# **Tutorial Manual** for the program *ELPLA*



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

## ELPLA-Tutorial

Cont	ents	Page
1	Introduction	3
2	Installing ELPLA	3
3	Starting ELPLA	6
4	Description of the examples	10

### 1 Introduction

This Tutorial Manual contains an overview of dealing with structural problems such as rafts, slabs, grids, plane frame and plane stress. It describes the processes of modeling the problems, carrying out the calculations, viewing and printing the results. It provides the user skills, which he needs to use *ELPLA*. It also takes the user step by step through some simple examples. Carrying out these examples will help the user to become familiar with the most important functions of *ELPLA*. Before attempting a real project with *ELPLA*, it is recommended that the user tries to carry out the given problems.

This Tutorial Manual will not present the theoretical background of modeling the problems. For more information concerning the methods of analysis, a complete reference for calculation methods and numerical models is well documented in the User's Guide of *ELPLA*. Also a complete reference for all menus and dialog boxes of the program is to be found in the User's Guide or in the online help system.

### 2 Installing *ELPLA*

*ELPLA* is distributed on a CD-ROM containing an installer program to install the *ELPLA* software on your computer.

To install *ELPLA* follow these steps:

- Insert the CD-ROM into your drive

The installer program is automatically loaded when the CD-ROM is inserted into the drive, Figure 1. The installer will guide you through the steps required to install *ELPLA* on your computer.



Figure 1 ELPLA Installer form

- Click "Next" button in the form of Figure 1 to install *ELPLA* on your computer

ELPLA Installer program begins execution. Follow the instructions (Figure 2).

- Click "Next" button in the form of Figure 2 to install *ELPLA* software to the specified destination folder

You can specify the folder for *ELPLA* files. By default, *ELPLA* suggests ...\Program Files\ELPLA PE 9.x. However, you can optionally indicate a different folder name, if desired. *ELPLA* creates the folder name you specify.

🙀 ELPLA PE 9	.0	
Select Ins	tallation Folder	
The installer w	ill install ELPLA PE 9.0 in the following folder.	
To install in thi below or click	is folder, click "Next". To install to a different new or existing "Browse".	folder, enter one
<u>F</u> older:	J:\Program Files\ELPLA PE 9.0\	Browse
You can instal	I the software on the following drives:	
Volume		Disk 📥
<b>C</b> :		204
🗇 D:		204 🗸
<		>
		Disk Cost
	<u>C</u> ancel <u>P</u> revious	Next

Figure 2 "Select Installation Folder" form

After selecting the installation folder, *ELPLA* Installer program will be ready to install *ELPLA* software on your computer, Figure 3.

- Click "Next" button in the form of Figure 3 to start the installation

ELPLA will be installed and a statues form will show the process of installation (Figure 4).

🕫 ELPLA PE 9.0	
Confirm Installation	
The installer is ready to install ELPLA PE 9.0 on your computer.	
Click "Next" to start the installation.	
	<u>N</u> ext

Figure 3 "Confirm Installation" form

🕲 ELPLA PE 9.0	
Installing ELPLA PE 9.0	
ELPLA PE 9.0 is being installed.	
Please wait	
<b>Cancel</b> Previous	<u>N</u> ext

Figure 4 "Installing *ELPLA*" form

*ELPLA* installation will be completed and a message appears to inform you that the installation was completed successfully, Figure 5.

🛃 ELPLA PE 9.0			
Installation Complete			≺ ] +
ELPLA PE 9.0 has been sucessfully inst	alled.		
Click "Close" to exit.			
[	<u>C</u> ancel	Previous	

Figure 5Final Installer message

### Note

Installer cannot install update system or shared files, if they are in use. Before installing *ELPLA*, it is recommended that you close any application may be running.

### 3 Starting ELPLA

After successfully installing *ELPLA*, a new program group and program items will be created automatically for *ELPLA* in the Windows-Start-Menu. Also a program icon will be created on the Windows desktop, Figure 6. *ELPLA* professional package contains the individual programs *ELPLA-Boring*, *ELPLA-Data*, *ELPLA-Graphic*, *ELPLA-List*, *ELPLA-Section*, *ELPLA-Solver* and *GEOTEC-Text*, besides the help program *ELPLA-Help*. All those programs can run separately or through the main program *ELPLA*. The usage of the program is typically such that first data files are created describing a certain problem by the program *ELPLA-Data*. Then the project problem is analyzed by using the program *ELPLA-Solver*. Finally, the results can be presented as graphical drawings, graphs and tables using the five separate programs *ELPLA-Graphic*, *ELPLA-Section*, *ELPLA-List*, *ELPLA-Boring* and *GEOTEC-Text*. Names and short descriptions of the function of the *ELPLA* sub programs are given in Table 1.



Figure 6 Starting *ELPLA* 

Start the main program *ELPLA* PE 9.x by choosing it from the Windows-Start-Menu, Figure 6. A shortcut to start *ELPLA* is double clicking on *ELPLA* PE 9.x icon on the Windows desktop. After starting *ELPLA* PE 9.x for the first time, the window in Figure 7 appears. This window belongs to the sub program *ELPLA-Data*. The function of *ELPLA-Data* is defining the project data such as FE-Net, soil properties, raft material, boundary conditions, loads, etc.

On the upper-left corner of this window, the menu bar of *ELPLA-Data* appears which is used for entering the project data. In order to use the programs *ELPLA-Solver*, *ELPLA-Graphic*, *ELPLA-List*, *ELPLA-Section*, *ELPLA-Boring*, first the user must define the project data by the program *ELPLA-Data*.

Table 1 Names and descriptions of <i>ELPLA</i> sub 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-Text	Simple word processing program			

Table 1Names and descriptions of *ELPLA* sub programs

In the following section the user will find a brief description of some of the essential interface commands. This section will help the user to be familiar with some of the commands, which will be used in this Tutorial.

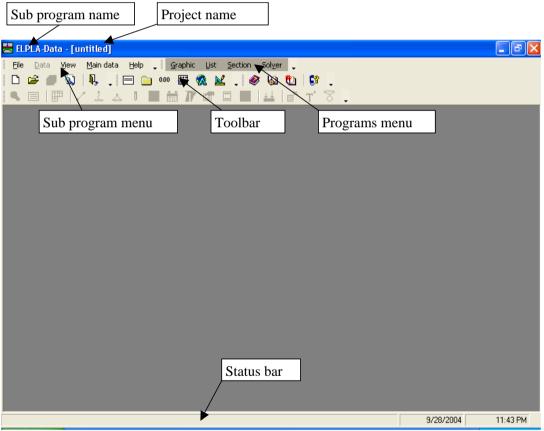


Figure 7 Opening screen of the sub program *ELPLA-Data* 

### Switching between ELPLA sub programs

On the upper-right corner of the window in Figure 7 the menu bar of the sub programs appears which is used for switching between the individual subprograms of *ELPLA* package. The user can switch to a specified sub program by clicking on the name of that sub program. When switching to a new sub program, *ELPLA* automatically opens the data file of the current example and displays the data file name in the title bar of this sub program.

### **Toolbar Buttons and Keyboard Shortcuts**

Many of *ELPLA* commands can be accessed by clicking buttons on the toolbars, or by pressing shortcut keys on the keyboard. When you select a menu, the available toolbar buttons and keyboard shortcuts for that menu are shown next to their corresponding commands as shown in Figure 8. You can also directly see what command is associated with a toolbar button by holding the mouse cursor over the button. After a brief pause a legend (Screen Tip) will appear next to the cursor showing the menu command associated with the button.



Figure 8 Available toolbar buttons and keyboard shortcuts for the file menu

### **Mouse Cursor Modes**

*ELPLA-Data* has two modes of mouse cursor operation in *ELPLA-Data* window: View mode and Edit mode. The program can only be in one mode at a time. By default, the program is in view mode and the mouse cursor is an arrow. For edit mode, the cursor will change from an arrow to a cross hair indicating the mode in which is being operated. Press "Esc" key to exit the edit mode and return to the view mode.

### **Node Selection**

In edit mode 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. Then, 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.

### **Undo and Redo Commands**

*ELPLA-Data* allows you to go back up to 12 steps at a time when defining project process. Therefore it is possible to undo a series of actions previously performed. If you go too far in the undo process you may redo those actions. You can undo/ redo most drawing, editing and assignment operations. To undo/ redo a certain action, choose the "Undo/ Redo" command from "Edit" menu.

### **Snap Tools**

The snap tools are essentially a fast and accurate way to draw and edit objects. Snap tools find the closest snap location to your pointer as you move it over your model. The snap tools can be turned on and off as you draw, so you can snap to different locations for every point. More than one snap tool can also be set at the same time giving you a choice of snap locations. The snap options are set by the data located in the "Plot Parameters" dialog box under the "Options" menu.

### **Defining Data**

Most of *ELPLA* data can be defined either graphically or numerically (in tables). In this tutorial the user will learn how to carry out the data using one method depending on the specified example. By completing the all examples, the user will learn to define most of the data both graphically and numerically.

### **Project identification**

The user can define three lines of texts to identify a project and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered.

### **Reinforcement data**

The design code parameters such as partial safety factors for concrete strength, steel strength and internal stresses are defined by choosing the "Design code parameters" command from the "Main data" menu in *ELPLA-Data*, while reinforcement data such as design code, concrete grade, steel grade and concrete covers are defined by choosing the "Reinforcement" command from "Data" menu in *ELPLA-Data*. Design code parameters are standard data for all projects while reinforcement data may be varied from project to another.

### **Edit List Box**

Some *ELPLA* data are defined by list box dialog as shown in Figure 9. In this list box the "Insert", "Copy" and "Delete" commands are applied for the selected row. To define or modify a value in this table, type that value in the corresponding cell, then press "Enter" key.

Point	oads					
No.   [-]	Column types I	Load P [kN]	x-position × [m]	y-position y [m]		
1	1	1265.0	1.50	1.40		Cancel
2	1	1600.0	1.50	5.50		Insert
3	1	1350.0	1.50	9.90		· ·
4	1	1368.0	1.50	12.60		<u>C</u> opy
5	1	1560.0	5.00	1.40		
6	1	1538.0	5.00	12.60		<u>D</u> elete
7	1	800.0	9.20	1.40		
8	1	750.0	9.20	5.50		New
9	1	1565.0	9.20	12.60		
10	1	2150.0	13.40	5.50		Help
11	1	1450.0	13.40	9.90		
40	1	4054.0	42.40	40.60	$\sim$	Excel

Figure 9 List box used by *ELPLA* 

### Analysis progress

During the analysis of problems all computations and analysis progression according to the defined method are progressively reported. As shown in Figure 10, the analysis progress menu reports the various phases of calculations. Also the status bar of *ELPLA-Solver* displays information about the progress of calculation as the program analyzes problems.

Determining flexibility coefficients of the soil		
Assembling the flexibility matrix of the slab 1!		
Time remaining = 00:00:00 I = 122 from 145 steps 		

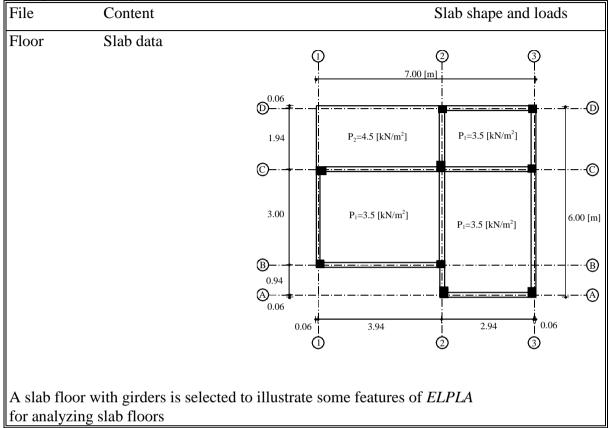
Figure 10 Analysis progress menu

### 4 Description of the examples

This Tutorial contains 13 examples. These examples are presented in order to illustrate how to use *ELPLA* for analyzing rafts, pile groups, piled raft, slab floor, grid, truss, frame and beam. For each example discussed in this Tutorial, data files and some computed files are included in *ELPLA* software package. The file names and contents of the examples are given as follows. Besides, a key figure of each problem is shown that contains the main data concerning the structure shape and loads. Examples can be run again by *ELPLA* to examine the details of the analysis or to see how the problem was defined or computed and to display, print or plot the results.

