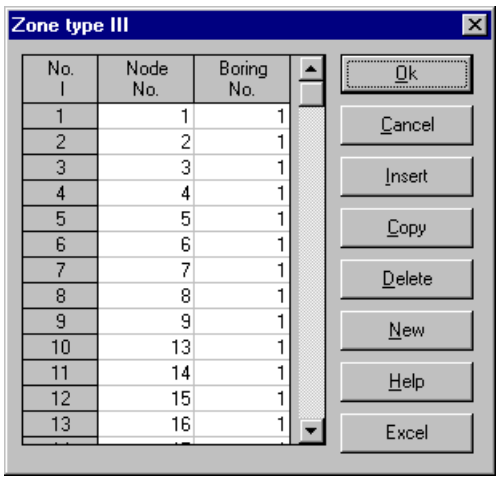


Figure C-122 "Zone type III" Dialog box

5.13.5 Boring fields Menu

Using the Boring fields Menu allows the user to choose one of the three different possibilities to determine the boring fields. This menu contains the following command:

- Interpolation method
- Subarea method
- Hand-Division of boring logs to nodes

Boring fields Menu–"Interpolation method" command
Interpolation method is an accurate one to determine the three-dimensional flexibility coefficient or variable modulus of subgrade reaction for arbitrary foundation on irregular subsoil. The "Interpolation method" command allows the user to define the interpolation zones I and II, and also Zone type III automatically. When the "Interpolation method" command is chosen, the following Figure C-123 appears.

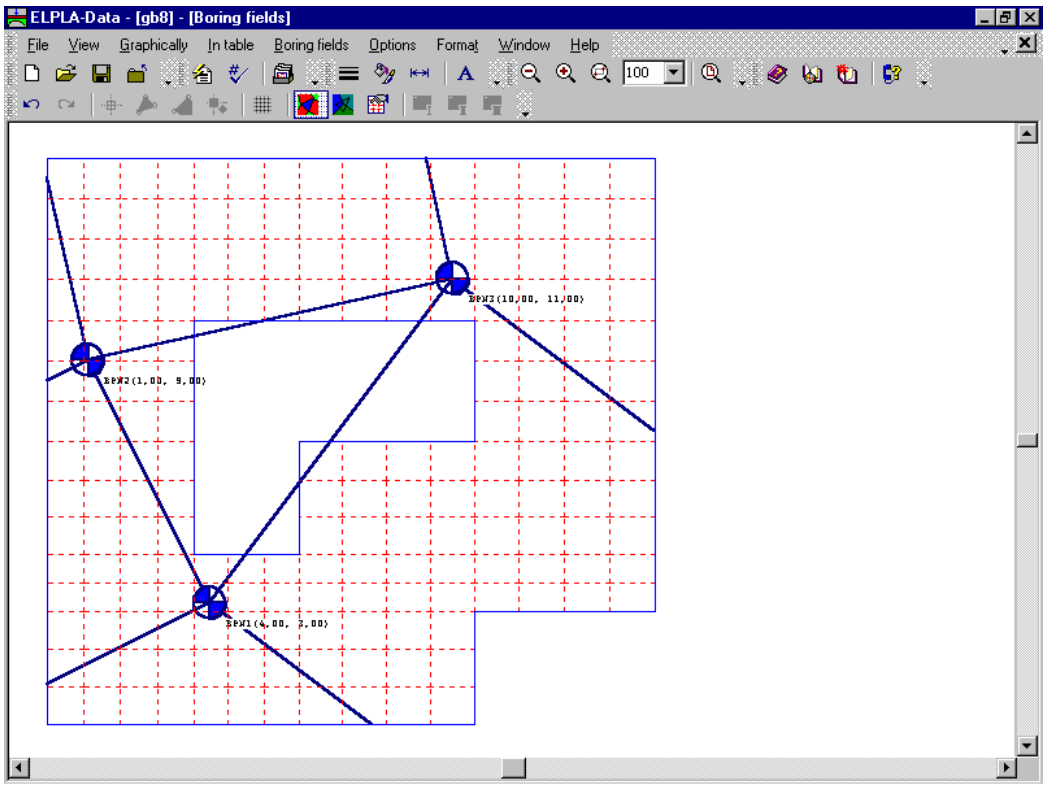


Figure C-123 Boring locations and region types by Interpolation method

Boring fields Menu–"Subarea method" command

In the Subarea method the entire foundation area is divided into subareas. Each subarea is corresponding to one of the boring logs. This method may be used if there is a little difference in soil properties of boring logs. The "Subarea method" command allows the user to define the subareas automatically. When the command is chosen, the following Figure C-124 appears.

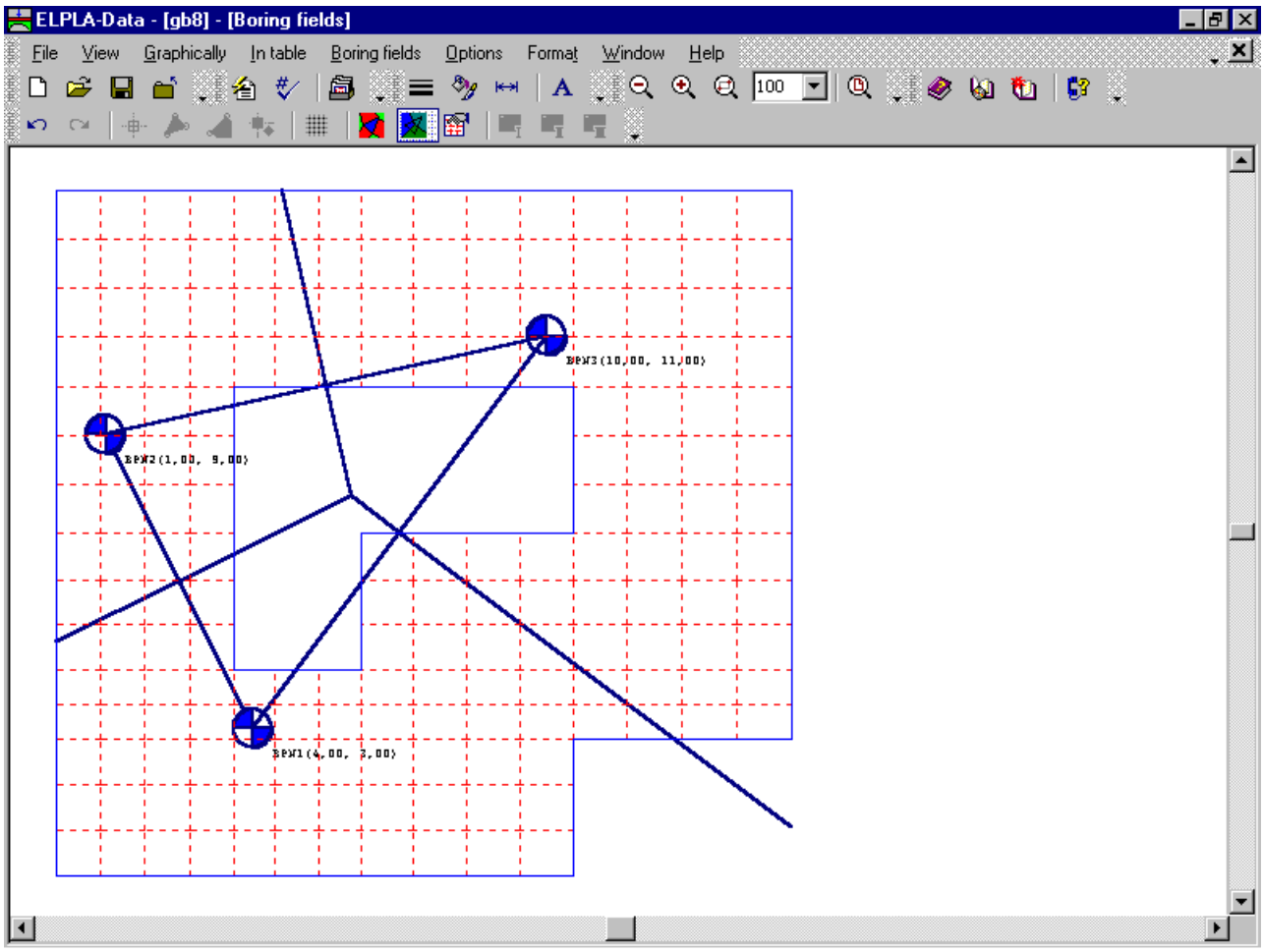


Figure C-124 Boring locations and subareas by Subarea method

Boring fields Menu–"Hand-Division of boring logs to nodes" command
This command allows the user to define the zone types I, II, III graphically by using the commands of Graphically Menu or manually through In table Menu.

5.13.6 Options Menu (See paragraph 5.3.6)

5.13.7 Format Menu (See paragraph 5.3.7)

5.13.8 Window Menu (See paragraph 5.3.8)

5.13.9 Help Menu (See paragraph 5.3.9)

Numbering of borings
To determine the subareas or the interpolation zones automatically by ELPLA, the boring No. 1 must be always a central position for the other borings, if the subsoil is characterized through more than three borings. Figure C-125 to C-129 show five borings defining the subsoil under a raft. It can be carried out different arrangements of interpolation zones using the above role. Another arrangement for the boring numbering may cause some errors. In this case, the Hand-Division of the boring logs to the nodes by the user must be used.

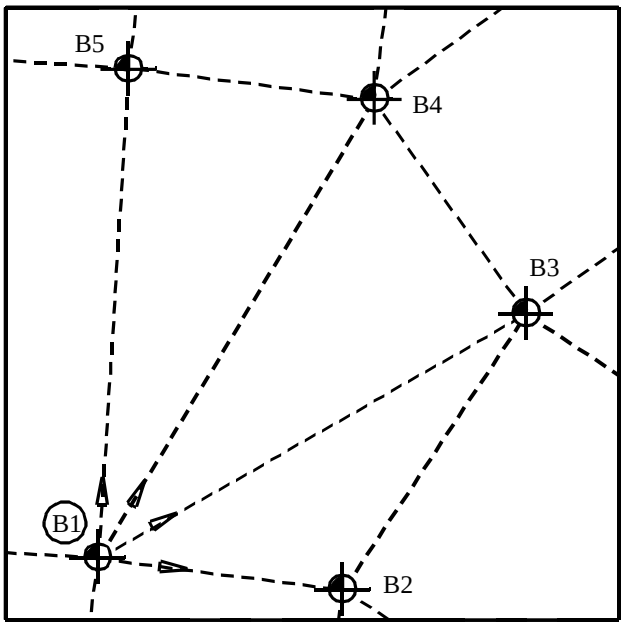


Figure C-125 Division of interpolation zones (numbering of boring a)

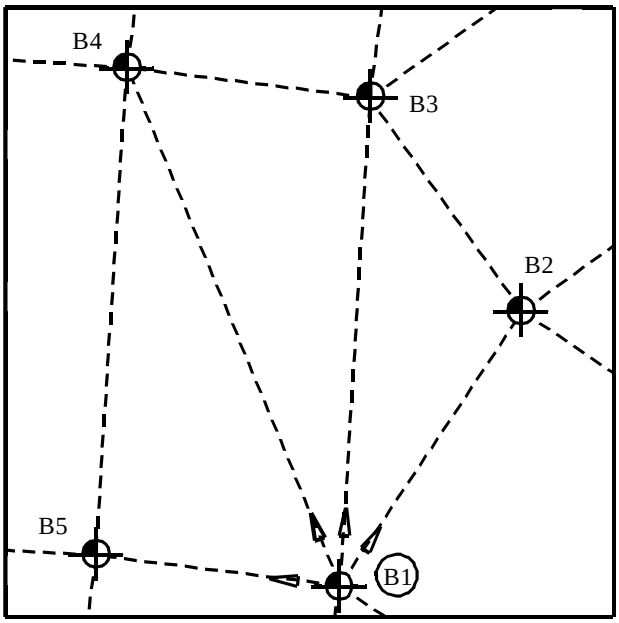


Figure C-126 Division of interpolation zones (numbering of boring b)

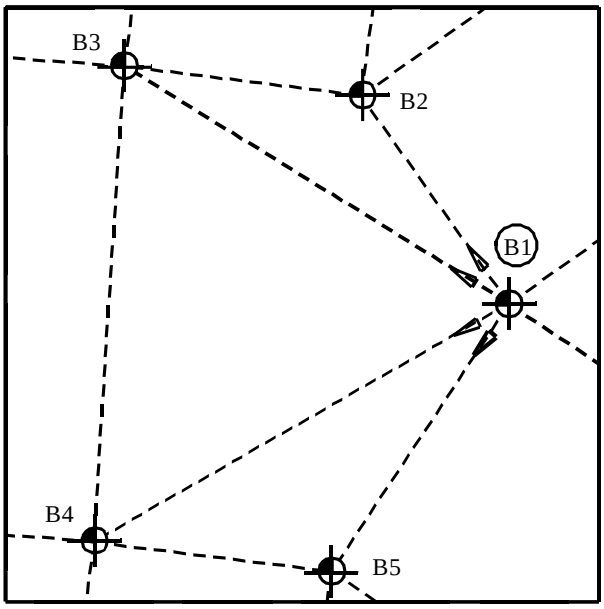


Figure C-127 Division of interpolation zones (numbering of boring c)

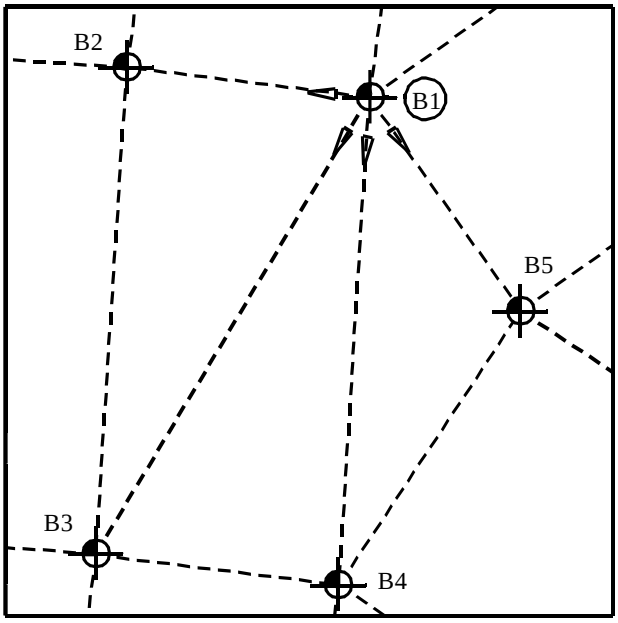


Figure C-128 Division of interpolation zones (numbering of boring d)

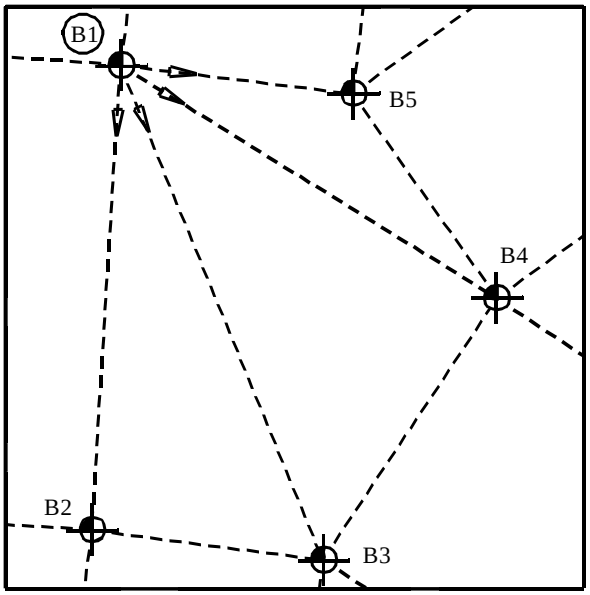


Figure C-129 Division of interpolation zones (numbering of boring e)

5.14 Data Menu–"Loads" command

By the "Loads" command the loads on the slab such as point loads, line loads, distributed loads or moments at any position independently on the FE-Net are defined. When this command is chosen, the following embedded program appears, Figure C-130.

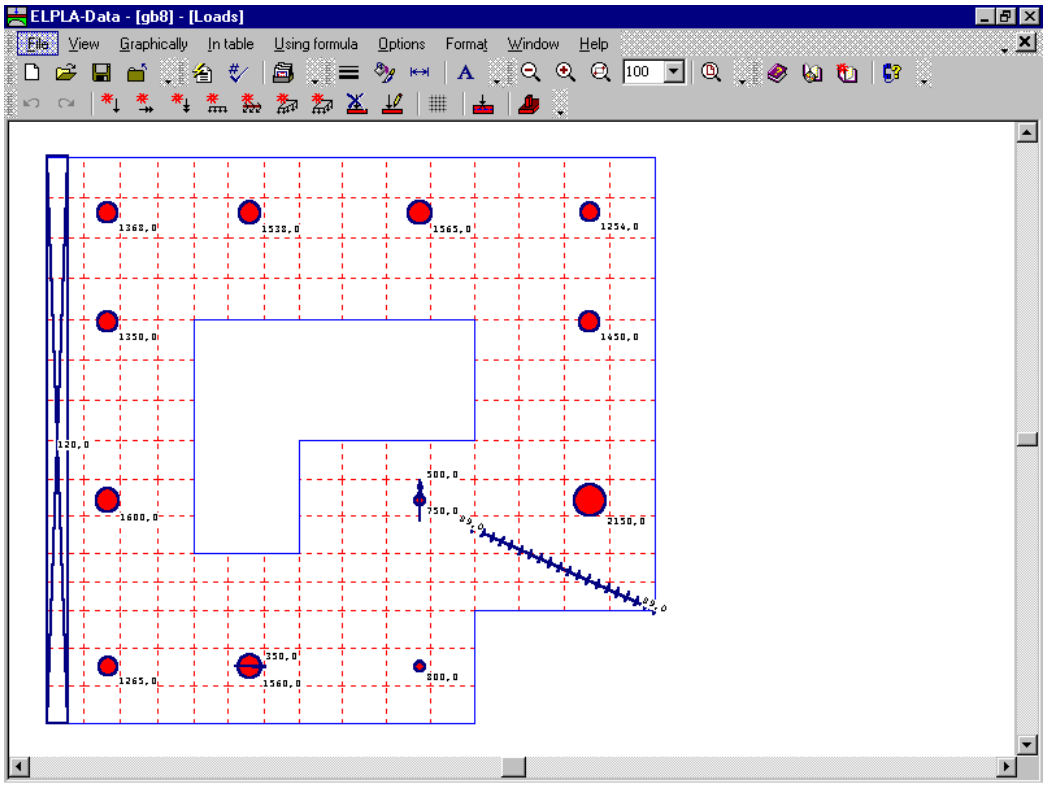


Figure C-130 "Loads" embedded program

The menu head of Figure C-130 contains the following nine commands:

- File
- View
- Graphically
- In table
- Using formula
- Options
- Format
- Window
- Help

After clicking one of the nine commands (options) other sub-commands or options become available. The nine menu commands and their sub-commands are presented and described in the following paragraphs.

5.14.1 File Menu

This menu contains five commands:

- New loads
- Open loads
- Save loads
- Save loads as
- Close loads

File Menu–"New loads" command
Defines new loads

File Menu–"Open loads" command
Opens existing load file again on the screen. Then the loads, if desired, can be redefined.

File Menu–"Save loads" command
Saves the active loads under the available name

File Menu–"Save loads as" command
Saves the active loads under a new name

File Menu–"Close loads" command
Closes the loads-embedded program and returns to ELPLA-Data

5.14.2 View Menu (See paragraph 5.3.2)

5.14.3 Graphically Menu

This menu contains the following commands:

- Undo
- Redo
- Point loads
- Moments M_x
- Moments M_y
- Line loads
- Line moments
- Distributed loads (Polygon)
- Distributed loads (Rectangle)
- Remove loads
- Edit loads
- Cartesian grid

Graphically Menu–"Undo" command
This command is used to undo the effects of a selected command and return to a previous state.

Graphically Menu–"Redo" command
This command is used to redo the last action of "Undo" command.

Graphically Menu–"Point loads" command
By the "Point loads" command the vertical concentrated loads can be defined at any position (x, y). The position of the load is independent on the FE-Net. 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 (slab). When the screen is clicked, the following Dialog box of Figure C-131 appears and the following data is required to define:

- Column type
- Load value
- Position of the load (x, y) in the local coordinates

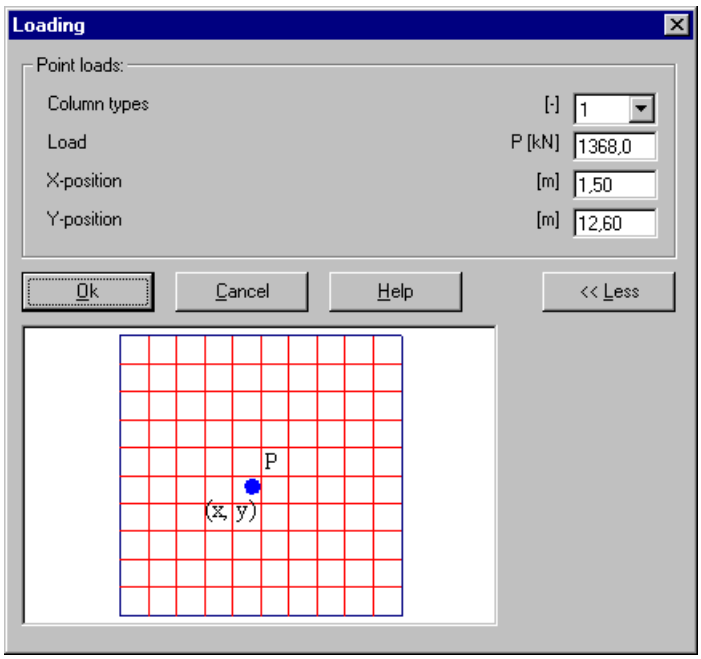


Figure C-131 "Point loads P" Dialog box

Graphically Menu–"Moments M_x" command

By this command the applied moment about x-axis can be defined at any position (x, y). The position of applied moment is independent on the FE-Net. When the command is chosen, the cursor is changed from an arrow to a cross hair. Then, the moment can be defined by clicking on the screen. When the screen is clicked, the following Dialog box of Figure C-132 appears and the following data is required to define:

- Moment value
- Position of the moment (x, y) in the local coordinates

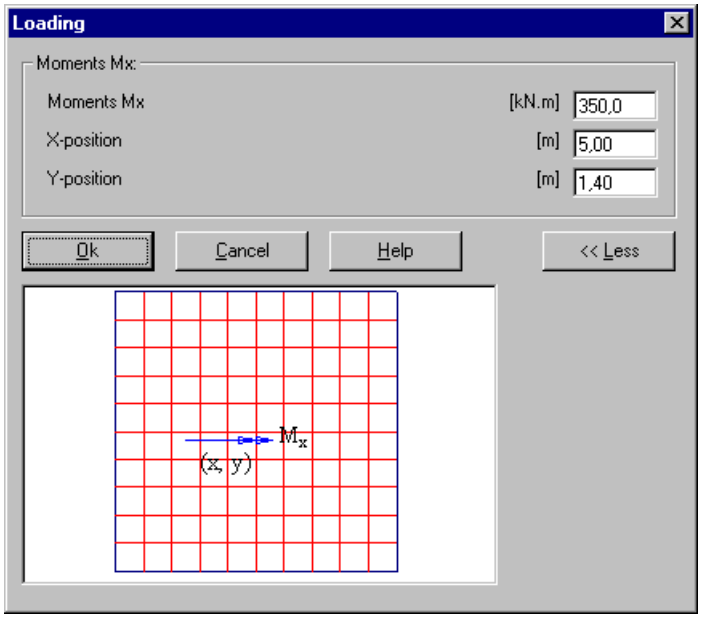


Figure C-132 "Moments M_x" Dialog box

Graphically Menu–"Moments M_y" command

By the "Moments M_y" command the applied moment about y-axis can be defined at any position (x, y). The position of applied moment is independent on the FE-Net. When "Moments M_y" command is chosen, the cursor is changed from an arrow to a cross hair. Then, the moment can be defined by clicking on the screen. When the screen is clicked, the following Dialog box of Figure C-133 appears and the following data is required to define:

- Moment value
- Position of the moment (x, y) in the local coordinates

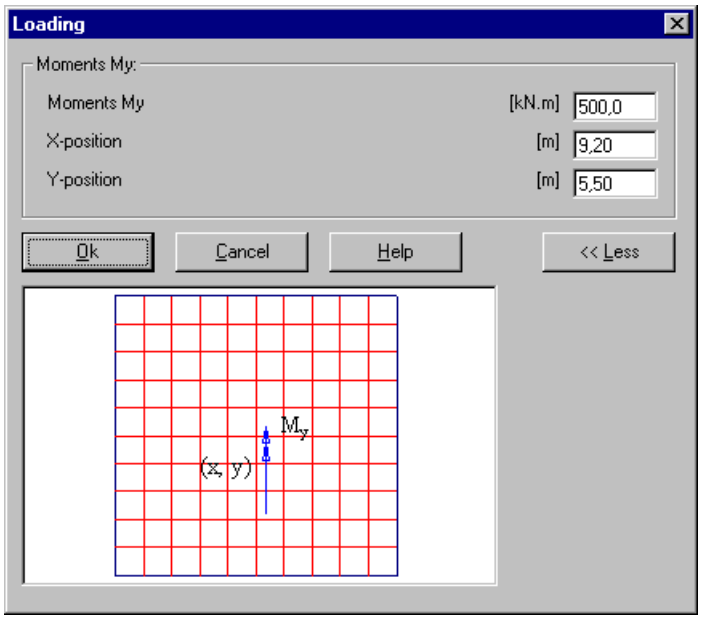


Figure C-133 "Moments M_y" Dialog box

Graphically Menu–"Line loads" command

By the "Line loads" command the applied load per Meter can be defined at any position from point (x_1, y_1) to point (x_2, y_2). The position of the applied load is independent on the FE-Net. When "Line loads" command is chosen, the cursor is changed from an arrow to a cross hair. Then, the line load can be defined by clicking on the starting point of the line load. As the cursor is moved, a black line appears; indicating a line load is being defined. When the ending point is clicked, the following Dialog box of Figure C-134 appears and the following data is required to define:

- Values of start and end points of the load
- Position of starting the line load (x_1, y_1) in the local coordinates
- Position of ending the line load (x_2, y_2) in the local coordinates

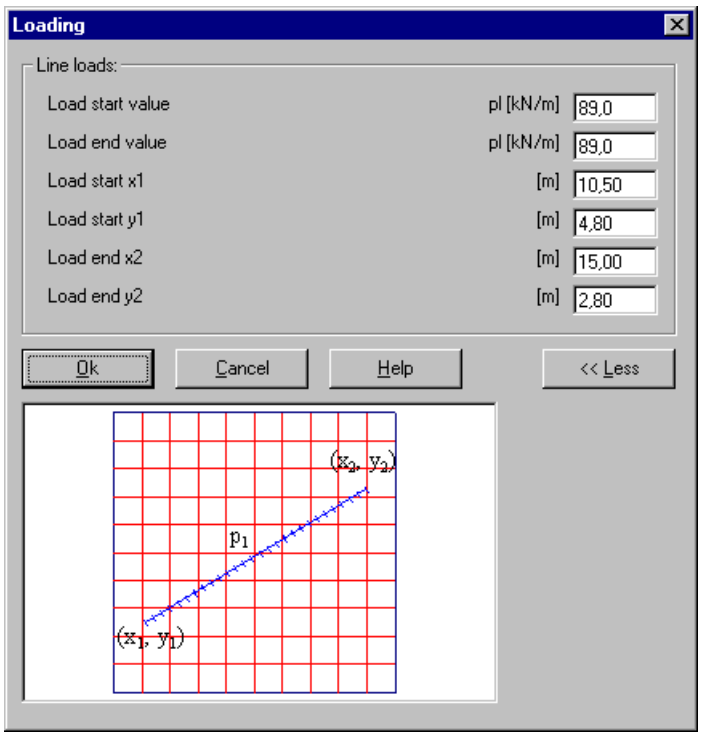


Figure C-134 "Line loads pl" Dialog box

Graphically Menu–"Line moments" command

By the "Line moments" command the applied moment can be defined at any position from point (x_1, y_1) to point (x_2, y_2). The position of the applied moment is independent on the FE-Net. When "Line moments" command is chosen, the cursor is changed from an arrow to a cross hair. Then, the line moment can be defined by clicking on the starting point of the line moment. As the cursor is moved, a black line appears; indicating a line moment is being defined. When the ending point is clicked, the following Dialog box of Figure C-135 appears and the following data is required to define:

- Values of start and end points of the moment
- Position of starting the line moment (x_1, y_1) in the local coordinates
- Position of ending the line moment (x_2, y_2) in the local coordinates

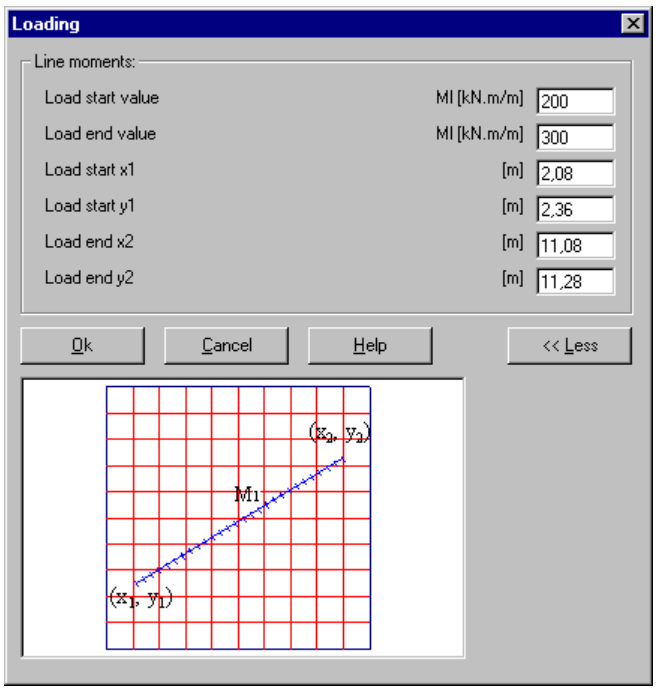


Figure C-135 "Line moments Ml" Dialog box

Graphically Menu–"Distributed loads (Polygon)" command

By this command the applied load per Meter square can be defined for a polygon load. The position of the applied load is independent on the FE-Net. When this command is chosen, the cursor is changed from an arrow to a cross hair. To define a polygon load, click the polygon corners by Mouse to define the polygon. When double-click by Mouse on the last corner of the polygon, the following Dialog box of Figure C-136 appears and the following data is required to define:

- Load intensity at corners
- Corners position of the polygon

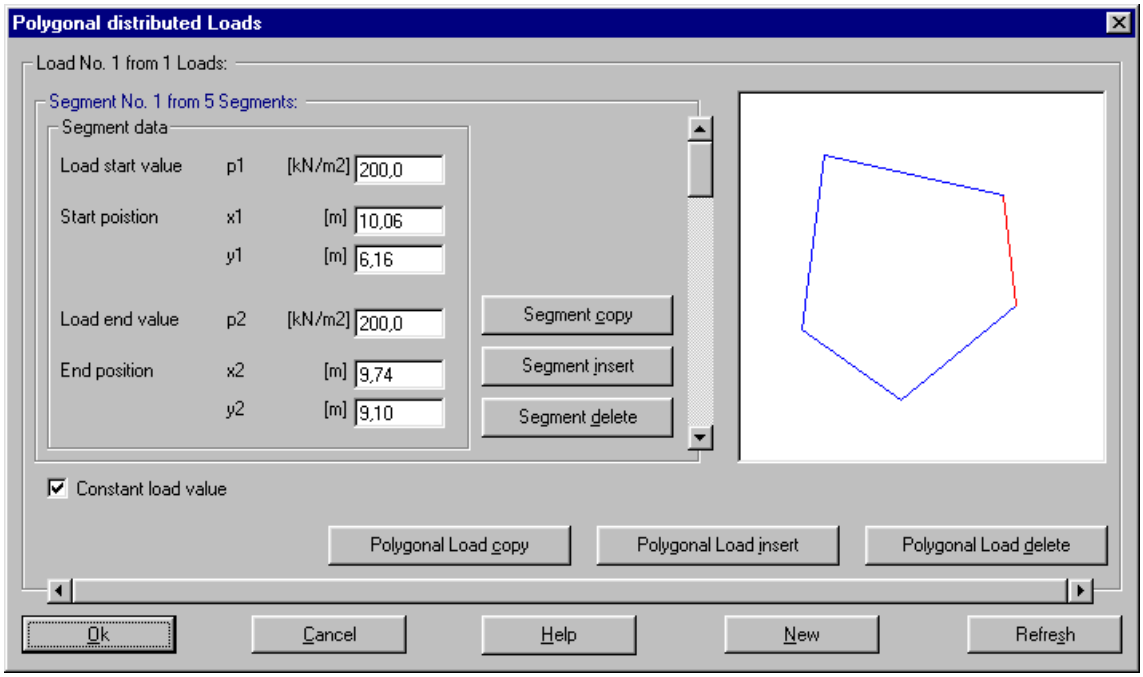


Figure C-136 "Distributed loads p" Dialog box

Graphically Menu–"Distributed loads (Rectangle)" command
By this command the applied load per Meter square can be defined at any diagonal position (x_1, y_1) to (x_2, y_2). The position of the applied load is independent on the FE-Net. 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 of Figure C-137 appears and the following data is required to define:

- Distributed load value
- Position of the starting point (x_1, y_1) in the local coordinates
- Position of the ending point (x_2, y_2) in the local coordinates

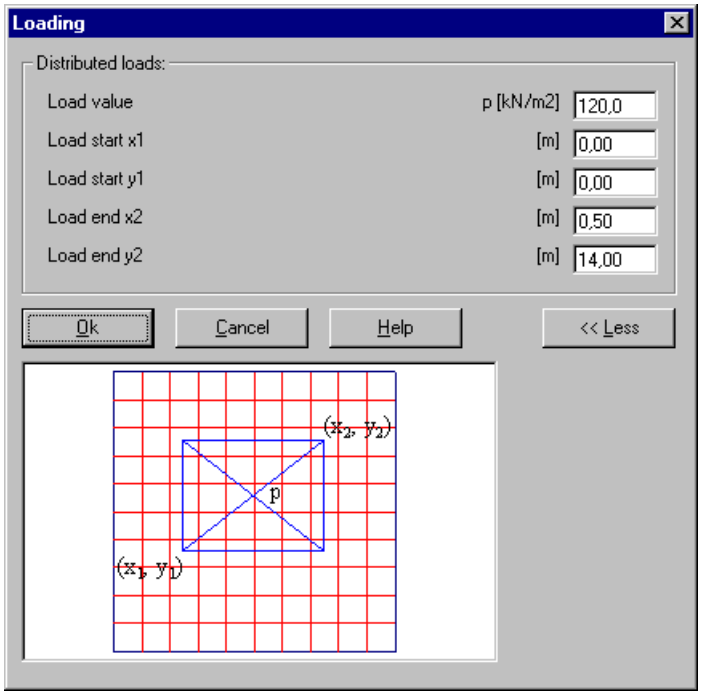


Figure C-137 "Distributed loads (Rectangle)" Dialog box

Graphically Menu–"Remove loads" command
The main function of "Remove loads" command is to provide a method for deleting loads. When the command is chosen, the cursor changes from an arrow to a cross hair. Then, the desired loads can be removed by double clicking on each load individually.

Graphically Menu–"Edit Loads" command
The main function of this command is to provide a method for redefine or editing loads. When "Edit loads" command is chosen, the cursor changes from an arrow to a cross hair. Then, the desired loads can be selected by double clicking on each load individually.

Note
If the snap to grid option in "Grid" Dialog box is checked, the cursor will snap to a grid point each time the screen is clicked at a point.

Graphically Menu–"Cartesian grid" command
(See paragraph 5.3.3)

5.14.4 In table Menu

This menu contains the following commands:

- Distributing the point load
- Column type
- Point loads
- Moments M_x
- Moments M_y
- Line loads
- Line moments
- Distributed loads (Polygon)
- Distributed loads (Rectangular)

In table Menu –"Distributing the point load" command
Point load never applied in realty. If a point load represents a column load on a mesh of refine finite elements, the moment under the column will be higher than the real moment. To take the effect of the load distribution through the slab thickness, the column load must be distributed outward at 45 [°] from the column face until reaching the center line of the slab. To convert the point load to an equivalent uniform load over an appropriate area, check the "Distribute column load" check box in the Dialog box of Figure C-138.

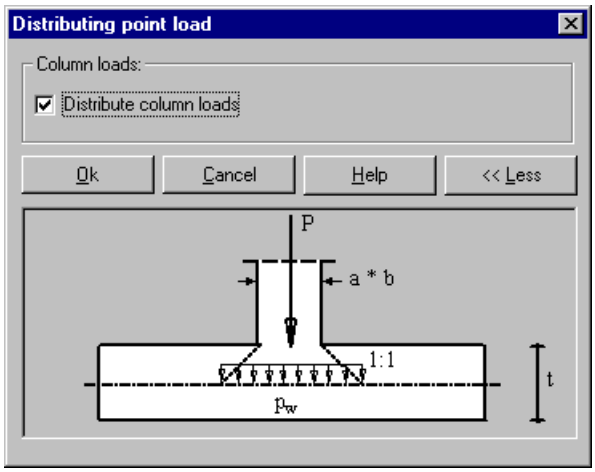


Figure C-138 Distributing the column point load over an appropriate area

In table Menu–"Column types" command
When the command is chosen, the following Table in Figure C-139 appears to define the column dimensions. Column dimensions are required for design of the slab for punching shear.

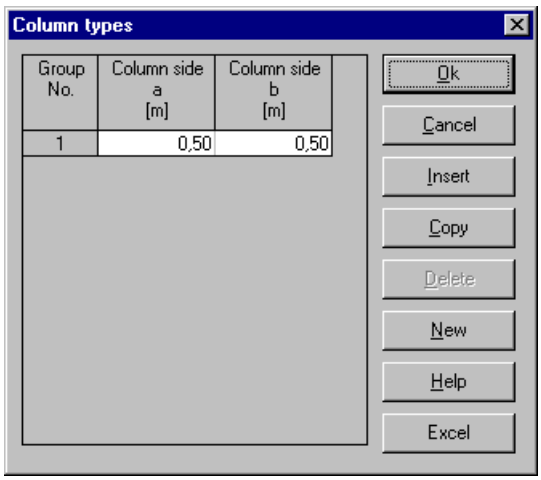


Figure C-139 Defining column dimensions

In table Menu –"Point loads" command

In the Dialog box of Figure C-140 the external point load P at the position (x, y) is defined. The coordinates for the input load are related to the left bottom corner of the corresponding slab (local coordinates).

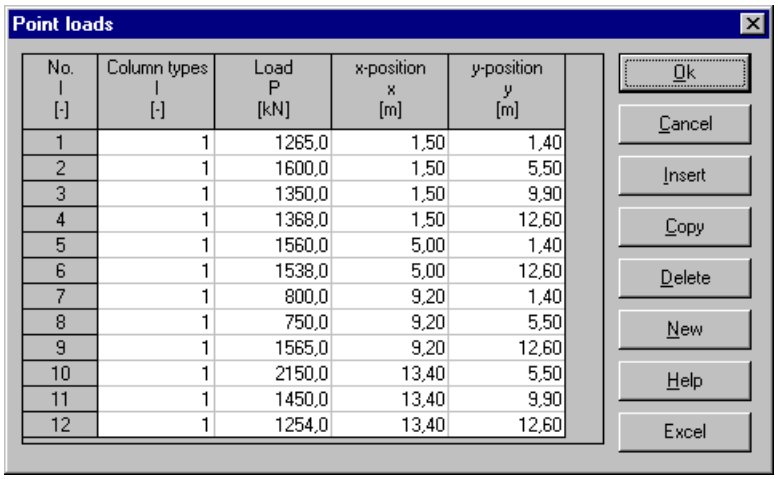


Figure C-140 "Point loads P" Dialog box

In table Menu–"Moments M_x" command

In the Dialog box of Figure C-141 the external moment M_x at the position (x, y) is defined. The coordinates for the input moment M_x are related to the left bottom corner of the corresponding slab (local coordinates).

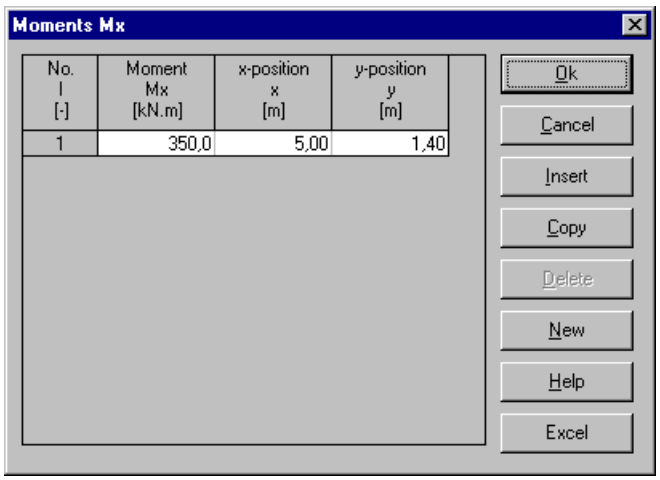


Figure C-141 "Moments M_x" Dialog box

In table Menu–"Moments M_y" command

In the Dialog box of Figure C-142 the external moment M_y at the position (x, y) is defined. The coordinates for the input moment M_y are related to the left bottom corner of the corresponding slab (local coordinates).

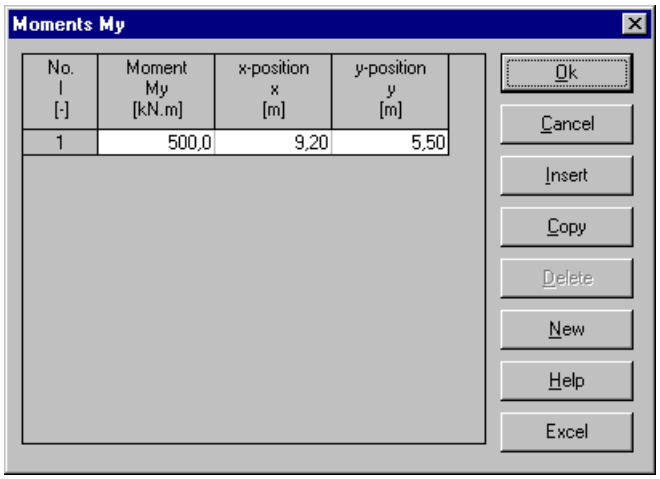


Figure C-142 "Moments M_y" Dialog box

In table Menu–"Line loads" command

In the Dialog box of Figure C-143 the external line load pl from the position (x_1, y_1) to (x_2, y_2) is defined. The coordinates for the input line load pl are related to the left bottom corner of the corresponding slab (local coordinates).