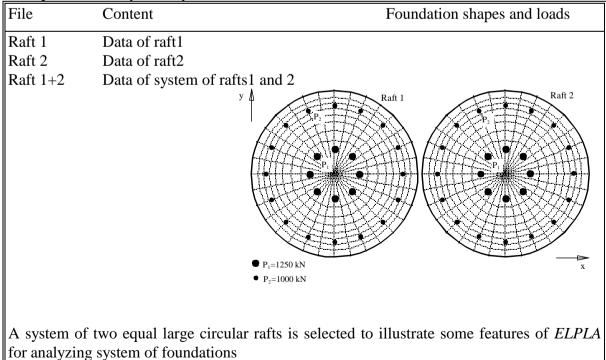
Example 1	A malarata of an	irregular shape	ad watte are inne	
сханные г	A DALVSIS OF AL	і птеўшаг упапе	-0 ган он шт	2011/ar Shoson
L'Aumpie 1	i mai yono or ar	i moguna shape	cu ruit on nit	Sala Sacooli

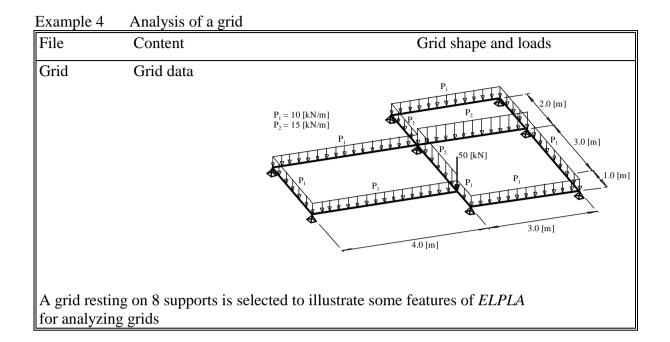
File	Content	Foundation shape and load
Example	Raft data	$\begin{array}{c} 4^{49,90} \\ 4^{40,90} \\ 4^{40,90} \\ 1565 \\ 1560 \\ 1560 \\ 1500 \\ 1$
	r shaped raft on <i>ELPLA</i> for analy	irregular subsoil is selected to illustrate some of the essential zing rafts



Example 2 Analysis of a slab floor

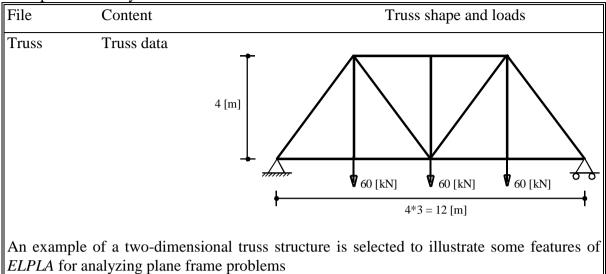
Example 3 Analysis of system of two circular rafts





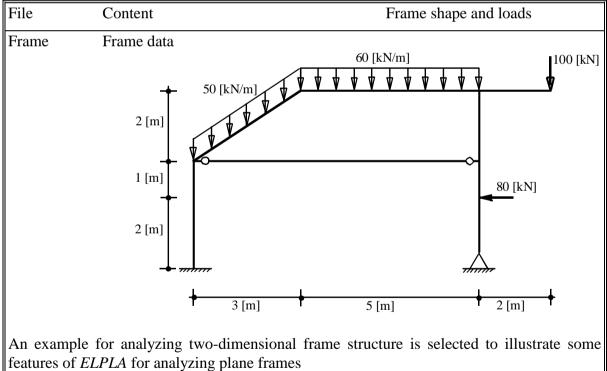
Example 5	Plane Stress Analysis of a Wrench

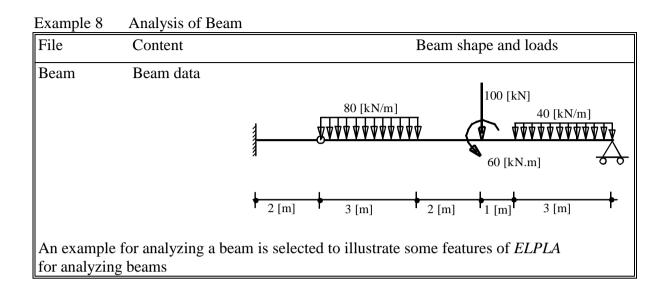
File	Content		Wrench shape and loads
Wrench	Wrench data	45 [mm] 35 [nm] <sup>R=10 [mm]</sup> 2.5 [mm] 45 [mm] 45 [mm]	100 [mm] 2 [N/mm] C R = 10 [mm] D Thickness = 10 [mm] 195 [mm]
-		e stresses and deformations analyzing plane stress proble	of a wrench is selected to illustrate ems

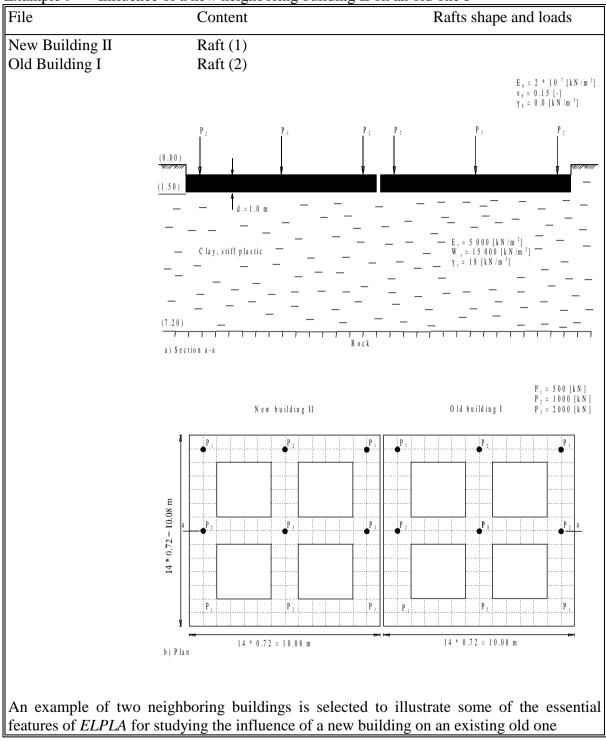


Example 6 Analysis of Two-dimensional Truss

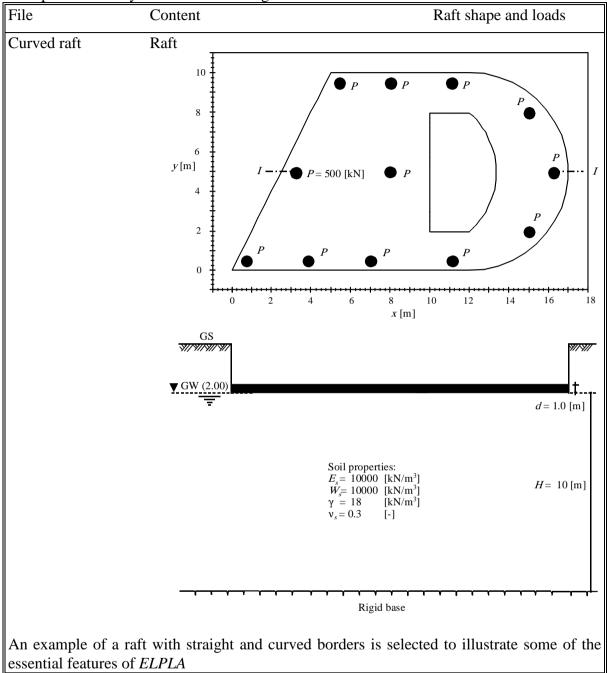
Example 7	Analysis of Plane Frame
-----------	-------------------------







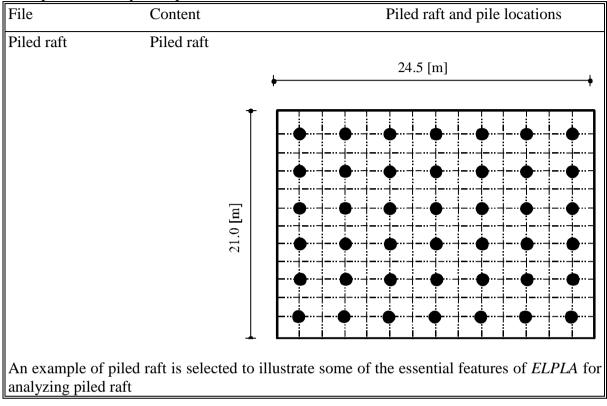
Example 9 Influence of a new neighboring building II on an old one I



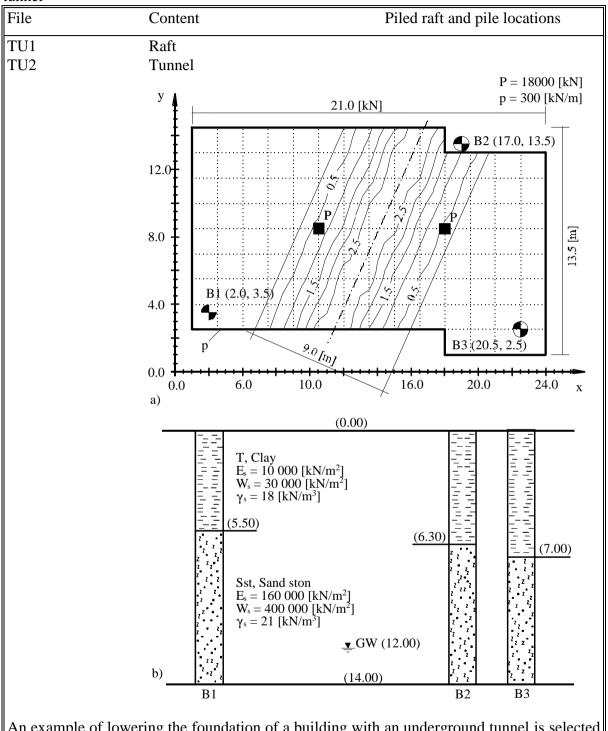
Example 10 Analysis of raft with straight and curved borders

File	Content			Pil	e group	s and lo	ad	
Pile groups	Pile groups		+	3.8		1.4 <i>y</i>	1.2	
		4				Ø	۲	
						Ø	Ø	2.8
		5 = 8 [m]				P = 800	00 [kN]	+ ∞.
		1.6*5	···-		_  	<u>°</u> .		······································
			Ø	۲	Ø	۲	Ø	3.4
		V	Ø	Ø		₿	٢	↓ ↓
					1.6*4 =	6.4 [m]		
An example of p for analyzing pile	vile groups is selecte e groups	ed to illus	strate so	me of tl	he essei	ntial fea	tures of	f ELPLA

Example 11 Analysis of pile groups



## Example 12 Analysis of piled raft



Example 13 Interaction of lowering the foundation of a building with an underground tunnel

An example of lowering the foundation of a building with an underground tunnel is selected to illustrate some of the essential features of *ELPLA* for studying the influence of an underground tunnel on a building foundation.

# Example 1

## Analysis of an irregular shaped raft on irregular subsoil

## ELPLA-Tutorial

## Contents

1	Desc	cription of the problem	3
	1.1	Loads	3
	1.2	Raft material and thickness	3
	1.3	Soil properties	4
	1.4	Method of analysis	4
2	Crea	ting the project	7
	2.1	Calculation method	7
	2.2	Project identification	10
	2.3	FE-Net data	11
	2.4	Soil properties	14
	2.5	Foundation properties	21
	2.6	Boring fields	24
	2.7	Loads	25
3	Carr	ying out the calculations	27
	3.1	Starting ELPLA-Solver	27
	3.2	Carrying out all computations	28
4	View	ving data and results	29
	4.1	Viewing data and result graphics	30
	4.2	Plot a diagram of the results at a specified section	32
	4.3	Print the drawing	34
	4.4	Listing data and results in tables	35
5	Index	X	38

### **1** Description of the problem

A simple example of an irregular shaped raft on irregular subsoil is selected to illustrate some of the essential features of *ELPLA* for analyzing rafts.

### 1.1 Loads

The raft carries many types of external loads: concentrated loads [kN], uniform load [kN/m<sup>2</sup>], line load [kN/m] and moments [kN.m] in both *x*- and *y*-directions as shown in Figure 1-1 and in Table 1-1 to Table 1-5.

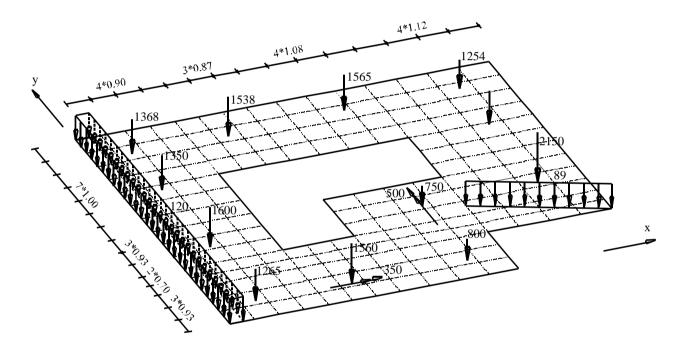


Figure 1-1 Raft dimensions [m] and loads

### **1.2** Raft material and thickness

Raft material and thickness are supposed to have the following parameters:

Young's modulus	$E_b$	$=2 * 10^{7}$	$[kN/m^2]$
Poisson's ratio	$\mathbf{V}_b$	= 0.25	[-]
Unit weight of raft material	$\gamma_b$	= 0.0	$[kN/m^3]$
Raft thickness	d	= 0.5	[m]

Unit weight of raft material is chosen to be  $\gamma_b = 0.0 \text{ [kN/m^3]}$  to neglect the own weight of the raft in the analysis.

### 1.3 Soil properties

Three boring logs characterize the subsoil under the raft. Each boring has three layers with different soil materials. The Moduli of Elasticity of the three layers for loading are  $E_{s1} = 9500$  [kN/m<sup>2</sup>],  $E_{s2} = 22000$  [kN/m<sup>2</sup>] and  $E_{s3} = 120000$  [kN/m<sup>2</sup>] while for reloading are  $W_{s1} = 26000$  [kN/m<sup>2</sup>],  $W_{s2} = 52000$  [kN/m<sup>2</sup>] and  $W_{s3} = 220000$  [kN/m<sup>2</sup>]. *Poisson's* ratio is 0.3 [-] for all soil layers. Unit weight of the soil above the ground water is  $\gamma_s = 19$  [kN/m<sup>3</sup>] while that under the ground water is  $\gamma_s = 9$  [kN/m<sup>3</sup>]. The foundation depth  $d_f = 2.7$  [m], the ground water depth under the ground surface is GW = 1.5 [m]. Figure 1-2 shows boring logs and locations.