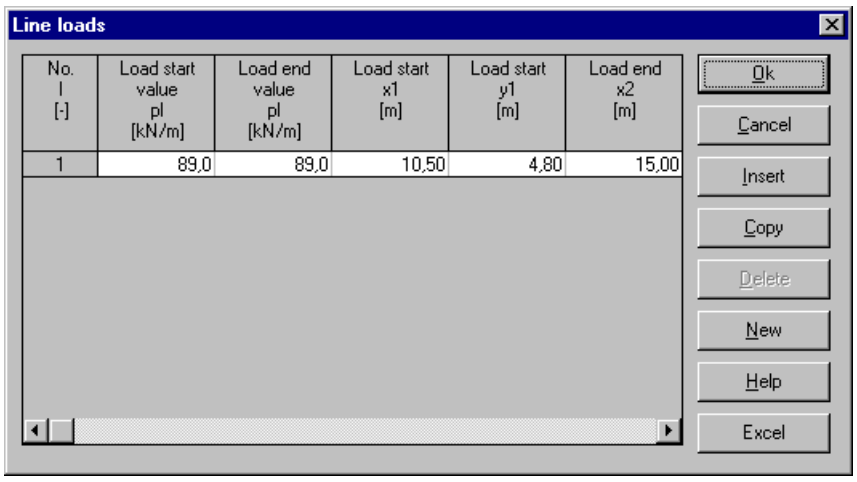


Figure C-143 "Line loads pl" Dialog box

In table Menu–"Line moments" command
In the Dialog box of Figure C-144 the external line moment Ml from the position (x_1, y_1) to (x_2, y_2) is defined. The coordinates for the input line moment Ml are related to the left bottom corner of the corresponding slab (local coordinates).

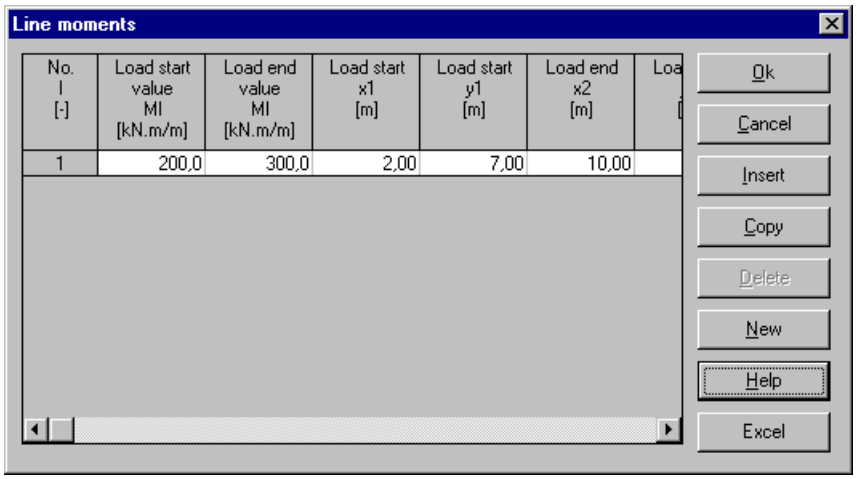


Figure C-144 "Line moments Ml" Dialog box

In table Menu–"Distributed loads (Polygon)" command
When "Distributed loads (Polygon)" command is chosen, the Dialog box of Figure C-136 appears to define load intensity at corners and corners position of the polygon.

In table Menu–"Distributed loads (Rectangle)" command
In the Dialog box of Figure C-145 the external distributed load p from the diagonal position (x_1, y_1) to (x_2, y_2) is defined. The coordinates for the input distributed load p are related to the left bottom corner of the corresponding slab (local coordinates).

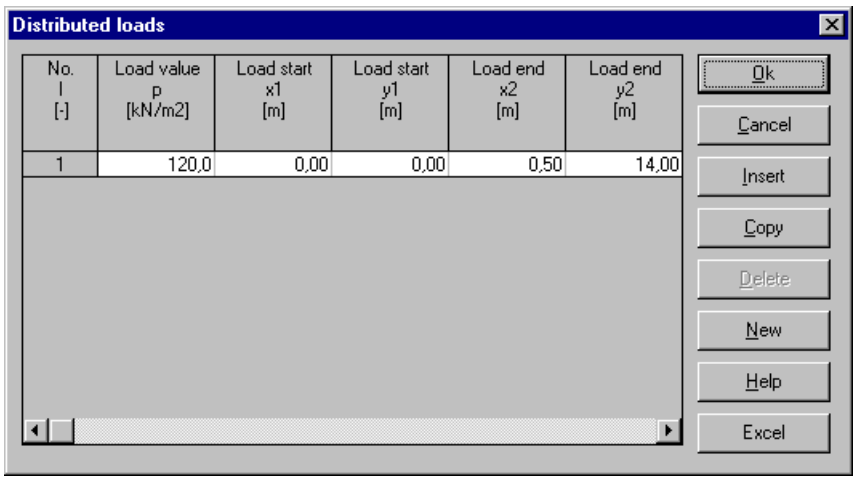


Figure C-145 "Distributed loads p" Dialog box

Note
If an element loaded area is overlap over another, the last input loaded area will be the valid one.

5.14.5 Using formula Menu

This menu contains the following commands:

- Point loads
- Moments M_x
- Moments M_y
- Line loads
- Line moments
- Distributed loads (Polygon)
- Distributed loads (Rectangle)

Using formula–"Point loads" command
This option is used to modify the load values through formula for existing load data. The option also may use to modify the load positions through formula in order to control the eccentricities e_x and e_y, Figure C-146.

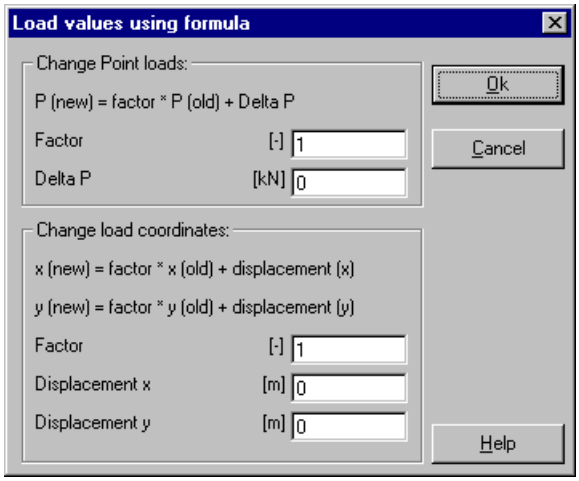


Figure C-146 "Load values using formula" Dialog box

Line loads, moments and distributed loads may be also modified independently using formula in the dialog box of Figure C-146.

5.14.6 Options Menu (See paragraph 5.3.6)

5.14.7 Format Menu (See paragraph 5.3.7)

5.14.8 Window Menu (See paragraph 5.3.8)

5.14.9 Help Menu (See paragraph 5.3.9)

5.15 Data Menu–"Neighboring foundations" command

To take the effect of neighboring foundations on the slab, the filenames for projects of neighboring foundations are required. In the Dialog box of Figure C-147 the filenames for projects of neighboring foundations can be defined.

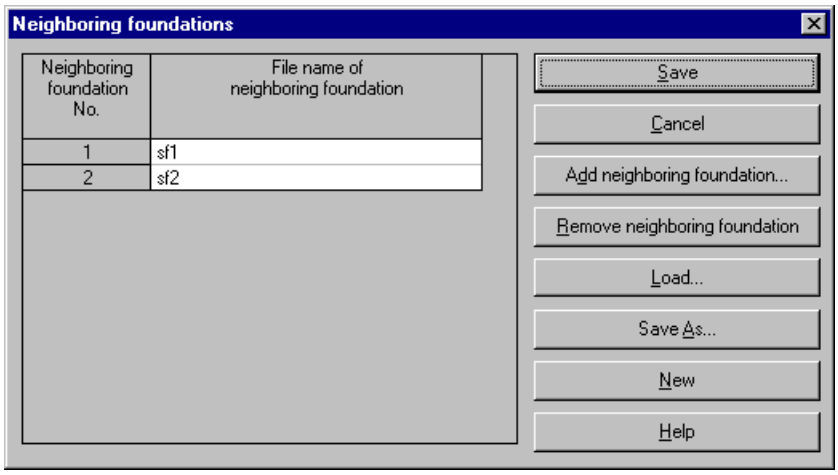


Figure C-147 "Neighboring foundations" Dialog box

5.16 Data Menu–"Temperature change" command

To take the effect of temperature change on the slab, the temperature difference Td and the coefficient of thermal expansion of the slab material are required in Figure C-148. Temperature difference Td occurs between the upper and lower surface of the slab foundation. If Td is 0 that means no temperature effect will occur. A positive Td means the temperature above the slab is greater than that under the slab. According to German Standard DIN 1045, the coefficient of thermal expansion of the concrete slab $\alpha = 0.00001$ [1/°C].

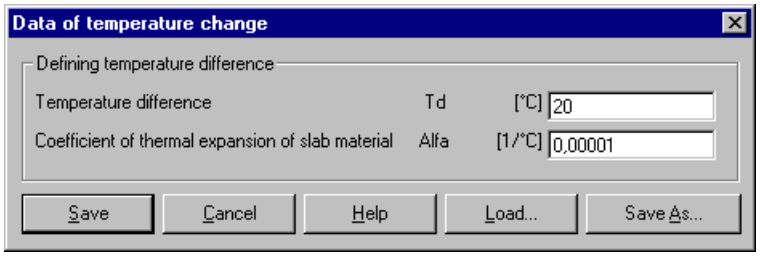


Figure C-148 "Data of temperature change" Dialog box

5.17 Data Menu–"Additional settlements" command

By this command additional settlements on the slab can be defined. It is also possible to define the additional settlements graphically, which makes the definition of the additional settlements very easy, or numerically (in a table). When the "Additional settlements" command is chosen, the following embedded program appears (Figure C-149).

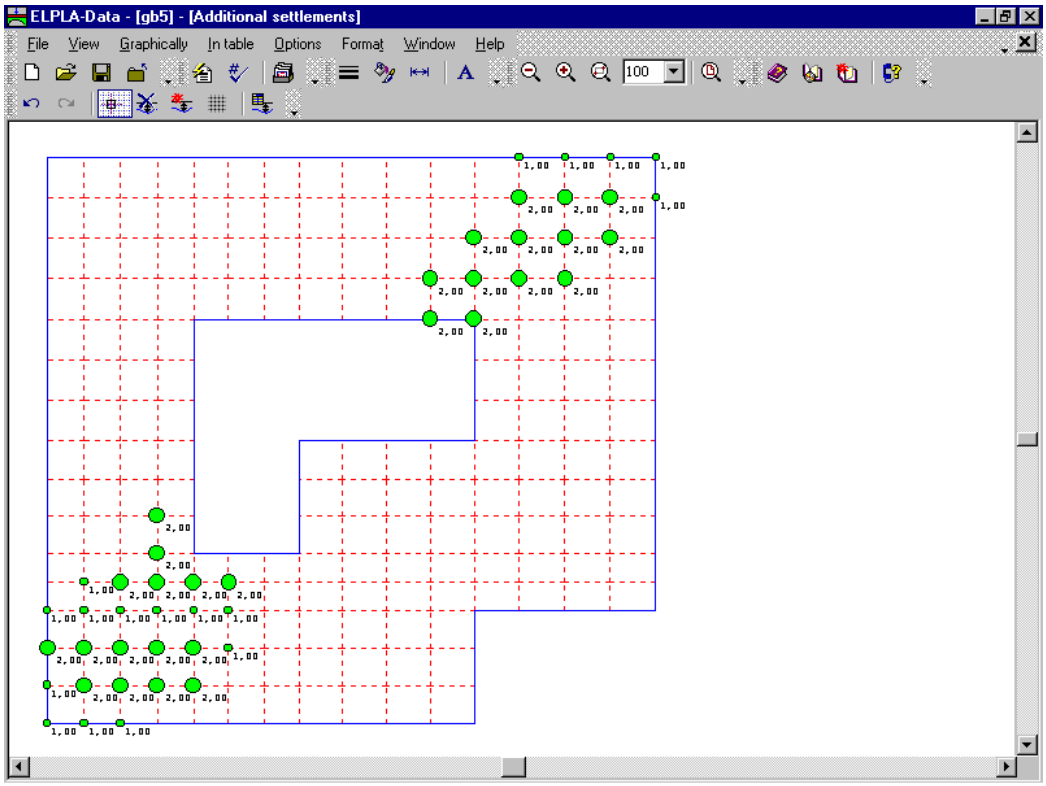


Figure C-149 "Additional settlements" embedded program

The menu head of Figure C-149 contains the following eight commands:

- File
- View
- Graphically
- In table
- Options
- Format
- Window
- Help

After clicking one of the eight commands (options) other sub-commands or options become available. The following paragraph presents and describes the eight menu commands and their sub-commands.