### 1.4 Method of analysis

It is required to analyze the raft according to the following soil models and numerical calculation methods:

- Layered soil medium Continuum Model
- Modulus of compressibility method for an elastic raft on layered soil medium (Solving system of linear equations by iteration-method 6)

In the analysis the following items will be taken into account:

- The effect of reloading of the soil due to the overburden pressure
- The effect of water pressure on the raft
- The irregularity of the subsoil under the raft using the interpolation method (Figure 1-2)

This Tutorial Manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for the soil models and numerical calculation methods is well documented in the User's Guide of *ELPLA*.

y-position y [m]	x-position x [m]	Load value P [kN]	Load No. I [-]
1.4 5.5 9.9 12.6 1.4 12.6 1.4 5.5 12.6 5.5 9.9 12.6	1.5 1.5 1.5 1.5 5 9.2 9.2 9.2 9.2 9.2 13.4 13.4 13.4	1265 1600 1350 1368 1560 1538 800 750 1565 2150 1450 1254	1 2 3 4 5 6 7 8 9 10 11 12

Table 1-1Point loads P

#### Table 1-2Moment Mx

Moment	Moment value	x-position	y-position
No.	Mx	x	y
[-]	[kN.m]	[m]	[m]
1	350	5	1.4

### Table 1-3Moment My

Moment No. [-]	Moment value My [kN.m]	x-position x [m]	y-position y [m]
1 1	500	9.2	5.5

### Table 1-4Line load *pl*

I	Load value Pl [kN/m]	x1	уl	Load end x2 [m]	Load end y2 [m]
1	89	10.5	4.8	15	2.8

## Table 1-5Distributed load p

Load No.	Load value	Load start	Load start	Load end	Load end
I	P	x1	y1	x2	y2
[-]	[kN/m2]	[m]	[m]	[m]	[m]
1	120	0	0	0.5	13.97

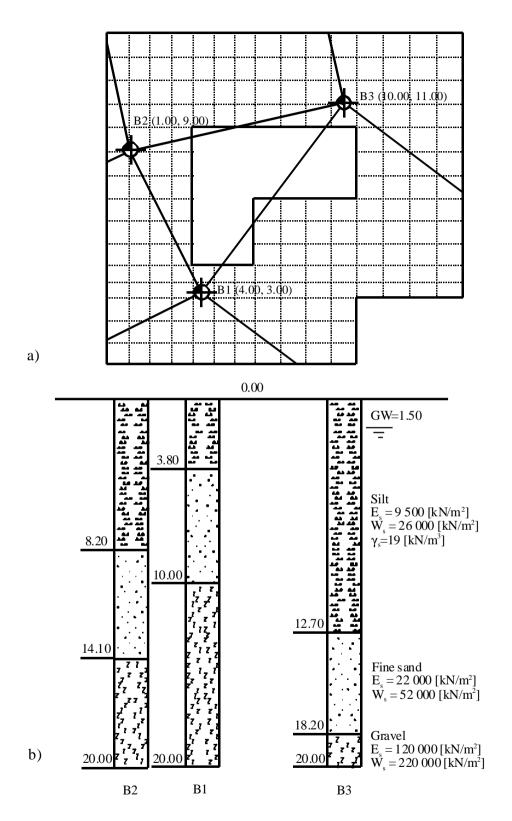


Figure 1-2 a) Boring locations and interpolation zones b) Boring logs B1 to B3

### 2 Creating the project

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

### 2.1 Calculation method

Choose "New project" command from the "File" menu of *ELPLA-Data*. The following "Calculation methods" wizard appears, Figure 1-3. 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 1-3).

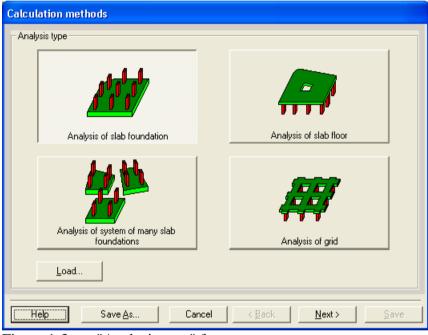


Figure 1-3 "Analysis type" form

In the "Analysis type" form in Figure 1-3 define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a foundation problem, select "Analysis of slab foundation" then click "Next" button to go to the next form. After clicking "Next" button, the "Calculation methods" form appears, Figure 1-4.

To define the calculation method

- Select the calculation method "6-Modulus of Compressibility (Iteration)"
- Select subsoil model "Layered soil model"
- Click "Next" button to go to the next form

C	alculation methods			
[	Calculation methods:			
	C 1- Linear Contact Pressure			
	C 2/3- Constant/ Variable Modulus of Subgrade Reaction			
	C 4- Modification of Modulus of Subgrade Reaction by Iteration			
	C 5- Isotropic Elastic Half Space			
	6 Hodulus of Compressibility (Iteration)			
	C 7- Modulus of Compressibility (Elimination)			
	C 8- Rigid slab			
	C 9- Flexible foundation			
	Subsoil model:			
	C Half Space model			
	Layered soil model			
-				
	Help Save <u>A</u> s Cancel < <u>B</u> ack <u>Next</u> > <u>S</u> ave			

Figure 1-4 "Calculation methods" form

The next form is the "System symmetry" in Figure 1-5. In this form select "Unsymmetrical system" and then click "Next" button to go to the next form.

Calculation methods
System symmetry:
Unsymmetrical system
Symmetrical system
Symmetrical system about y-axis Anti-symmetrical system in x-axis
Help     Save As     Cancel     < Back     Next>

Figure 1-5 "System symmetry" form

The last form in the wizard is the "Options" form, Figure 1-6. In this form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Since no option will be considered in the analysis, click the "Save" button.

Calculation methods
Options:         Slab with girders         Additional springs         Supports/Boundary conditions         Piled raft foundation         Determination of limit depth         Concrete design         Nonlinear subsoil model         Determination of stresses in soil         Determination of strains in soil         Determination of strains in soil         Determination of strains on the slab         T' Influence of neighboring foundations on the slab         O Influence of additional settlements on the slab         Select <u>All</u>
Help         Save As         Cancel         < Back         Next >         Save

Figure 1-6 "Options" form

After clicking "Save" button, the "Save as" dialog box appears, Figure 1-7. In this dialog box type a file name for the current project in "File name" edit box. For example type "Example". *ELPLA* will use automatically this file name in all reading and writing processes.

Save As				? 🗙
Save jn: 🔀	ELPLA PE 9.0	<b>-</b> +	£ 🗎	* 🎟 •
File <u>n</u> ame:	Example			<u>S</u> ave
Save as <u>t</u> ype:	Isolated slab foundation-files (*.P01)		•	Cancel

Figure 1-7 "Save as" dialog box

Click "Save" button to complete the definition of the calculation method and the file name of the project. *ELPLA* will activate the "Data" menu. Also the file name of the current project [Example] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, see Figure 1-8.

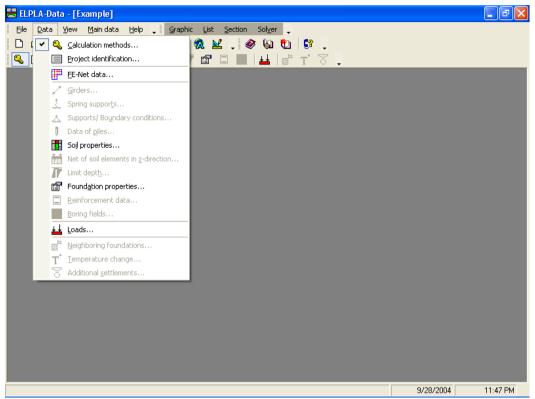


Figure 1-8 ELPLA-Data after defining the calculation method

In the "Data" menu of *ELPLA-Data*, the user can enter the remaining data of the project using the same sequence of commands in this menu. The first command in the menu is "Calculation methods", which has been already entered. Therefore, *ELPLA* has put the sign " $\sqrt{}$ " beside this command (Figure 1-8). *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.

### 2.2 **Project identification**

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project identification" command from "Data" menu. The dialog box in Figure 1-9 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of an irregular raft on irregular subsoil"
- Type the date of the project in the "Date" edit box
- Type the word "Example" in the "Project" edit box
- Click "Save" button

Project identification										
Project identification:										
Title	Title Analysis of an irregular raft on irregular subsoil									
Date	Tueso	day, September	28, 2004			•				
Project	Exam	ple								
Save	,	<u>C</u> ancel	He	p	Load	Save <u>A</u> s				

Figure 1-9 "Project identification" dialog box

### 2.3 FE-Net data

For the given problem, the raft has irregular shape and is divided into 15 \* 15 elements. Element size in both *x*- and *y*-directions is variable as shown in Figure 1-1. *ELPLA* has different procedures for defining the FE-Net. The easy procedure to define the FE-Net of this raft is generating a mesh for the entire area first and then removing the unnecessary nodes to get the foundation shape.

To define the FE-Net for this raft, choose "FE-Net data" command from the "Data" menu. "FE-Net generation" wizard appears as shown in Figure 1-10. This wizard will guide you through the steps required to generate a FE-Net. As shown in Figure 1-10, the first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

FE-Net generation
Slab type
Rectangular slab:
Length of rectangular slab L [m] 15.01
Width of rectangular slab B [m] 13.98
Help         Cancel         < Back         Next >         Einish

Figure 1-10 "FE-Net generation" wizard with "Slab type" form

To generate the FE-Net

- In the "Slab type" options choose the rectangular slab option
- In the "Rectangular slab" frame enter the total length and width of the raft in the corresponding edit boxes
- Click "Next" button to go to the next form

After clicking "Next" in "FE-Net generation" wizard, the following "Generation type" form appears, Figure 1-11. *ELPLA* can deal with various types of generations with triangle and/ or rectangular elements. Choose the first type of rectangular elements, then click "Next" button.

FE-Net generation		
Generation type		
······································		
Help	Cancel < <u>B</u> ack	<u>N</u> ext >Einish

Figure 1-11 "Generation type" form

The next form of the "FE-Net generation" wizard is the "Grid definition" form. In this form, the default values of constant element size appear, Figure 1-12. To edit the variable grid spacing in *x*-direction, do the following steps in "Grid in *x*-direction" frame:

- Uncheck the "Constant grid spacing" check box. "Grid spacing" button will be activated
- Click the button "Grid spacing". "Grid spacing in *x*-direction" list box appears, Figure 1-13. In this list box the spacing width can be entered individually for every space
- Enter the grid spacing in *x*-direction in this list box
- Click "OK" button to return to "Net of finite elements" dialog box

To edit the variable grid spacing in *y*-direction, repeat the previous steps. After that, click "Finish" button in "FE-Net generation" wizard to see the FE-Net on the screen.

FE-Net generation	
Grid definition	
Grids in x-direction	
Constant grid spacing	
No. of grid spaces	10 +
Grid spacing	1.00
Grids in y-direction	
Constant grid spacing	
No. of grid spaces	10 :
Grid spacing	1.00
Help	Cancel < <u>B</u> ack <u>N</u> ext > <u>F</u> inish

Figure 1-12 "Grid definition" form

C	Grid spa	cing in x-di	rect	ion 🛛 🔀
	No. T	Dx [m]		Qk
	1	0.90	_	Cancel
	2	0.90		
	3	0.90		Insert
	4	0.90		
	5	0.87		Copy
	6	0.87		
	7	0.87		Delete
	8	1.08		
	9	1.08		New
	10	1.08		Hole
	11	1.08		Help
	12	1.12	~	Excel
				EXOCI

Figure 1-13 "Grid spacing in *x*-direction" list box

### **Deleting nodes from the FE-Net**

To select the unnecessary nodes which are required to be removed from the net, first choose "Select nodes" command from "Graphically" menu. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair. The command "Remove nodes" in the menu "Graphically" will be enabled, indicating the mode in which is being operated. Next, select the required nodes by clicking on each node individually or selecting a group of nodes as shown in Figure 1-14. To remove the selected nodes, choose "Remove nodes" command from the "Graphically" menu. The action of this command is indicated in Figure 1-15. To leave the graphic mode, press "Esc" key.

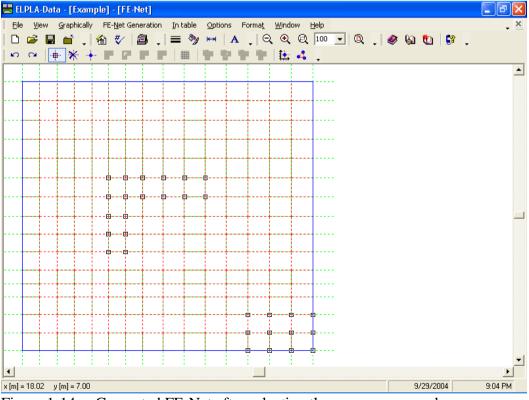


Figure 1-14 Generated FE-Net after selecting the unnecessary nodes

📇 EL	PLA	Dat	a - [l	Exam	iple]	- [[F	E-N	et]																	_ 6	
													Forma <u>t</u>					_								• ×
													. 9				- (	Q	•	<u>ک</u> (	<u>ا الأ</u>	1	63	-		
2	0	-	- *	÷ -		P 6	7			₩					4	÷										
																	_									-
																	-									
																	-									
									+				-+				-									
									ļ		ļ						-									
																	_									
																	-									
													-+				-									
								_	_	_							-									
									ļ	ļ							_									
									1		1	1					-									
																	_									
											-						-									
								İ	†	<u>†</u>	1						-									
																	-									-
•							i	;	i	i .	1	; ;	; 	;	i	;										•
x [m] =	16.7	2 у	[m] =	1.38																	9/2	29/20	04		9:04 P	M

Figure 1-15 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save FE-Net" command from "File" menu in Figure 1-15 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 1-15 to close the "FE-Net" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "FE-Net data" command in the "Data" menu of *ELPLA-Data*.

### 2.4 Soil properties

In *ELPLA* there are three different soil models with several calculation methods. Therefore, the soil properties for each method are required to be defined according to the used soil model. In the current example, the soil model, which is used in the analysis, is Layered Soil Model. This model requires that the subsoil has to be defined by boring logs. In the example, three boring logs at different locations define the soil under the raft. Each boring log has multi-layers with different soil materials. The geotechnical data for each layer are unit weight of the soil  $\gamma_s$  and modulus of Elasticity for loading  $E_s$  and reloading  $W_s$ .

To define the soil properties, choose "Soil properties" command from "Data" menu. The following sub program in Figure 1-16 appears with a default-boring log.

📅 ELPLA-Boring - [Example]	∎₽⊠
File View Data Graphically Options Format Window Main data Help	-
📓 🗖 = 🦻 🛏 A Q Q Q 💷 🖭 Q 🗄 🛠 .	
	-
-•• <u>·</u>	
	_
د. د. المتعربين من عن المتعربين من المتعربين من المتع	
	-
•	
9/29/2004	9:10 PM

Figure 1-16 *ELPLA-Boring* with a default-boring log

In Figure 1-16, soil properties are defined through the "Data" menu which contains the following two commands:

- "Soil data" command defines the individual boring logs
- "Main soil data" command defines the general data for all soil layers

To enter the soil properties for the three boring logs of the current example

- Choose "Soil data" command from "Data" menu in the window of Figure 1-16. The following dialog box in Figure 1-17 with default-boring log data appears

Soil data		$\mathbf{X}$
Boring log No. 1 from 3 boring logs: Layer No. 1 from 4 layers: Soil and rock symbols:	Geotechnical data of the layer:	1
Main soil type 1 U, Silt	Soil properties are defined by Modulus of Elasticity E	
Main soil type 2     -, No symbole       Submain soil 1     -, No symbole	E [kN/m2] 9500 Fhi ["] 30	
Submain soil 2 _, No symbole	VV [kN/m2] 26000 c [kN/m2] 5	
Color ol, olive	Gam [kN/m3] 19 Nue [-] 0.3	
	Layer depth under the ground surface [m] 1.50	
Layer <u>c</u> opy Layer insert La	ayer <u>d</u> elete	J
Boring log copy Boring log insert:	x-coordinate of boring log [m] 4.00	
From a file Boring insert	y-coordinate of boring log [m] 3.00	
Boring delete	Label of boring log B1	
	1	•  -
<u>Ok</u> <u>C</u> ancel	<u>N</u> ew <u>H</u> elp	

Figure 1-17 "Soil data" dialog box with default-boring log data

In the "Geotechnical data of the layer" dialog group box in Figure 1-17, define the geotechnical data of the first soil layer of the first boring log as follows:

$E_s$	= 9 500	$[kN/m^2]$
$W_s$	$= 26\ 000$	$[kN/m^2]$
Gam	= 19	$[kN/m^3]$

In the current example, the angle of internal friction  $\varphi$  and the cohesion *c* of the soil are not required because the selected type of the analysis is linear analysis. Therefore, the user can let the default values of the internal friction and the cohesion.

Due to the presence of the ground water, the soil above the ground water level has a differential unit weight from the soil under that level. Therefore, the layer depth of the first layer for all boring logs is taken to be 1.5 [m], which is equal to the ground water level. Now, type this value in "Layer depth under the ground surface" edit box.

In order to draw the soil layers by different symbols according to the German Standard DIN 4023, the soil type and color for each layer must be defined.

To define the soil type and color for the first layer, select "U, Silt" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box in Figure 1-17. The color of the silt according to the German Standard DIN 4023 will be automatically created. The user can change this color. Also, a short text "U" will be automatically created for the silt.

To enter the second layer of the first boring log

- Click "Layer copy" button in Figure 1-17. A layer that has the same properties of the first layer will be copied
- Use the vertical scrollbar to move to the second soil layer. Layer No. will be typed automatically at the upper-left corner of the main dialog box of soil layers as a head title
- Change the value of the unit weight of the soil for the second soil layer from 19 [kN/m<sup>3</sup>] to 9 [kN/m<sup>3</sup>]
- Change the value of the layer depth under the ground surface from 1.5 [m] to 3.8 [m]

To enter the fine sand and gravel layers

- Click twice "Layer insert" button in Figure 1-17, two layers will be inserted
- Use the vertical scrollbar to move to the third soil layer
- In "Geotechnical data of the layer" dialog group box in Figure 1-17, define the geotechnical data of the fine sand layer as follows:

Es	$= 22\ 000$	$[kN/m^2]$	Phi	= 30	[°]
Ws	= 52 000	$[kN/m^2]$	С	= 5	$[kN/m^2]$
Gam	= 9	$[kN/m^3]$			

- Select "fs, Fine sand" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box
- Type 10 in "Layer depth under the ground surface" edit box
- Use the vertical scrollbar to move to the fourth soil layer
- Type the following data for the gravel layer:

Es	$= 22\ 000$	$[kN/m^2]$	Phi	= 30	[°]
Ws	= 52 000	$[kN/m^2]$	С	= 5	$[kN/m^2]$
Gam	= 9	$[kN/m^3]$			

- Select "G, Gravel" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box
- Type 20 in the "Layer depth under the ground surface" edit box

Note that the unit weight of the soil is used to determine the overburden pressure  $q_v$  [kN/m<sup>2</sup>] due to the removed soil, which is equal to  $\gamma_s * d_f$ . This means that the unit weight of the soil under the foundation depth  $d_f$  is not required. However, the unit weight of the soil under the foundation depth for all soil layers is entered by the value 9 [kN/m<sup>3</sup>].

After editing the geotechnical data for the first boring log, the boring coordinates and labels which describe the boring will be entered.

To enter the boring coordinates and the label

- Type 4 for *x*-coordinate in "*x*-coordinate of boring log [m]" edit box
- Type 3 for y-coordinate in "y-coordinate of boring log [m]" edit box
- Type B1 as a label name for the first boring in "Label of boring log" edit box

Now all data and parameters for the first boring log have been entered. The next step is to enter the data of the other two boring logs. As the three boring logs contain the same soil layers, data of the other two boring logs are created by first copying the data of the first boring log and then modifying boring logs individually. Only layer depths, boring coordinates and labels are required to be modified.

To create the other two boring logs, click twice "Boring log copy" button in Figure 1-17. Two boring logs with the same data and parameters of the first boring log will be copied.

### Modifying data of boring logs

Modifying boring coordinates is carried out only numerically while modifying the other data of boring logs may be carried out either numerically or graphically. In this example all data will be modified numerically.

To modify the boring coordinates and labels

- Use the horizontal scrollbar to switch to the second boring log. Boring log No. will be typed automatically at the upper-left corner of the main dialog box of boring logs as a head title
- Type 1 as x-coordinate in "x-coordinate of boring log [m]" edit box in Figure 1-17
- Type 9 as y-coordinate in "y-coordinate of boring log [m]" edit box in Figure 1-17
- Type B2 as a label name for the second boring in "Label of boring log" dialog box in Figure 1-17
- Use the vertical scrollbar to move from a layer to another. Then modify the layer depth under the ground surface for each layer
- In "Layer depth under the ground surface" edit box in Figure 1-17 type the following values for layer depths:

Layer depth under the ground surface (2 <sup>nd</sup> layer)	8.2	[m]
Layer depth under the ground surface (3 <sup>rd</sup> layer)	14.1	[m]
Layer depth under the ground surface (4 <sup>th</sup> layer)	20	[m]

Repeat the previous steps to modify the boring data for the third boring log. Data, which are required to be modified for the third boring log, are:

<i>x</i> -coordinate of boring log	10	[m]
y-coordinate of boring log	11	[m]
Label of boring log	B3	
Layer depth under the ground surface (2 <sup>nd</sup> layer)	12.7	[m]
Layer depth under the ground surface (3 <sup>rd</sup> layer)	18.2	[m]
Layer depth under the ground surface (4 <sup>th</sup> layer)	20	[m]

Now, after finishing the creation of boring logs, click "OK" button in "Soil data" dialog box in Figure 1-17 to see the defined boring logs on the screen where the user can control or modify the input data and parameters. As a default plot parameter, *ELPLA* displays only the first boring log on the screen (Figure 1-18).

Ele yew Data Graphically Options Format Window Maindata Help	ELPLA-Boring - [Example]				- 7	×
			Help			÷
	E 2 ↓ = 3 → A	. ♥ ७ ७ ₿ . . ୧ ୧ ୧ ₪ . ୯	S . * .			
		stant power, we see the second s				
		aanganada, no - 30 (). Aanganada, no				
	<u></u> =					<u> </u>
▲ 9/29/2004 9:13 PM	•		]	0/20/2004		1

Figure 1-18 First boring log on the screen

To display all boring logs or specified boring logs on the screen, choose "Drawing boring logs" command from "Graphically" menu in Figure 1-18. The following list box in Figure 1-19 appears.

To select the boring logs you want to display

- Select the boring log that is required to be displayed from the list of the available boring logs in Figure 1-19
- Click "Boring insert" button. Double clicking on the required boring log in the list of the available boring logs gives the same action. Removing a boring log from the drawing list is carried out by double clicking on that boring log in the list of the selected boring logs
- Click "OK" button in Figure 1-19. The selected boring logs appear on the screen to control or modify the boring data graphically, Figure 1-20

	boring log			Þ
-List of	f selected boi	ring logs t	to draw:	Ok
No.	Boring log N	lo.	Label of boring log	
1		1 B1		
				<u>C</u> ancel
				Help
List of	f the available	boring lo	ogs:	New
	f the available	boring la	ogs:	
			-	 New
	ng log No.	-	-	
	ng log No. 1 E	1	-	 New

Figure 1-19 "List of boring logs" list box

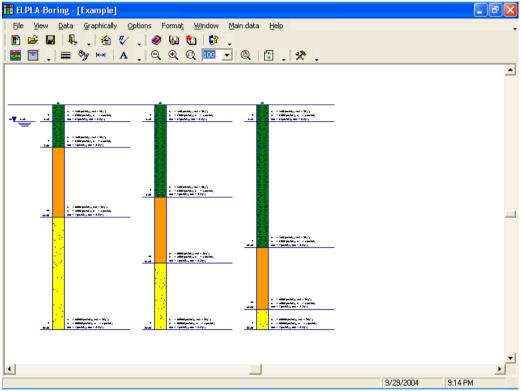


Figure 1-20 Boring logs on the screen

To enter the main soil data for all layers, choose "Main soil data" command from "Data" menu in Figure 1-20. The following dialog box in Figure 1-21 appears with default main soil data. The main soil data for the current example, which are required to be defined, are the settlement reduction factor  $\alpha$  [-] and the groundwater depth under the ground surface  $G_w$  [m]. Any other data corresponding to main soil data are not required in this example. Therefore, the user can take these data from the default soil properties.

In the dialog box of Figure 1-21 enter the settlement reduction factor  $\alpha$  [-] and the groundwater depth under the ground surface  $G_w$  [m]. Then click "OK" button.

Main soil data			
Soil properties Calculation parameters of flexibility coe	efficients Bearing cap	oacity factors	
Main soil data:			
Settlement reduction factor Alfa <= 1	Alfa	[-]	1
Groundwater depth under the ground surface	Gw	[m]	1.50
	Help	1	

Figure 1-21 "Main soil data" dialog box

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

- Choose "Save boring logs" command from "File" menu in Figure 1-20 to save the data of boring logs
- Choose "Close boring logs" command from "File" menu in Figure 1-20 to close the sub program *ELPLA-Boring* and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Soil properties" command in the "Data" menu of *ELPLA-Data*.

### 2.5 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu. The following embedded program in Figure 1-22 appears with default foundation properties. The data of foundation properties for the current example, which are required to be defined, are raft material, raft thickness and foundation depth. Any other data corresponding to foundation properties in the program menus are not required in this example. Therefore, the user can take these data from the default foundation properties.

To enter the raft material and thickness, choose "Element groups" command from "In Table" menu. The following list box in Figure 1-23 appears. In this list box, enter E-Modulus of the raft, *Poisson's* ratio of the raft and raft thickness. Then click "OK" button to go to the next step.

### ELPLA-Tutorial

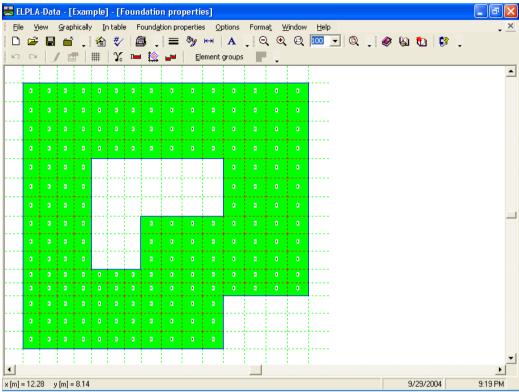


Bild 1-22 "Foundation properties" embedded program

I	Defining	element grou	ps (with the sa	me thickness an	d slab mate 🔀
	Group No.	E-Modulus of slab [kN/m2]	Poisson's ratio of slab [-]	Slab thickness d [m]	<u>Ok</u> Cancel
	1	2E+07	0.25	0.5	
					Insert
					Сору
					Delete
					New
					Help
					Excel

Figure 1-23 "Defining element groups" list box

To enter the unit weight of the raft, choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 1-22. The following dialog box in Figure 1-24 with a default unit weight of 25  $[kN/m^3]$  appears. To neglect the self-weight of the raft in the analysis, type 0 in the "Unit weight of the foundation" edit box. Click "OK" button.

Unit weight of the foundation	
Unit weight of the foundation:	Gb [kN/m3] 0
<u> </u>	<u>Cancel</u> <u>H</u> elp

Figure 1-24 "Unit weight of the foundation" dialog box

To enter the foundation depth under the ground surface, choose "Foundation depth" command from "Foundation properties" menu in the window of Figure 1-22. The following dialog box in Figure 1-25 appears to define the foundation depth under the ground surface. Type 2.7 in the "Foundation depth under the ground surface" edit box. Then click "OK" button.

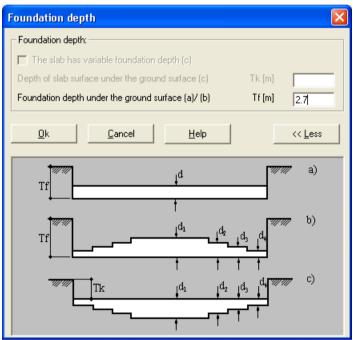


Figure 1-25 "Foundation depth" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 1-22 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 1-22 to close the "Foundation properties" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside "Foundation properties" command in "Data" menu of *ELPLA-Data*.

### 2.6 Boring fields

If the subsoil under the raft is defined by two boring logs or more such as in the current example, the irregularity of the subsoil must be taken into account. "Boring fields" command let the user define which method is to be used to consider the irregularity of the subsoil. In the current example, the Interpolation Method will be used.

To consider the Interpolation Method in the analysis, choose "Boring fields" command from "Data" menu. The following embedded program in Figure 1-26 appears with a default method. *ELPLA* considers that the Interpolation Method is the default one, which takes into account the irregularity of the subsoil. In most cases *ELPLA* defines the interpolation zone types I, II, III automatically such as in this example. But in the case of extreme boring arrangements, the user must define these zones.

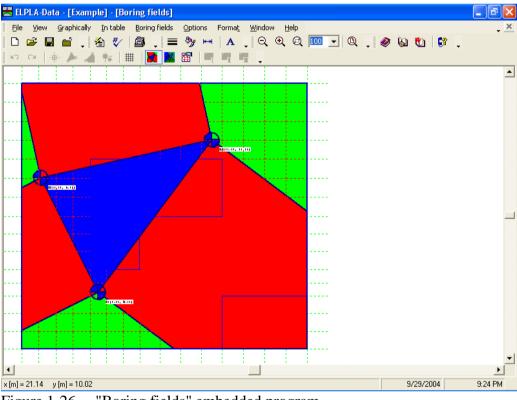


Figure 1-26 "Boring fields" embedded program

You do not need to change anything. Now do the following two steps:

- Choose "Save boring fields" command from "File" menu in Figure 1-26 to save the data of boring fields
- Choose "Close boring fields" command from "File" menu in Figure 1-26 to close the "Boring fields" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside "Boring fields" command in "Data" menu of *ELPLA-Data*.