5.17.1 File Menu

This menu contains five commands:

- New additional settlements
- Open additional settlements
- Save additional settlements
- Save additional settlements as
- Close additional settlements

File Menu–"New additional settlements" command
Defines new additional settlements

File Menu–"Open additional settlements" command
Opens existing additional settlements-file again on the screen. Then the additional settlements, if desired, can be redefined.

File Menu–"Save additional settlements" command
Saves the active additional settlements under the available name

File Menu–"Save additional settlements as" command
Saves the active additional settlements under a new name

File Menu–"Close additional settlements" command
Closes the additional settlements-embedded program and returns to ELPLA-Data

5.17.2 View Menu (See paragraph 5.3.2)

5.17.3 Graphically Menu

This menu contains the following commands:

- Undo
- Redo
- Select nodes
- Remove additional settlements
- Add additional settlements
- Cartesian grid

Graphically Menu–"Undo" command
This command is used to undo the effects of a selected command and return to a previous state.

Graphically Menu–"Redo" command
This command is used to redo the last action of "Undo" command.

Graphically Menu–"Select nodes" command
The main function of "Select nodes" command is to provide a method for removing or adding additional settlement on nodes. When "Select nodes" command is chosen, the cursor is changed from an arrow to a cross hair. In this case, "Add additional settlements" and also "Remove additional settlements" will be enabled, indicating the modes in which are being operated. The

desired nodes are selected by clicking on each node individually or selecting a group of nodes. A group of nodes can be selected by holding the left mouse button down at the corner of the region and dragging the mouse until a rectangle encompasses the desired group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

Graphically Menu–"Remove additional settlements" command
The command is used to make the selected nodes free from additional settlements.

Graphically Menu–"Add additional settlements" command
This command is used to define the settlement value for the selected nodes. Any old additional settlements of the selected nodes will be replaced by the new editing. When this command is chosen, the following Dialog box of Figure C-150 appears to define the additional settlement.

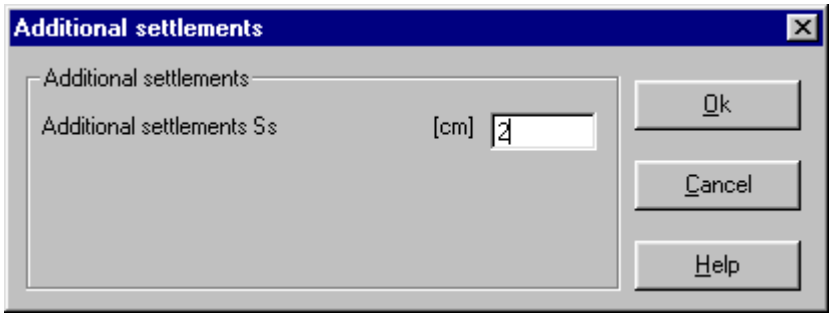


Figure C-150 "Add additional settlements" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.17.4 In table Menu

This menu contains the following command:

- Additional settlements

In table Menu–"Additional settlements" command
When the "Additional settlements" command is chosen, the following Dialog box appears. In this Dialog box (Figure C-151) define the additional settlement s_i at node i.

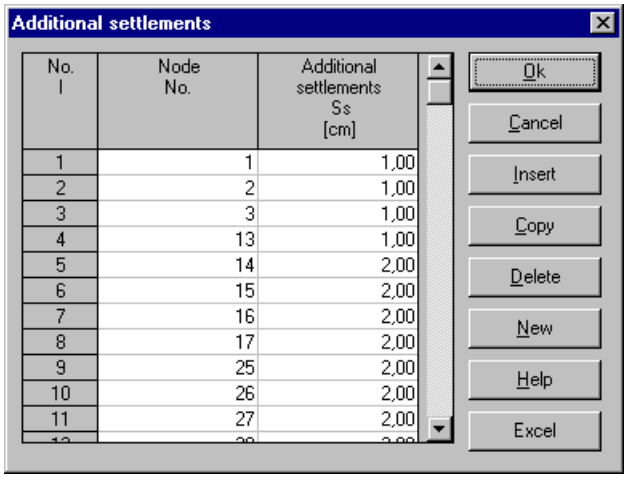


Figure C-151 Defining additional settlements in a table

5.17.5 Options Menu (See paragraph 5.3.6)

5.17.6 Format Menu (See paragraph 5.3.7

5.17.7 Window Menu (See paragraph 5.3.8)

5.17.8 Help Menu (See paragraph 5.3.9)

6 View Menu (See paragraph 5.3.2)

7 Main Data Menu

The Main Data Menu has the following commands:

- Firm header
- Directory of data
- Preferences
- Help language setting
- System of units
- Number formats
- Design code parameters

7.1 Main Data–"Firm header" command

The "Firm header" is two lines text to give information about your firm, company, institute or office, Figure C-152. The information is printed as headers at the top of the pages, which contain the tables of data and results that created by ELPLA-List. The information is also printed at the identification box for graphical drawings of data and results created by ELPLA-Graphic, ELPLA-Sections and ELPLA-Boring.

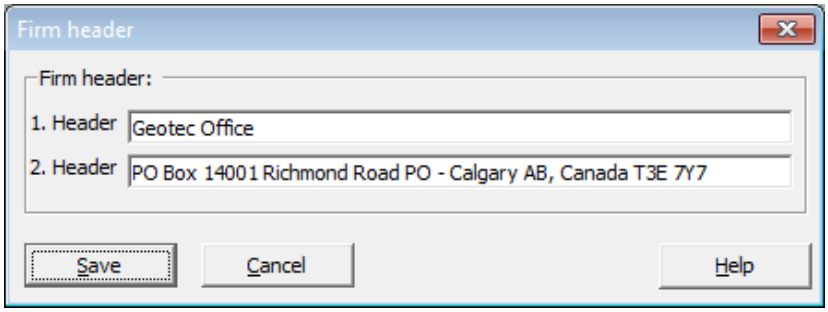


Figure C-152 "Firm header" Dialog box

7.2 Main Data–"Directory of data" command

Instead of storing hundreds of project files, the files of input data, intermediate results or final results for a project can optionally be stored automatically in one compressed file. This makes it easier to send projects to other persons or to simply manage your own files. It also reduces the amount of disk space required to store all of your data files. The compressed file is ZIP-compatible, allowing you to manually extract the data files using WinZip or other data compression tools if you wish.

In the Dialog box of Figure C-153 specify which directory is used as default directory for files that are saved or opened by ELPLA. Also, check the files to be compressed.

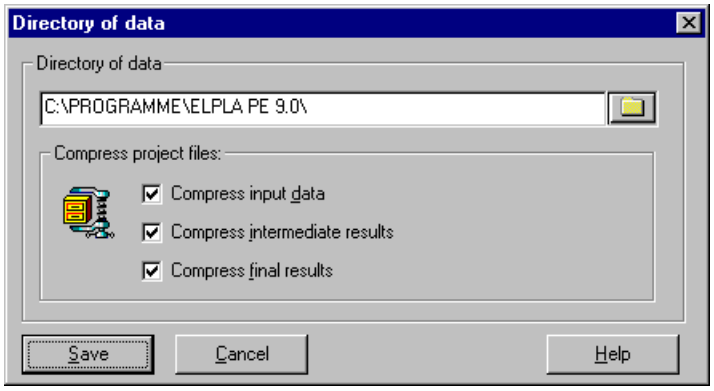


Figure C-153 "Directory of data" Dialog box

7.3 Main Data–"Preferences" command

In the Dialog box of Figure C-154 define the FE-Net and calculation of internal force preferences. To improve the distribution of the internal forces on the FE-Net, two possibilities for determining internal forces are available:

1. The internal forces are determined firstly at the element centers, and then distributed to the element nodes (recommended for triangular elements)

2. The internal forces are determined directly at the element nodes (recommended for rectangular elements)

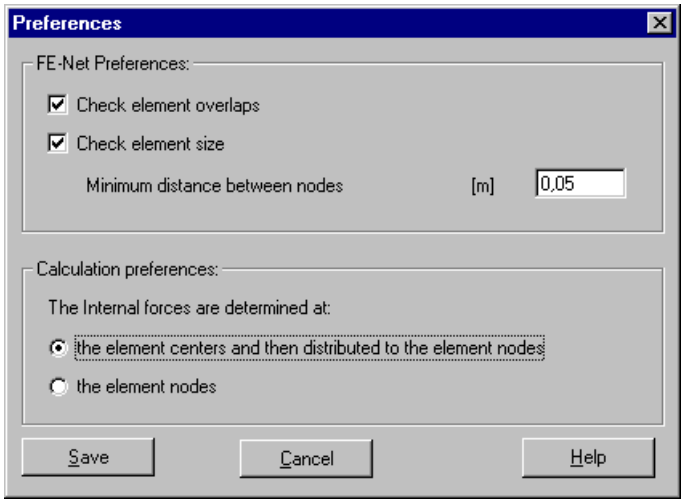


Figure C-154 "Preferences" Dialog box

7.4 Main Data–"Help language setting" command

It can define the language of the help system used in ELPLA applications (Figure C-155). The three languages are English, German and Arabic.

Figure C-155 "Help language setting" Dialog box

7.5 Main Data–"System of unit" command

It is possible to set different unit systems such as SI-system or English-system without changing the real value of any previously defined data, Figure C-156.