### 2.7 Loads

In *ELPLA*, loads on the raft such as point loads, line loads, uniform loads or moments may be applied to the net of the finite elements at any position independently on the node position. The coordinates of the loads are related to the lower-left corner of the raft (local coordinates).

To define the loads, choose "Loads" command from "Data" menu. The following embedded program in Figure 1-27 appears.

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Graphically" menu in Figure 1-27. In this example the user will learn the definition of loads numerically.

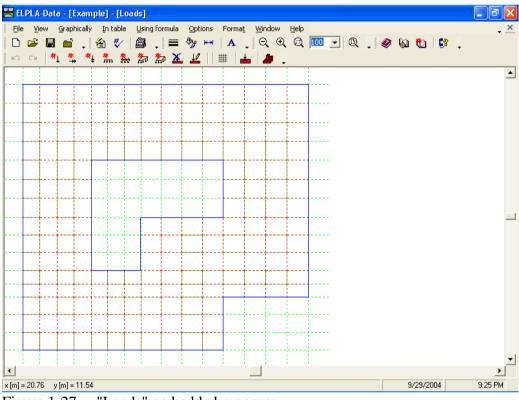


Figure 1-27 "Loads" embedded program

To enter point loads

- Choose "Point loads" command from "In Table" menu in the window of Figure 1-27. The following list box in Figure 1-28 appears. *ELPLA* can distribute concentrated loads under columns. In this example, data corresponding to column dimensions are not required. Therefore, the user can take these data from the default column dimensions and consider all loads have column type 1
- Enter the external point loads P [kN] and their corresponding coordinates (x, y) in the list box of Figure 1-28 by typing the value in the corresponding cell and pressing "Enter" key
- The coordinates of the point load are related to the lower-left corner of the raft (local coordinates)
- Click "OK" button

Repeat the previous steps for moments Mx, moments My, line loads and distributed loads using "Moments Mx", "Moments My", "Line loads" and "Distributed loads" commands from "In Table" menu respectively. After that the screen should look like the following Figure 1-29.

Point loa	ads					
No. I [-]	Column types I	Load P [kN]	x-position × [m]	y-position y [m]	^	
1	1	1265.0	1.50	1.40		Cancel
2	1	1600.0	1.50	5.50		Insert
3	1	1350.0	1.50	9.90		· · ·
4	1	1368.0	1.50	12.60		Copy
5	1	1560.0	5.00	1.40		
6	1	1538.0	5.00	12.60		<u>D</u> elete
7	1	800.0	9.20	1.40		
8	1	750.0	9.20	5.50		New
9	1	1565.0	9.20	12.60		
10	1	2150.0	13.40	5.50		Help
11	1	1450.0	13.40	9.90		
40	1	4064.0	42.40	40.60	×	Excel

Figure 1-28 "Point loads *P*" List box

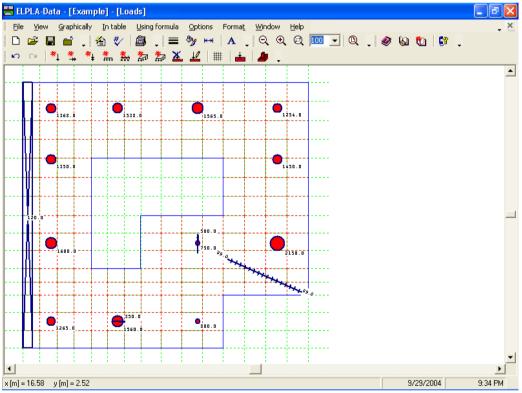


Figure 1-29 Loads on the screen

To save the load data, choose "Save loads" command from "File" menu. Then choose "Close loads" command from this menu to close "Loads" embedded program and return to the main window of *ELPLA-Data*. Note that the sign " $\sqrt{}$ " is typed automatically beside the "Loads" command in the "Data" menu of *ELPLA-Data*.

Creating the project of the raft is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

### **3** Carrying out the calculations

### 3.1 Starting *ELPLA-Solver*

To analyze the problem which you have just defined, leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar at the upper-right corner of *ELPLA-Data*. *ELPLA-Solver* window appears, Figure 1-30. This window belongs to *ELPLA-Solver*. Like *ELPLA-Data*, on the upper-right corner of *ELPLA-Solver* window appears the menu bar of the sub programs, which are used for switching between individual sub programs of *ELPLA* package. On the upper-left corner of this window appears the menu bar of *ELPLA-Solver*, which is used for analyzing the problem.

In Figure 1-30, the "Calculation" menu is active. This menu contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For the current example, the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Iteration process
- Determining deformation, internal forces, contact pressures

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

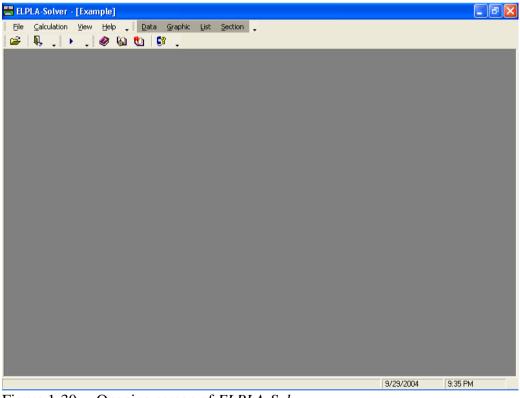


Figure 1-30 Opening screen of ELPLA-Solver

### **3.2** Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Calculation" menu in *ELPLA-Solver* Window. The following "Iteration parameters" option box in Figure 1-31 appears
- In "Iteration parameters" option box, select which option is ending the iteration process. For this example, choose an accuracy of 0.0001 [m] to end the iteration process
- Click "OK" button

Iteration parameters					
Which option is ending the iteration	n process?				
Accuracy [m]	0.0001				
C Iteration No.	10				
Qk <u>C</u> ancel	Help				

Figure 1-31 "Iteration parameters" option box

The progress of all computations according to the defined method will be carried out automatically with displaying information through menus and messages.

#### Analysis progress

Analysis progress menu in Figure 1-32 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

Determining flexibility coefficients of the soil
Assembling the flexibility matrix of the slab 1!
Time remaining = 00:00:00
I = 122 from 145 steps

Figure 1-32 Analysis progress menu

#### **Iteration process**

Information about the convergence progress of the computations is displayed in the "Iteration process" list box in Figure 1-33 during the iteration process.

Iteration process					
Iteration No.	Accuracy [m]	Stop			
1	0.02564279000	]			
2	0.00305188200	Pause			
3	0.00107387100				
Iteration cycles is ended at accuracy [m]<= 0.0001					
Computation time = 00:00:04					

Figure 1-33 "Iteration process" list box

### Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 1-34. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

Check of the solutio	n		
V - Load:			
Total load		[kN] =	15895.2
Sum of contact pressu	res	[kN] =	15891.5
X - Moment:			
Sum Mx from loads		[kN.m] =	7035.1
Sum M× from contact p	ressures	[kN.m] =	7036.9
Y - Moment:			
Sum My from loads		[kN.m] =	-6679.1
Sum My from contact p	ressures	[kN.m] =	-6683.4
Ok	Help		

Figure 1-34 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

### 4 Viewing data and results

*ELPLA* can view and print a wide variety of results in graphics, diagrams or tables through the three sub programs *ELPLA-Graphic*, *ELPLA-Section* and *ELPLA-List*. Data can also be viewed again and printed by the sub programs *ELPLA-Graphic* and *ELPLA-List*. Note that *ELPLA-Data* is used only to define and view the data of the problem. *ELPLA-Graphic* is used to print data graphically while *ELPLA-List* is used to print data numerically.

#### 4.1 Viewing data and result graphics

To view graphically the data and results of a problem that has already been defined and analyzed. switch to ELPLA-Graphic. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of ELPLA-Solver window. ELPLA-Graphic window appears, Figure 1-35. This window belongs to the sub program ELPLA-Graphic. Like in other sub programs of ELPLA, on the upper-right corner of ELPLA-Graphic window appears the menu bar, which is used for switching between individual sub programs of the ELPLA package. On the upper-left corner of this window appears the menu bar of ELPLA-Graphic, which is used for displaying the data and results.

🗮 ELPLA-Graphic - [	Example]		
	Options Format Window Help 🚛 Data List Section Solver 🗸		
	K ∕a ♥   D @ L. = #   b   # .   < < < ∞ .		
I • 💋 • ## •	⊢ • ● •   ⇔ ४ क़ •   ∿ ♡ + @   ₩ • ⊞ •   ◙   圖	± - ₩ - y	/
		9/29/2004	9:57 PM
		972972014	19'57 PM

Opening screen of the sub program *ELPLA-Graphic* 

The "Graphic" menu contains the commands of drawing. These commands depend on the used calculation method in the analysis. For the current example, the commands for presenting the data and results are:

- Results in isometric view
- Results as contour lines \_
- Result values in the plan \_
- Distribution of results in the plan
- Results as circular diagrams -
- Slab deformation

- Principal moments as streaks \_
- Data in isometric view \_
- Data in the plan -
- **Boring** locations -
- Boring logs \_

Only the first command of the "Graphic" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Options", "Format" and "Window" menus, which are used to define the preferences of the drawing such as plot parameters, scale, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

To view the results in isometric view

- Choose "Results in isometric view" command from "Graphic" menu. The following option box in Figure 1-36 appears
- In the "Results in isometric view" option box, select "Contact pressures q" as an example for the results to be displayed
- Click "OK" button

The contact pressures are now displayed in an isometric view as shown in Figure 1-37.

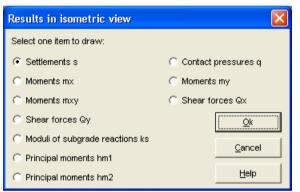


Figure 1-36 "Results in isometric view" option box

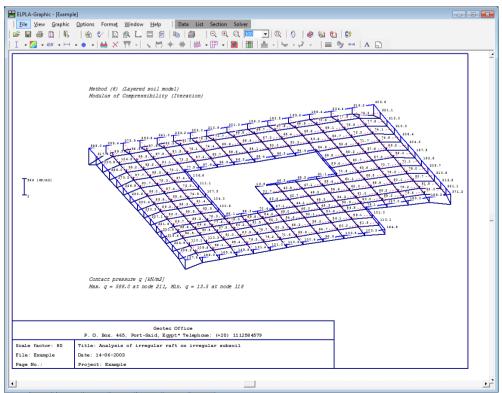


Figure 1-37 Contact pressures in an isometric view

### 4.2 Plot a diagram of the results at a specified section

To plot a diagram of the results, switch to *ELPLA-Section*. This is done by clicking on "Section" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Graphic* window. *ELPLA-Section* window appears, Figure 1-38. The function of *ELPLA-Section* is plotting and printing the results in diagrams. *ELPLA-Section* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Section* window.

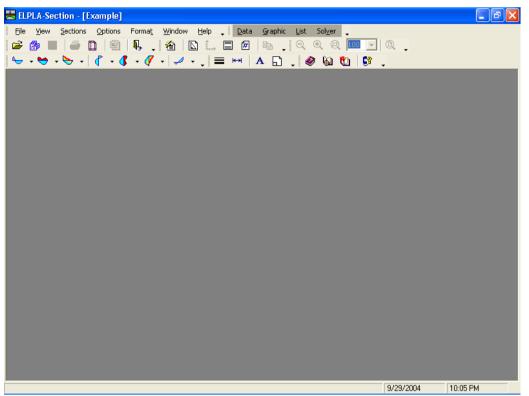


Figure 1-38 Opening screen of the sub program ELPLA-Section

The "Section" menu in *ELPLA-Section* contains the commands of drawing the diagrams. The commands for presenting the results in diagrams are:

- Section in *x*-direction Section in *y*-direction
- Max./ Min. values in x-direction
- Overlapping in *x*-direction

- Max./ Min. values in y-direction
- Overlapping in y-direction

- Arbitrary section

Only the first command of the "Sections" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Options", "Format" and "Window" menus, which are used to define the preferences of the drawing such as plot parameters, scale, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

To plot a section in *x*-direction

- Choose "Section in *x*-direction" command from "Sections" menu. The following option box in Figure 1-39 appears
- In the "Section in *x*-direction" option box, select "Settlements *s*" as an example for the results to be plotted in a diagram
- Click "OK" button

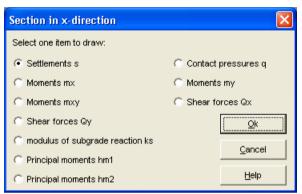


Figure 1-39 "Section in *x*-direction" option box

The following dialog box in Figure 1-40 appears to specify the section in x-direction. In this dialog box, click "OK" button to plot the default section. The settlements are now plotted in a diagram as shown in Figure 1-41.

Section in x-direction	
Section in x-direction:	
Section at y-coordinate	(m) (þ.00
Set range in x-direction:	
Start range at x-coordinate	[m] 0.00
End range at x-coordinate	[m] 15.01
<u>Ok</u> <u>Cancel</u> <u>H</u> elp << <u>L</u> ess	

Figure 1-40 "Section in *x*-direction" dialog box with a default section

### ELPLA-Tutorial

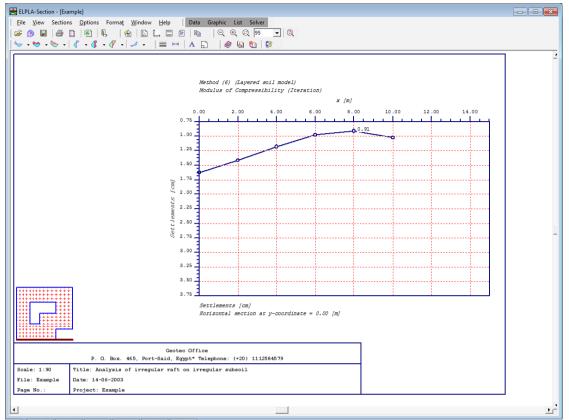


Figure 1-41 Diagram of settlements in x-direction

### 4.3 Print the drawing

To print the drawing

- Choose "Print" command from "File" menu in *ELPLA-Graphic* or *ELPLA-Section* Window. The following "Print" dialog box in Figure 1-42 appears
- In the "Print" dialog box, define the printer properties and drawing copies
- Click "OK" button

Only the objects currently displayed on the drawing are printed.

🌢 Print	? 🛛
General	
Select Printer	
Add Printer Epson Stylus COLOR 440 ESC/P 2	
Status: Ready	Print to file Preferences
Location: Comment:	Fin <u>d</u> Printer
Page Range	
⊙ All	Number of <u>c</u> opies: 1
C Selection C Current Page	
	Print Cancel

Figure 1-42 "Print" dialog box

### 4.4 Listing data and results in tables

To list tables of data and results, switch to *ELPLA-List*. This is done by clicking on "List" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Section* window. *ELPLA-List* window appears, Figure 1-43. The function of *ELPLA-List* is listing and printing data and results in tables. Data and results can be exported to other Windows applications to prepare reports or add further information. *ELPLA-List* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-List* window.

The "List" menu in *ELPLA-List* contains the commands of listing data and results. The commands for listing data and results in tables are:

-	Display tables of data Print tables of data List tables of data through Text- Editor	-	Display tables of results Print tables of results List tables of results through Text- Editor
	Editor		Editor

Only the first command of the "List" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Format" and "Window" menus, which are used to define the preferences of the tables such as page format, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

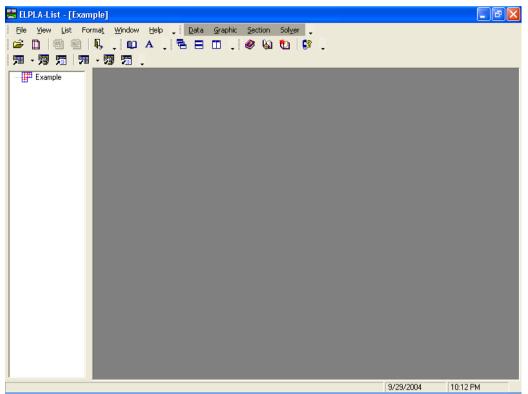


Figure 1-43 Opening screen of the sub program ELPLA-List

To list data in a table

- Choose "Display tables of data" command from "List" menu. The following option box in Figure 1-44 appears
- In the "Display tables of data" option box, select "Loading" as an example for the data to be listed in a table
- Click "OK" button. The loading data are now listed on the screen (Figure 1-45)
- Choose "Send to Word" from "File" menu if you wish to export the table to a MS Word application, Figure 1-46

Display tables of data	
Select one item to list:	
Node coordinates	C Connectivity Nodes
C Soil properties	Qk
C Slab properties/ Foundation level/ Global coordinates	Cancel
C Data of boring fields	
C Loading	Help

Figure 1-44 "Display tables of data" option box

### ELPLA-Tutorial

■・腎 短 見	· 🛛 🗖 📜		J 🧶 🖗 🛍	- •		
Example	🖶 Loading					
toduling	Loadi	ng				~
	Column and		ame properties):			
		es (wich che sa				
	Group No. I	Column side				
	[-]	a [m]	ь [m]			
	1	0	0			
	Point load	15: 				
			Load value	-		
	I [-]	I [-]	-	x [m]	У [m]	
	1 2	1	1265 1600	1.5 1.5		
	3	1	1350	1.5		
	4	1	1368	1.5	12.6	
	5	1	1560 1538	5	1.4 12.6	
	7	1	800	9.2	1.4	×
	<					>



	<b>d - test (Read-On</b> w <u>I</u> nsert F <u>o</u> rmat		/indow Help		_
	• <u>•</u> <u>*</u>				
ndard 💌	Courier New	- 8 -	BIU	▙▎ॾॾॾॾऻॾॾॾॿऺ	
	2 • 1 • 3 • 1 • 4 •	1 · 5 · 1 · 6 · 1 ·	7 • 1 • 8 • 1 • 9 •	···10····111····12····13····14····15····人···17····18·	
Loadi Point load					
Logd No	Load value				
ICACINO: I	p p	x posición x	y posición y		
[-]	[kN]		[m]		
1	1265	1,5	1,4		
2	1600	1,5	5,5		
3	1350	1,5	9,9		
4	1368	1,5	12,6		
5	1560	-,-	1,4		
6	1538	5	12,6		
7	800	9,2	1,4		
8	750	9,2	5,5		
9	1565	9,2	12,6		
10	2150	13,4	5,5		
11	1450	13,4	9,9		
12	1254	13,4	12,6		
Moments Mx					
	oment value	x-position	y-position		
		x	У		
[-]	[kN.m]	[m]	[m]		
	350	5	1,4		

Figure 1-46 Exported data in MS Word

### 5 Index

# A

Analysis type7
----------------

# B

Boring fields					24
Boring logs	4,	14,	16,	18,	19

# C

Calculation	27, 28
Calculation methods	. 7, 14, 27
Check of the solution	
Computation of all	
Contact pressures	

# D

Deformation	27
Diagrams	29, 32

### F

FE-Net	
Flexibility coefficients	
Foundation depth	
Foundation properties	
1 1	

# G

Generation type	12
Graphics	30
Grid spacing	12
1 0	

# I

Internal forces	27
Iteration process	27, 28

# L

Load vector	27
Loads	25

# M

Main soil data	15, 20
Max./ Min. values in x-direction	
Max./ Min. values in y-direction	
Moments	

# $\overline{N}$

New	project	7
-----	---------	---

### 0

# P

Point loads	5,	25
Project identification		10

# R

Raft material	3, 21
Remove nodes	13
Results	30, 31

### S

Save as	9
Section in x-direction	
Section in y-direction	
Select nodes	
Soil model	
Soil properties	14, 15, 20
Soil stiffness matrix	
System symmetry	

# U

Unit weight of the raft 22
----------------------------

# Example 2

# Analysis of a slab floor

### Contents

# Page

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

#### **1** Description of the problem

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

#### 1.1 Loads and dimensions

The slab floor has a thickness of 10 [cm] and carries uniform loads with different intensities as shown in Figure 2-1. All girders have the same dimensions of 15 [cm] \* 60 [cm]. Own weight of the girder may be taken as 1.875 [kN/m].

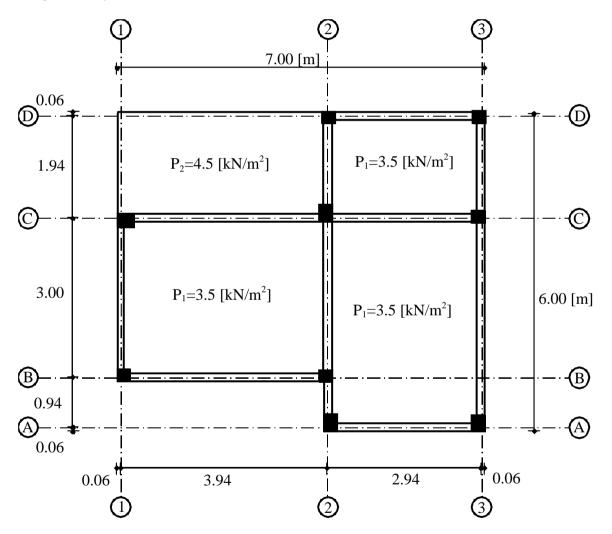


Figure 2-1 Dimensions of the slab with loads

#### 1.2 Slab material

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

Young's modulus of concrete	$E_b$	$= 3.2 * 10^7$	$[kN/m^2]$
Poisson's ratio of concrete	Vb	= 0.20	[-]
Unit weight of concrete	$\gamma_b$	= 25	$[kN/m^3]$
Shear modulus of concrete	$G_b = 0.5 E_b \left(1 + \mathbf{v}_b\right)$	$= 1.3 * 10^{7}$	$[kN/m^2]$

### 1.3 Analysis and concrete design

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

Concrete grade	C 30/37		
Steel grade	BSt 500		
Characteristic compr	essive cylinder strength of concrete $f_{ck}$	= 30	$[MN/m^2]$
Characteristic tensile	yield strength of reinforcement $f_{yk}$	= 500	$[MN/m^2]$
Partial safety factor f	for concrete strength $\gamma_c$	= 1.5	[-]
Design concrete com	pressive strength $f_{cd} = f_{ck} / \gamma_c$	= 30/1.5 = 20	$[MN/m^2]$
Partial safety factor f	for steel strength $\gamma_s$	= 1.15	[-]
Design tensile yield s	strength of reinforcing steel $f_{vd} = f_{vk} / \gamma_s$	= 500/1.15 = 435	$[MN/m^2]$

This Tutorial Manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for numerical calculation methods is well documented in the User's Guide of *ELPLA*.

### 2 Creating the project

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

### 2.1 Calculation method

To create the project start the sub program *ELPLA-Data* and choose "New project" command from "File" menu. The "Calculation method" wizard appears, Figure 2-2. As shown in this Figure, the first form of the wizard is the "Analysis type" form.

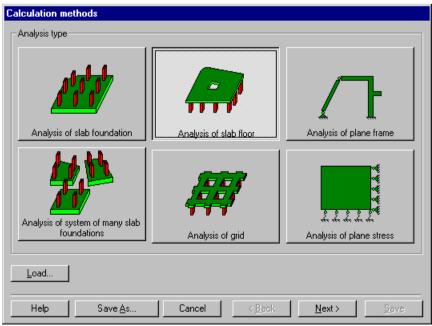


Figure 2-2 "Calculation method" wizard with "Analysis type" form

In the "Analysis type" form in Figure 2-2, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a slab floor problem, select "Analysis of slab floor" then click "Next" button to go to the next form. The next form is the "System symmetry", Figure 2-3.

In this form

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

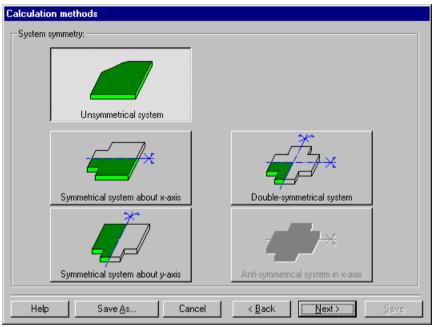


Figure 2-3 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 2-4. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other.

In this list

- Check "Supports/ Boundary conditions" check box
- Check "Slab with girders" check box
- Check "Concrete design" check box
- Click "Save" button

culation methods
· V 🗸 Slab with girders
🙏 Addtional springs
Piled raft foundation
Determining limit depth
Concrete design
Nonlinear subsoil model
Determining displacements in soil
Determining stresses in soil
Determining strains in soil
Influence of neighboring foundations on the slab
Influence of temperature change on the slab
Influence of additional settlements on the slab
Select All
Help Save As Cancel < Back Next> Save

Figure 2-4 "Options" list

After clicking "Save" button, the "Save as" dialog box in Figure 2-5 appears. In this dialog box

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

Save As				? ×
Savejn: 🔁	ELPLA PE 8.0	•	💣 🎟 •	
Example				
Floor Raft 1				
Raft 2				
File <u>n</u> ame:	Floor		<u>S</u> ave	е
Save as <u>t</u> ype:	Isolated slab foundation-files (*.P01)	•	Canc	el

Figure 2-5 "Save as" dialog box

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

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

🗮 ELPLA-Data - [Floor] \_ 8 × File Data View Main data Help 📲 Graphic List Section Solver 🖕 🔕 🔟 🤌 🚬 😡 🛍 💱 🗋 🚺 🔍 🔍 Calculation methods... ¥ Г 🗳 🗖 📕 🖬 🖬 т' 🏹 S ( Project identification... E-Net data... 🖌 <u>G</u>irders... 🧶 Spring supports... 🙏 Supports/ Boundary conditions... piles.. Sojl properties... Net of soil elements in z-direction ... T Limit depth. Slab properties... Beinforcement... Boring fields.. <u>↓↓</u> Loads... B<sup>B:</sup> Neighboring foundations...  $\boldsymbol{T}^{^{*}}$ Temperature change.. Additional settlements. Figure 2-6 ELPLA-Data after defining the calculation method

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

### 2.2 **Project identification**

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

In this dialog box

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

P	roject ide	Intification	х
	– Project id	entification:	
	Title	Analysis of a slab floor	
	Date	Mondy, 14. June 2003	
	Project	Slab floor	
	<u>S</u> ave	<u>C</u> ancel <u>H</u> elp <u>L</u> oad Save <u>A</u> s	

Figure 2-7 "Project identification" dialog box

### 2.3 FE-Net data

For the given problem, the slab has irregular shape and is divided into 7 \* 6 elements. Element size in both *x*- and *y*-directions is 1.0 [m] as shown in Figure 2-1. *ELPLA* has different procedures for defining the same problem. The easy procedure to define the FE-Net of this slab is generating a mesh for the entire area first and then removing the unnecessary nodes to get the slab shape.

To define the FE-Net for this slab, choose "FE-Net data" command from "Data" menu. The "FE-Net generation" wizard appears as shown in Figure 2-8. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 2-8, the first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have constant size in both *x*- and *y*-directions.

To generate the FE-Net

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

FE-Net generation	
Slab type	
Rectangular slab:	
Length of rectangular slab	L [m] 7
Width of rectangular slab	B [m] 6
Help Cancel < Bac	k. <u>N</u> ext≻ <u>F</u> inish

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

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

In the "Generation type" form

- Choose rectangular elements
- Click "Next"

FE-Net generation			
Generation type			
		× + × + × + × × + × + × + ×	
	Cancel	< <u>B</u> ack <u>N</u> ext >	• <u>Finish</u>

Figure 2-9 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "Grid definition" dialog box in Figure 2-10 appears with default values of constant element size.

In this dialog box

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

*ELPLA* will generate a FE-Net for a rectangular slab of 7 [m] length and 6 [m] width with square elements of 1.0 [m] each side. The following embedded program in Figure 2-11 appears with the generated net.