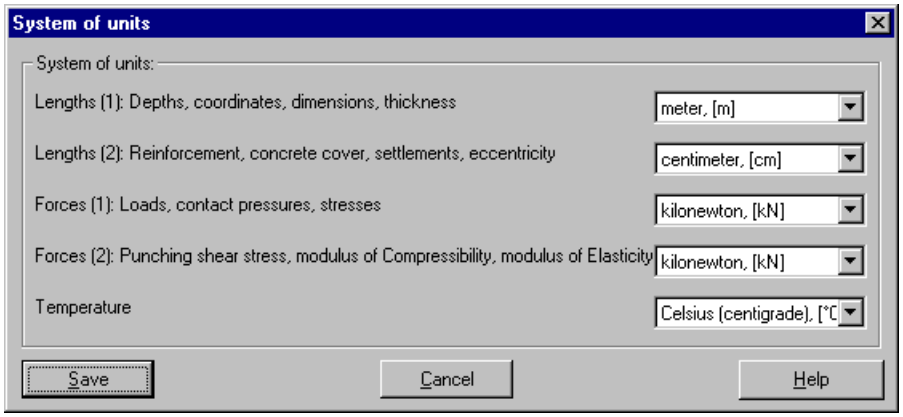


Figure C-156 "System of units" Dialog box

7.6 Main Data–"Number formats" command

By the "Number formats" command the user can choose, how the numbers of results and data are listed or printed, Figure C-157.

The following examples describe the number formats:

Number = 5459.3472
Format "0.000" gives 5459.347
Format "0.00" gives 5459.35
Format "0.0" gives 5459.4
Format "0" gives 5459
Format "00E+00" gives 55E+02 (Exponential format)

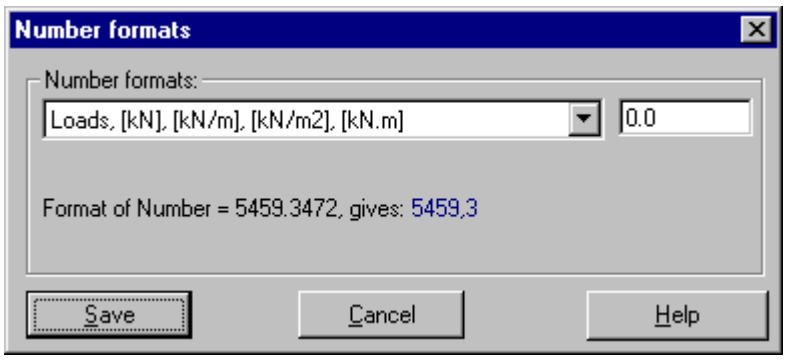


Figure C-157 "Number formats" Dialog box

7.7 Main data–"Design code parameters" command

The design of the slab for flexure moment and punching shear can be carried out according to the following design codes:

- EC 2
  European Committee for Standardization, Design of Concrete Structures - Eurocode 2

- DIN 1045
  German Institute for Standardization, Design and Construction of Reinforced Concrete

- ACI
  American Concrete Institute Building Code Requirements for Structural Concrete

- ECP
  Egyptian Code of Practice for Design and Construction of Reinforced Concrete Structures

In the menu of Figure C-158 the design code parameters may be redefined if desired. Also, the minimum reinforcements of tension and compression steel are defined.

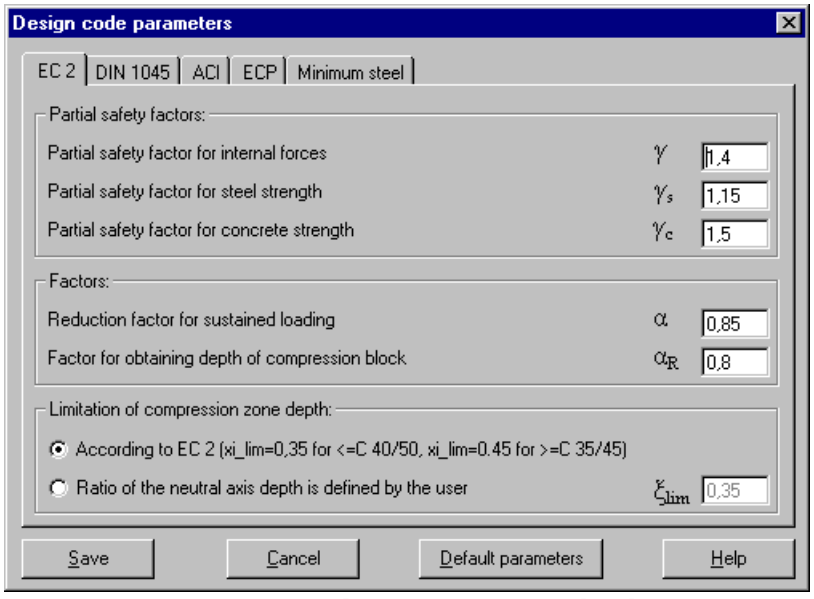


Figure C-158 "Design code parameters" Dialog box

8 Help Menu (See paragraph 5.3.9)

9 Tips and Tricks

9.1 Keyboard

The user can obtain all menu titles and commands also through Shortcut keys. The action of the Shortcut keys is listed in Table C-5 to Table C-19:

Table C-5 Shortcut keys of menu head

Shortcut keys	Action	
[Alt+f]	Calling menu head	"File"
[Alt+v]		"View"
[Alt+f]		"FE-Net Generation"
[Alt+i]		"In Table"
[Alt+g]		"Graphically"
[Alt+d]		"Data"
[Alt+m]		"Main data"
[Alt+a]		"Foundation properties"
[Alt+b]		"Boring fields"
[Alt+u]		"Using formula"
[Alt+o]		"Option"
[Alt+f]		"Format"
[Alt+w]		"Window"
[Alt+h]		"Help"

Table C-6 Shortcut keys of File-Command for embedded programs

Shortcut keys	Action
[Ctrl+n] or [Alt+f] then [n]	Calling command "New ***"
[Ctrl+o] or [Alt+f] then [o]	"Open ***"
[Alt+f] then [s]	"Save ***"
[Alt+f] then [a]	"Save *** as"
[Ctrl+q] or [Alt+f] then [c]	"Close ***"
[Alt+f] then [l]	"File list"
[Alt+f] then [1]	Calling the first project from the last four defined projects
[Alt+f] then [2]	Calling the second project from the last four defined projects
[Alt+f] then [3]	Calling the third project from the last four defined projects
[Alt+f] then [4]	Calling the fourth project from the last four defined projects
[Ctrl+q] or [Alt+f] then [x]	Calling command "Exit"

The asterisks (***) matches any of the embedded program name or the word “project”.

Table C-7 Shortcut keys of View-Command

Shortcut keys	Action	
[Alt+v] then [b]	Calling command	"Status bar"
[Alt+v] then [t]		"Tool bars"
[Alt+v] then [t], then [f]		"Tool bars-File"
[Alt+v] then [t], then [f]		"Tool bars-FE-Net"
[Alt+v] then [t], then [e]		"Tool bars-Edit"
[Alt+v] then [t], then [g]		"Tool bars-Graphically"
[Alt+v] then [t], then [d]		"Tool bars-Data"
[Alt+v] then [t], then [m]		"Tool bars-Main data"
[Alt+v] then [t], then [g]		"Tool bars-Girders"
[Alt+v] then [t], then [s]		"Tool bars-Spring supports"
[Alt+v] then [t], then [s]		"Tool bars-Supports/ Boundary conditions"
[Alt+v] then [t], then [p]		"Tool bars-Piles"
[Alt+v] then [t], then [z]		"Tool bars-Net of soil elements in z-direction"
[Alt+v] then [t], then [a]		"Tool bars-Foundation properties"
[Alt+v] then [t], then [b]		"Tool bars-Boring fields"
[Alt+v] then [t], then [l]		"Tool bars-Loads"
[Alt+v] then [t], then [a]		"Tool bars-Additional settlements"
[Alt+v] then [t], then [o]		"Tool bars-Option"
[Alt+v] then [t], then [t]		"Tool bars-Format"
[Alt+v] then [t], then [w]		"Tool bars-Window"
[Alt+v] then [t], then [h]		"Tool bars-Help"
[Alt+v] then [t], then [r]		"Tool bars-Reset Toolbar"

Table C-8 Shortcut keys of FE-Net Generation-Command

Shortcut keys	Action	
[Alt+n] then [g]	Calling command	"Generation type"
[Alt+n] then [n]		"New generation"
[Alt+n] then [f]		"Generating FE-Net"
[Alt+n] then [s]		"Smoothing mesh"
[Alt+n] then [b]		"Directing border elements"
[Alt+n] then [r]		"Refining mesh"

Table C-9 Shortcut keys of Graphically-Command for embedded programs

Shortcut keys	Action	
[Alt+g] then [r]	Calling command	"Undo"
[Alt+g] then [r]		"Redo"
[Alt+g] then [r]		"Remove girders, spring supports, boundaries, piles or loads"
[Alt+g] then [a]		"Add nodes, girders, spring supports, boundaries or piles"
[Alt+g] then [e]		"Edit node, girders or loads"
[Alt+g] then [s]		"Select nodes, elements"
[Alt+g] then [m]		"Slab corner by Mouse"
[Alt+g] then [a]		"Add opening"
[Alt+g] then [p]		"Add reference points"
[Alt+g] then [l]		"Add reference lines"
[Alt+g] then [o]		"Boring logs"
[Alt+g] then [z]		"Zone type I"
[Alt+g] then [o]		"Zone type II"
[Alt+g] then [n]		"Zone type III"
[Alt+g] then [p]		"Point loads"
[Alt+g] then [x]		"Moments Mx"
[Alt+g] then [y]		"Moments My"
[Alt+g] then [l]		"Line loads"
[Alt+g] then [m]		"Line moments"
[Alt+g] then [d]		"Distributed loads (Polygon)"
[Alt+g] then [t]		"Distributed loads (Rectangle)"
[Alt+g] then [c]		"Cartesian grid"

Table C-10 Shortcut keys of In table-Command for embedded programs

Shortcut keys	Action	
[Alt+i] then [c]	Calling command	"Column types"
[Alt+i] then [l]		"Distributing the point load"
[Alt+i] then [n]		"Net of finite elements"
[Alt+i] then [g]		"Girder groups"
[Alt+i] then [d]		"Girders"
[Alt+i] then [t]		"Spring supports"
[Alt+i] then [n]		"Node restraints"
[Alt+i] then [e]		"Element groups"
[Alt+i] then [r]		"Pile groups"
[Alt+i] then [p]		"Pile locations and groups"
[Alt+i] then [m]		"Pile material"
[Alt+i] then [g]		"Group regions"
[Alt+i] then [z]		"Zone type I"
[Alt+i] then [o]		"Zone type II"
[Alt+i] then [n]		"Zone type III"
[Alt+i] then [p]		"Point loads"
[Alt+i] then [x]		"Moments Mx"
[Alt+i] then [y]		"Moments My"
[Alt+i] then [l]		"Line loads"
[Alt+i] then [m]		"Line moments"
[Alt+i] then [d]		"Distributed loads (Polygon)"
[Alt+i] then [t]		"Distributed loads (Rectangle)"
[Alt+i] then [s]		"Additional settlements"