FE-Net generation			
Grid definition			
Grids in x-direction			
Constant grid interval			
No. of grid intervals	7 🛨		
Grid interval Dx [m]	1,00		
Grids in y-direction			
Constant grid interval			
No. of grid intervals	6		
Grid interval Dy [m]	1,00		
Help	Cancel < <u>B</u> ack	<u>N</u> ext >	<u> </u>

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

			-	<b>- (FE-</b> ly FE-		eneratio	n <u>I</u> r	table	<u>O</u> pl	tions	Formaj	: <u>₩</u> i	ndow	<u>H</u> el	р							_ 8 _ 1
2	<b>2</b>	8 6	5	쇱	#∕	ا 🛋	. =	- 🏷	7 <del>K N</del>	A	. į	e,	<b>e</b> (	<b>2</b> [1	.00	-   @	Q 📜	۲	6	٩	6	
7			* -	-	12	Ρ.		#		Р.	•		높 🔹	1								
-																			-			
																			_			
							_												_			
																			_			
																			-			
-																			-			
														_					-			
							į		į		_	-				į						

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

### **Deleting nodes from the FE-Net**

To select the unnecessary nodes, that are required to be removed from the net, first choose "Select nodes" command from the "Graphically" menu in Figure 2-12. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair. The command "Remove nodes" in the menu "Graphically" will be enabled, indicating the mode in which is being operated. Next, select the required nodes by clicking on each node individually or selecting a group of nodes as shown in Figure 2-12. A group of nodes can be selected by holding the left mouse button down at the corner of the region. Then, drag the mouse until a rectangle encompasses the required group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

_		) ata - [F													_ 🗗 🗙
<u>F</u> ile	. ⊻ie - ⊃	w <u>G</u> rapi	hically	FE- <u>N</u> et 0	Generation	i <u>I</u> nitable	e <u>O</u> ption	s Forma <u>t</u>	<u>W</u> indow			3 <b>~</b> ^			* <u>×</u>
	~		、 往	∕∜ נ	- <b>-</b>		y ⊷     ■ ■		ચ્ચ્ય	2 100 -	la a	, 🧶 🗞	1 🚺   🖗	ſ 🌲	
- 1	( .a	<b>۲ <u>اعتا</u> ا</b>	V 1 <b>7</b> 1						-   🎎	₩ ¥					
F	<u>.</u>		1	_ <u>_</u>		<u>ė                                    </u>									
	+-1	L+	1								-				-
•									]						

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

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

	w <u>G</u> ra										م ال	1 2	<u>ک</u> ا	<b>*</b>	<u>1</u> 2	•
			⊽⁄   ł ₽	•	!= ` !   ∰	″″,⊷							Ø 😡		<b>\$</b> ? .	
		<u> </u>			1			1007	-	*		_			_	
1											- i					
-	 		 				 									
-	 		 +				 									
-	 	1	 1													
-	 		 				 									
1							 									
	 	<u> </u>	 <u> </u>													

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

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

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

### 2.4 Girders

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

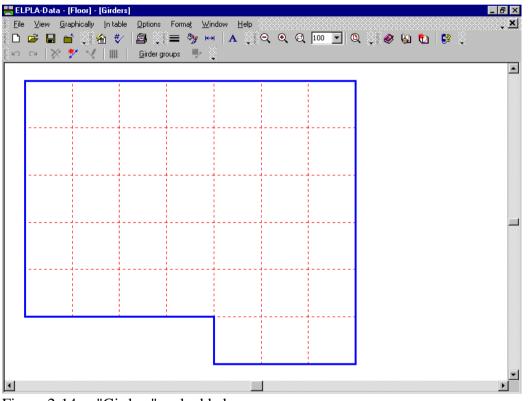


Figure 2-14 "Girders" embedded program

To enter the cross section of the girders

- Choose "Girder groups" command from "In Table" menu in Figure 2-14. The following option box in Figure 2-15 appears
- In this option box, select the option of cross section definition. Although the cross section of the girder must be defined whether it is T or L girder type, but for simplicity a rectangular cross section is chosen in this example to define the girder cross section
- Click "OK" button

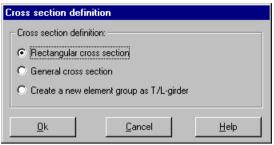


Figure 2-15 "Cross section definition" option box

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

In this list box

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

rder gra	oups (with th	e same prop	perties)			
Group No.	E-Modulus of girder E [kN/m2]	G-Modul of girder G [kN/m2]	Height ofgirder h [m]	Width of girder b [m]	Girder weight pb [kN/m]	<u>O</u> k <u>C</u> ancel
1	3,2E+07	1,3E+07	0,60	0,15	1,875	Insert
						Сору
						Delete
						New
						<u>H</u> elp
						Excel

Figure 2-16 "Defining girder groups" list box

### Defining the girder locations on the net

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

To define the girder locations on the net graphically

- Choose "Add girders" command from the "Graphically" menu in Figure 2-14. When "Add girders" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the start node of the first girder. Then drag the mouse until the end node of that girder (Figure 2-17) and click on the end node. The "Girder elements" dialog box in Figure 2-18 appears

200.2		ta - [Floor]									
≣ <u>F</u> ile ≣ ⊡	⊡⊻iew ≊ 📮	<u>G</u> raphically	In table An #∕	Options	Forma <u>t W</u> in ☰ ॐy ⊮→	dow <u>H</u> elp	<b>Q Q</b> @	100 🗨 🔇	D. 📜 🤣 I	ka 🖬 🛙 🛱	ب <mark>×</mark> پ
		× 💌 🤆			ips 🐺 🖕		•••-•		-• • • •		.*.
									_		
								+			
									]		
_								•	-		
											F

Figure 2-17 Add girder by mouse

In this dialog box click "OK" button.

Girder elements	×
Girder elements:	
Group No.	1
Start from node No.	[·] 22
End at node No.	[·] [26
<u>D</u> kCancel	<u>H</u> elp

Figure 2-18 "Girder elements" dialog box

Now, the first girder is defined as shown in Figure 2-19. Note that *ELPLA* has already typed 1 on the girder indicating the No. of girder group.

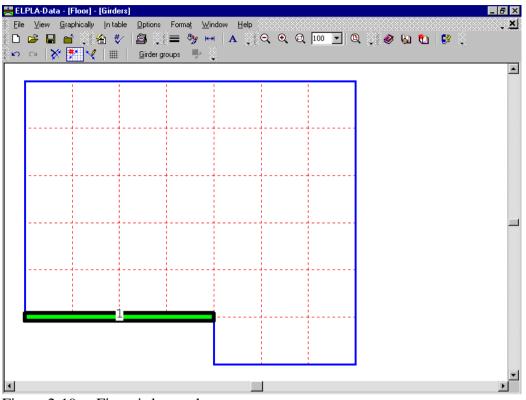


Figure 2-19 First girder on the screen

Repeat the previous steps to add the remaining girders on the net. After you have completed the definition of all girders, the screen should look like the following Figure 2-20.

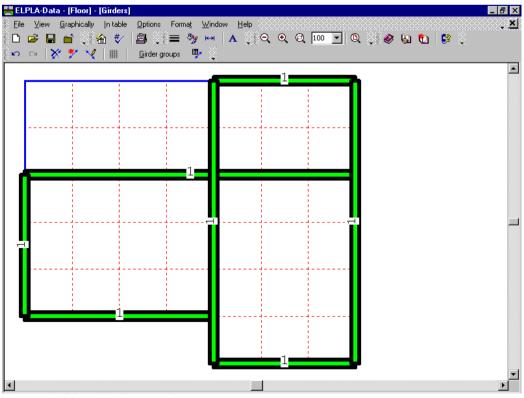


Figure 2-20 Girders on the screen

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

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

### 2.5 Supports/ Boundary conditions

In general, columns under the slab are considered as rigid supports. These supports are defined by the "Supports/ Boundary conditions" command. To define supports choose "Supports/ Boundary conditions" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 2-21 appears.

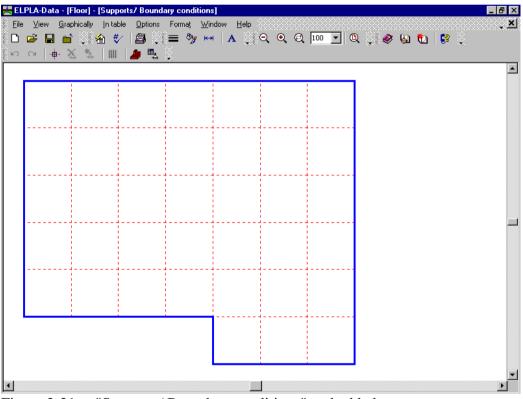


Figure 2-21 "Supports/ Boundary conditions" embedded program

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

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

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

View grouping	×
Select items to be displayed:	
Girders	<u>0</u> k
	<u>C</u> ancel
	Help
	🔽 Select <u>A</u> ll

Figure 2-22 "View grouping" check group box

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

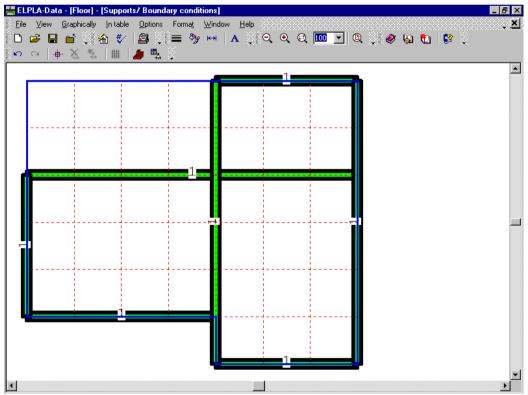


Figure 2-23 Girders in the window of the "Supports/ Boundary conditions" embedded program

### Defining supports on the net

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

To define supports on the net

- Choose "Select nodes" command from the "Graphically" menu in Figure 2-23. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nodes that have supports as shown in Figure 2-23
- After selecting nodes of supports, choose "Add boundaries" command from "Graphically" menu (Figure 2-23). The "Supports/ Boundary conditions" dialog box in Figure 2-25 appears

In this dialog box

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

*ELPLA* can calculate the punching stresses due to reactions of column supports. In this example, data corresponding to column dimensions are not required. Therefore, the user can take these data from the default column dimensions and consider all supports have column type 1.

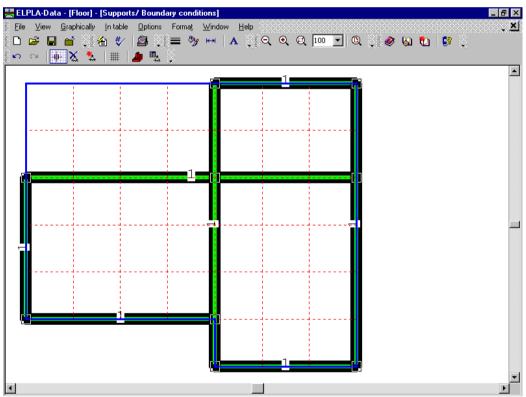


Figure 2-24 Selection of nodes that have supports

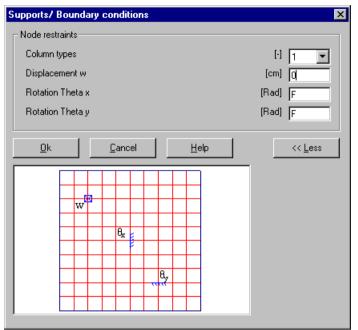
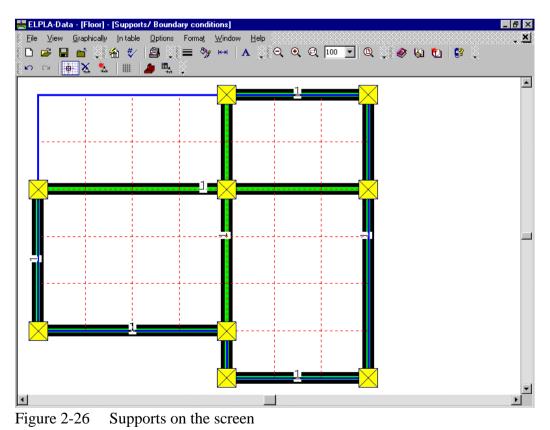


Figure 2-25 "Supports/ Boundary conditions" dialog box

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



After entering supports, do the following two steps:

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

### 2.6 Slab properties

To define the slab properties choose "Slab properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 2-27 appears with default slab properties. The data of slab properties for the current example, which are required to define, are raft material and slab thickness.

File	⊻iew <u>G</u> ra ≇ 🖬 🛍		ile Sl <u>a</u> b prop	= 🍫 😁	ns Forma <u>t</u> \ M   <b>A</b>	⊻indow <u>H</u> elp ⊃ <b>( ⊙( ©(</b>	100 💌 🔇	R 🕽 🔗 🕼 🕼 🕃	 
	≃[][		5 🖻 F			1		)	1
	1	1	1	1	1	1	1		
	1	1	1	1	1	1	1		
ſ	1	1	1	1	1	1	1		
	1	1	1	1	1	1	1		-
-	1	1	1	1	1	1	1		
L		!			1	1	1		
				Ļ				<b>J</b>	Þ

Figure 2-27 "Slab properties" embedded program

To enter the slab material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 2-27. The following list box in Figure 2-28 with default data appears. To enter or modify a value in this list box, type that value in the corresponding cell and then press "Enter" key. In the list box of Figure 2-28, enter E-Modulus of the slab, *Poisson's* ratio of the slab and slab thickness
- Click "OK" button

efining ele	ment groups (with	the same thickne	ess and slab material)	×
Group No.	E-Modulus of slab [kN/m2]	Poisson's ratio of slab [-]	Slab thickness d [m]	k Cancel
1	3,2E+07	0,20	0,10	
				<u>I</u> nsert
				<u>С</u> ору
				Delete
				New
				Help
				Excel

Figure 2-28 "Defining element groups" list box

To enter the unit weight of the slab

- Choose "Unit weight of the slab" command from "Slab properties" menu in the window of Figure 2-27. The following dialog box in Figure 2-29 with a default unit weight of 25 [kN/m<sup>3</sup>] appears
- Click "OK" button

Unit weight of the slab	×
Unit weight of the slab:	
Unit weight of the slab	Gb [kN/m3] 25
	,
<u>Ok</u> w	<u>C</u> ancel <u>H</u> elp

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

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

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

### 2.7 Reinforcement data

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

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

ELPLA-Tutorial

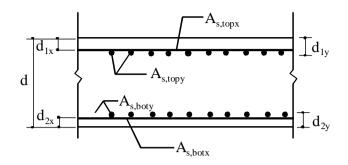


Figure 2-30 Section geometry and reinforcement parallel to *x*-direction

It is important to say that the design code parameters such as partial safety factors for concrete strength, steel strength and internal forces are defined by choosing the "Design code parameters" command from "Main data" menu in *ELPLA-Data*, while reinforcement data such as design code, concrete grade, steel grade and concrete covers are defined by choosing the "Reinforcement" command from "Data" menu in *ELPLA-Data*. Design code parameters are standard data for all projects while reinforcement data may be varied from one project to another.