Table C-11 Shortcut keys of Data-Command

Shortcut keys	Action	
[Alt+d] then [s]	Calling command	"Soil data"
[Alt+d] then [o]		"Main soil data"
[Alt+d] then [c]		"Calculation methods"
[Alt+d] then [p]		"Project identification"
[Alt+d] then [f]		"FE-Net data"
[Alt+d] then [g]		"Girders"
[Alt+d] then [t]		"Spring supports"
[Alt+d] then [u]		"Supports/ Boundary conditions"
[Alt+d] then [p]		"Piles"
[Alt+d] then [i]		"Soil properties"
[Alt+d] then [z]		"Net of soil elements in z-direction"
[Alt+d] then [h]		"Limit depth"
[Alt+d] then [a]		"Foundation (or slab) properties"
[Alt+d] then [r]		"Reinforcement data"
[Alt+d] then [b]		"Boring fields"
[Alt+d] then [l]		"Loads"
[Alt+d] then [n]		"Neighboring foundations"
[Alt+d] then [t]		"Temperature change"
[Alt+d] then [s]		"Additional settlements"
[Alt+d] then [f]		"Filenames of slab foundations"

Table C-12 Shortcut keys of Main data-Command for ELPLA-Data und the embedded program Soil properties

Shortcut keys	Action	
[Alt+m] then [f]	Calling command	"Firm header"
[Alt+m] then [d]		"Directory of data"
[Alt+m] then [n]		"Number formats"
[Alt+m] then [p]		"Preferences"
[Alt+m] then [h]		"Help language setting"
[Alt+m] then [s]		"System of units"
[Alt+m] then [r]		"Design code parameters"

Table C-13 Shortcut keys of Foundation properties-Command for the embedded program Foundation properties

Shortcut keys	Action	
[Alt+a] then [u]	Calling command	"Unit weight of the foundation"
[Alt+a] then [f]		"Foundation depth"
[Alt+a] then [o]		"Origin coordinates"
[Alt+a] then [f]		"Foundation level from fixed datum"

Table C-14 Shortcut keys of Boring fields-Command for the embedded program Boring fields

Shortcut keys	Action	
[Alt+ b] then [i]	Calling command	"Interpolation method"
[Alt+ b] then [s]		"Subarea method"
[Alt+ b] then [h]		"Hand-Division of borings to nodes"

Table C-15 Shortcut keys of Using formula-Command for the embedded program loads

Shortcut keys	Action	
[Alt+u] then [p]	Calling command	"Point loads"
[Alt+u] then [x]		"Moments Mx"
[Alt+u] then [y]		"Moments My"
[Alt+u] then [l]		"Line loads"
[Alt+i] then [m]		"Line moments"
[Alt+u] then [d]		"Distributed loads (Polygon)"
[Alt+u] then [t]		"Distributed loads (Rectangle)"

Table C-16 Shortcut keys of Options-Command

Shortcut keys	Action	
[Alt+o] then [l]	Calling command	"Plot parameters"
[Alt+o] then [d]		"Display values"
[Alt+o] then [v]		"View grouping"

Table C-17 Shortcut keys of Format-Command

Shortcut keys	Action	
[Alt+t] then [l]	Calling command	"Line formats"
[Alt+t] then [i]		"Fill color"
[Alt+t] then [x]		"Max. ordinate"
[Alt+t] then [f]		"Font"
[Alt+t] then [g]		"Grid"

Table C-18 Shortcut keys of Window-Command

Shortcut keys	Action	
[Alt+w] then [i]	Calling command	"Zoom in"
[Alt+w] then [o]		"Zoom out"
[Alt+w] then [w]		"Zoom window"
[Alt+w] then [z]		"Zoom %"
[Alt+w] then [r]		"Original size"

Table C-19 Shortcut keys of Help-Command

Shortcut keys	Action	
[Alt+h] then [c]	Calling command	"Contents"
[Alt+h] then [s]		"New in ELPLA"
[Alt+h] then [n]		"Short description of ELPLA"
[Alt+h] then [a]		"About ELPLA-Data"

9.2 Mouse

By clicking the right mouse Button on the screen, the user can also obtain the Popup-Main data-Menu, Figure C-159.

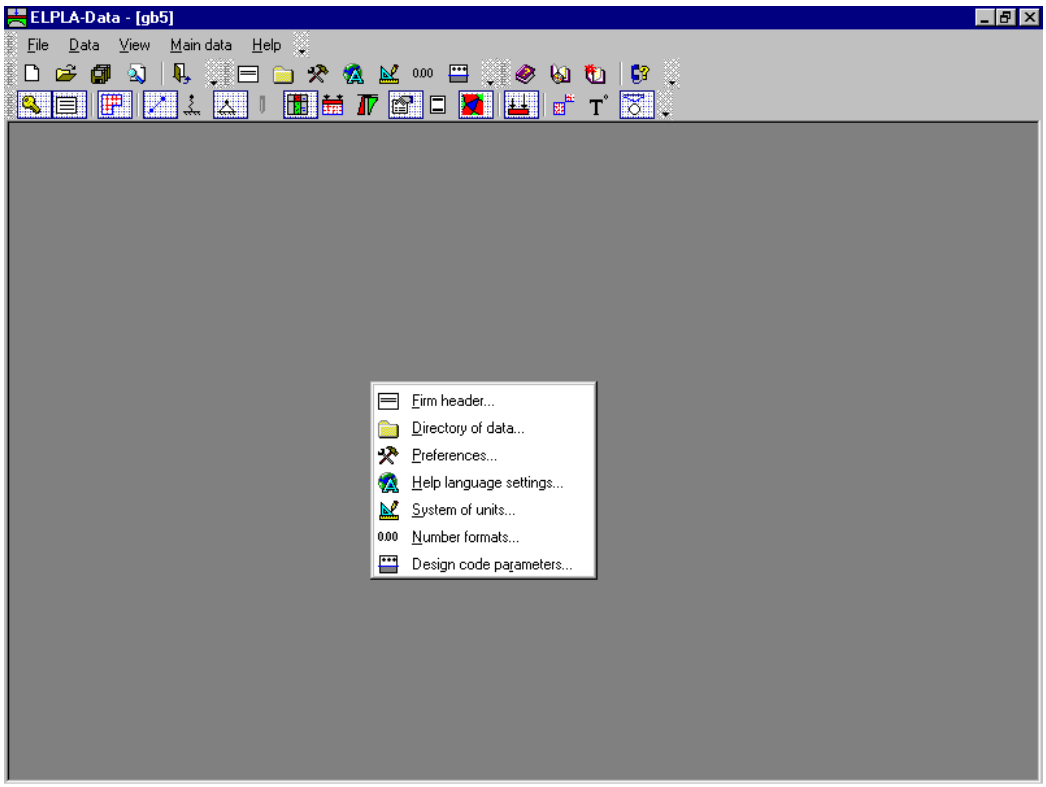


Figure C-159 Menu "Popup-Main data"

By clicking the right mouse Button on the screen for one of the embedded programs, the user can also obtain the Popup-Options-Menu, Figure C-160.

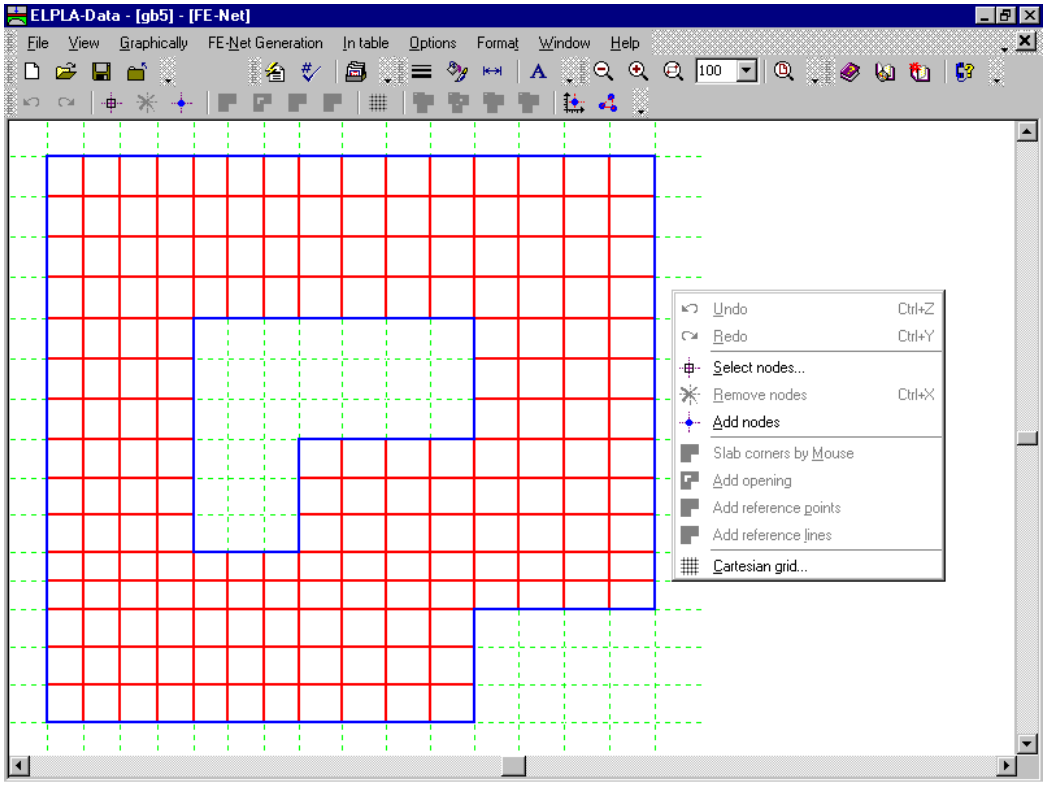


Figure C-160 Menu "Popup-Options"

10 References

[1] ACI (1995): Building Code Requirements for Reinforced Concrete (ACI 318-95) and Commentary (ACI 318R-95)
American Concrete Institute, Detroit, Michigan

[2] EC 2 (1993): Design of Concrete Structures
Deutsche Fassung: DIN V 18932 Teil 1
Beuth-Verlag GmbH Berlin und Beton-Kalender Oktober 1991

[3] ECP 464 (1989): The Egyptian Code of Practice, Design and Construction of Reinforced Concrete Structures (in Arabic)

[4] DIN 1045 (1988): Stahlbeton- und Spannbetonbau. Beton und Stahlbeton, Bemessung und Ausführung. Ausgabe Juli 1988

[5] DIN 1075 (04.1981): Betonbrücken; Bemessung und Ausführung

[6] DIN 4019 Teil 1 mit Beiblatt Baugrund (1974): Setzungsberechnungen bei lotrechter, mittiger Belastung
Beuth-Verlag, Berlin, Köln

[7] DIN 4023 (März 1984): Baugrund- und Wasserbohrungen.
Zeichnerische Darstellung der Ergebnisse
Beuth-Verlag, Berlin

[8] KANY, M./ El GENDY, M. (in Vorbereitung 2003): Computerberechnung und Entwurf von Fundamenten

11 Index

P

R

S

T

U

V

Z