To define the reinforcement data choose "Reinforcement" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 2-31 appears with default reinforcement data.

Reinforcement (Design for flexural moment)									
Design code:	Concrete grade:								
EC 2	haracteristic compressive cylinder stre	ength fck [kN/m2] 30	000,0						
	C Another C C 12/15 (	C 16/20 C 20/25 C	C 25/30						
	⊙ C 30/37 ○ C 35/45 C	CC 40/50 CC 45/55 C	C 50/60						
Steel grade:									
haracteristic tensile yield :	trength	fyk [kN/m2] 50	0000 ÷						
C Another C BS	220 O BSt 420 O BSt	500 O BSt 550 O B							
Concrete cover + 1/2	ar diameter:								
X-direction top	d1x [cm] 1,5	d1y							
X-direction bottom	d2x [cm] 1,5		• •						
Y-direction top	d1y [cm] 2,0								
Y-direction bottom	d2y [cm] 2,0		d2x						
Save	<u>C</u> ancel <u>H</u> elp	Load	Save <u>A</u> s						

Figure 2-31 "Reinforcement" dialog box

In this dialog box

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

#### 2.8 Loads

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

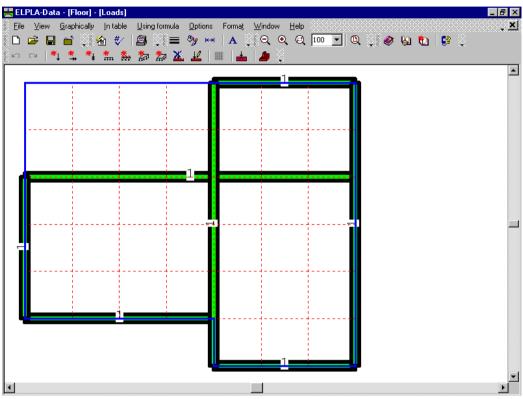


Figure 2-32 "Loads" embedded program

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

To enter the first distributed loads

- Choose "Distributed loads" command from "Graphically" menu in the window of Figure 2-32. When "Distributed loads" command is chosen, the cursor is changed from an arrow to a cross hair. Then the load can be defined by holding the left mouse button down at the starting point of the distributed load. As the mouse is dragged, a box appears, indicating a distributed load is being defined. When the left mouse button is released, the following dialog box in Figure 2-33 appears with the load value and coordinates

In this dialog box

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

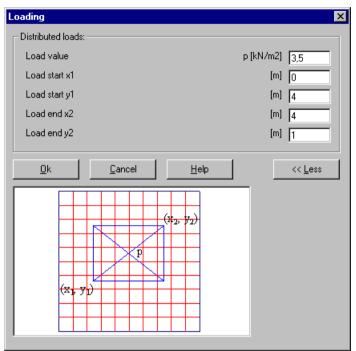


Figure 2-33 "Loading" dialog box

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

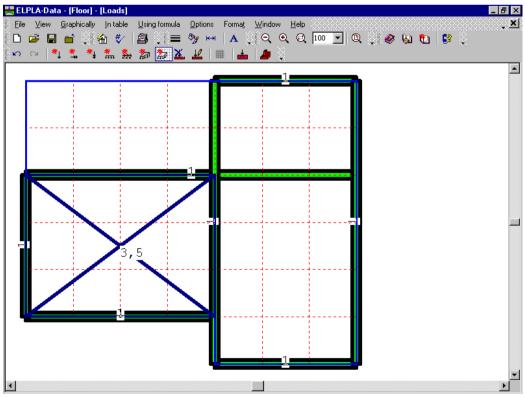


Figure 2-34 First distributed load on the screen

Repeat the previous steps to enter the remaining distributed loads on the net. After you have completed the definition of all loads on the net, the screen should look like the following Figure 2-35.

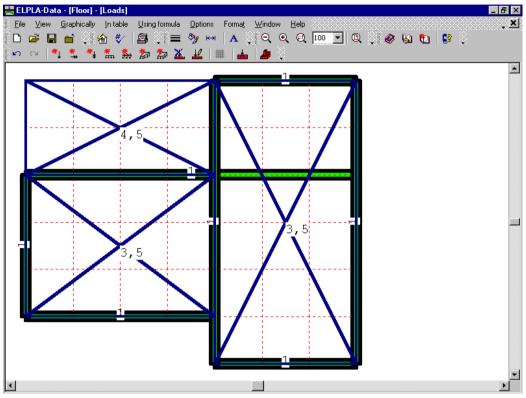


Figure 2-35 Loads on the FE-Net

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

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

Creating the project of the slab floor is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

### **3** Carrying out the calculations

### 3.1 Starting *ELPLA-Solver*

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

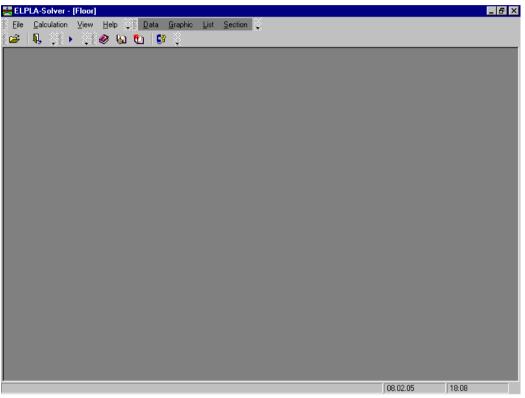


Figure 2-36 Opening screen of the sub program ELPLA-Solver

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. Also, *ELPLA* will active the "Calculation" menu. This menu contains commands of all calculations. Commands of the calculation depend on the analysis type. For the current example, the items, which are required to be calculated, are:

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

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

#### **3.2** Carrying out all computations

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

#### Analysis progress

Analysis progress menu in Figure 2-37 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

Solving system of linear equations (ban					
Please wait!					
File 'H:\ELPLA_Daten\Floor.PF1' is being read!					
(Slab stiffness matrix)					
[]					

Figure 2-37 Analysis progress menu

#### Check of the solution

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

Check of the solution	
V - Load:	
Total load	[kN] = 296,8
	• •
Sum of reactions	[kN] = 296,8
X - Moment:	
Sum Mx from loads	[kN.m] = -6.9
Sum Mx from reactions	[kN.m] = -6,9
Y - Moment:	
Sum My from loads	[kN.m] = 14,2
Sum My from reactions	[kN.m] = 14,2
OL	Help
	<u>H</u> eip

Figure 2-38 Menu "Check of the solution"

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

ELPLA-Tutorial

### 4 Viewing data and result graphics

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

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

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

🗮 ELF	PLA-Gra	phic - [l	loor]												_ 8 ×
<u> </u>	⊻iew	<u>G</u> raphic	<u>O</u> ption	s Forma <u>t</u>	<u>₩</u> indow	<u>H</u> elp	.∄ <u>D</u> a	ata <u>L</u> ist	<u>S</u> ection	Solver	L.				
1			<b>I</b> , 📜	'≜ ♥		1.	= #		<b>a</b> 1	$Q \oplus$	Q 10		0 0	. 🤣	⊌a ta   " 
ΞĪ.	- 💋 -	## -	H • (	• • 🗮	i 🗙 🎞	-	. 🛃	+ *	- 👹 -	🗜 🔹		≛ ∙	• • •	¥ -	<u>, ≡ "</u>
<b>T</b> .			~	•			.1	1				08.02	2.05	18:15	

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

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

In this option box

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

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

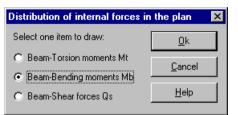


Figure 2-41 "Distribution of internal forces in the plan" option box

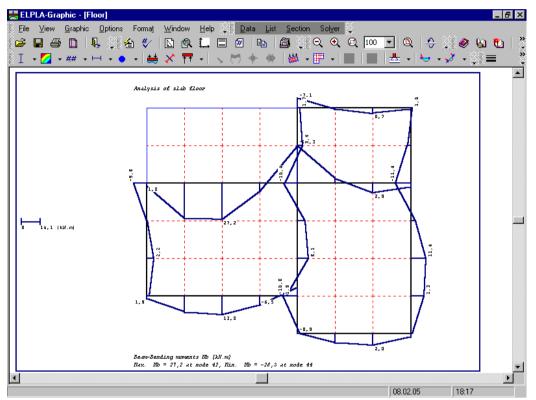


Figure 2-41 Beam-Bending moments *Mb* 

# 5 Index

# A

Analysis type	5
---------------	---

# B

Boundary conditions	 18

# С

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

# D

Deformation	
Design codes	21, 22

### E

Element groups	21

### F

# G

Generation type	8
Girder locations	
Girders	12, 15

### Ι

Internal forces	22,	26

## 

### M

faterial properties13
-----------------------

### N

New project
-------------

### P

Project identification7
-------------------------

### R

Reinforcement	21
Remove nodes10,	11

# S

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

### U

Unit weight of the slab......21

Example 3

Analysis of system of two circular rafts

### ELPLA-Tutorial

# Contents

# Page

1	Description of the problem1.1Loads and dimension1.2Raft material1.3Soil properties1.4Method of analysis	ons	3 3 3 4 4
2	Creating the project of raf	+ 1	5
2	2.1 Calculation metho		5
	2.1 Calculation metho 2.2 Project identificati		8
	2.2 FIOJect Identificati 2.3 FE-Net data	011	9
	2.4 Soil properties		11
	2.4 Son properties 2.5 Foundation proper	ties	16
	2.6 Loads		10
3	Creating the project of raft 2		19
	3.1 Modifying the proj		20
	3.2 Modifying origin c		20
4	Creating the project of the	system of rafts 1 and 2	22
	4.1 Filenames of slab	foundations	22
	4.2 Project identificati	on	24
5	Carrying out the calculation	ons	25
	5.1 Starting <i>ELPLA-Solver</i>		25
	5.2 Carrying out all co	mputations	26
6	Viewing data and results		29
	6.1 Viewing result gra	phics	29
	6.2 Plot a diagram of t		30
7	Index		34

### **1** Description of the problem

An example of a system of two equal large circular rafts is selected to illustrate some features of *ELPLA* for analyzing system of foundations.

### 1.1 Loads and dimensions

Each raft has a diameter of 22 [m] and thickness of 0.65 [m]. Loading on each raft consists of 24 column loads, in which 16 column loads have  $P_1 = 1250$  [kN] and 8 column loads have  $P_2 = 1000$  [kN] as shown in Figure 3-1 and Table 3-1. The origin coordinates for raft 1 in the global system are (0.0, 0.0) while for raft 2 they are (0.0, 22.5).

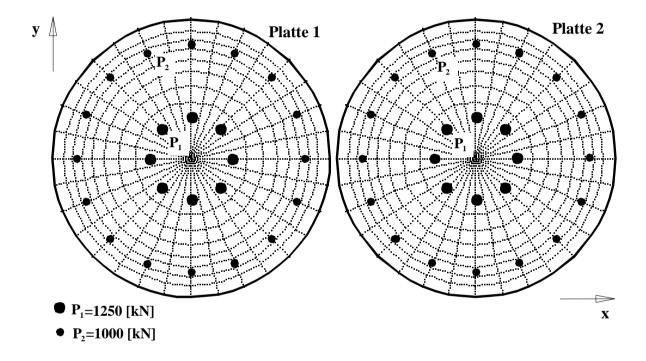


Figure 3-1 System of two equal circular rafts

### 1.2 Raft material

Material of the two rafts is supposed to have the following parameters:

Young's modulus	$E_b$	$= 2.6 * 10^7$	$[kN/m^2]$
Poisson's ratio	$\mathbf{v}_b$	= 0.15	[-]
Unit weight of raft material	$\gamma_b$	= 0.0	$[kN/m^3]$

Unit weight of the raft material is chosen to be  $\gamma_b = 0.0$  to neglect the own weight of the raft in the analysis.

#### 1.3 Soil properties

The rafts rest on a silt layer of 15 [m] thickness. The Modulus of compressibility of the silt is  $E_s = 9500 \text{ [kN/m^2]}$ . *Poisson's* ratio of the soil is taken to be  $v_s = 0.0$  [-]. The level of foundation under the ground surface is assumed to be  $d_f = 0.0$  [m]. The effect of the reloading pressure on the soil and the uplift pressure on the rafts are neglected.

### 1.4 Method of analysis

It is required to analyze the system of rafts together according to the following soil model and numerical calculation method:

- Layered soil medium Continuum Model
- Modulus of compressibility method for an elastic raft on layered soil medium (Solving system of linear equations by iteration-method 6)

This Tutorial Manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for the soil models and numerical calculation methods are well documented in the User's Guide of *ELPLA*.

Load No. I	Load value P	x-position x	y-position y
[-]	[kN]	[m]	[m]
1	1250	7.75	11
2	1250	14.25	11
3	1250	11	7.75
4	1250	11	14.25
5 6	1250	8.7	8.7
	1250	13.3	8.7
7	1250	8.7	13.3
8	1250	13.3	13.3
9	1000	2	11
10	1000	20	11
11	1000	11	2
12	1000	11	20
13	1000	4.64	4.64
14	1000	17.36	4.64
15	1000	4.64	17.36
16	1000	17.36	17.36
17	1000	2.69	7.56
18	1000	7.56	2.69
19 20	1000 1000	14.44	2.69
20 21		19.31	7.56
21	1000 1000	2.69	14.44 19.31
23	1000	7.56 14.44	19.31
24	1000	19.31	14.44

Table 3-1Point loads P

### 2 Creating the project of raft 1

In this section the user will learn how to create a project for analyzing system of two rafts. Thus is done by first entering the data of the two rafts individually in the same manner of the previous foundation example and then creating a project for the system of two rafts.

### 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation method" wizard appears, Figure 3-2. This wizard will guide you through the steps required to create the project. As shown in this Figure, the first form of the wizard is the "Analysis type" form. In this form define the analysis type of the problem, where *ELPLA* can deal with different structural systems. As the analysis type is a foundation problem, select "Analysis of slab foundation" then click "Next" button to go to the next form.

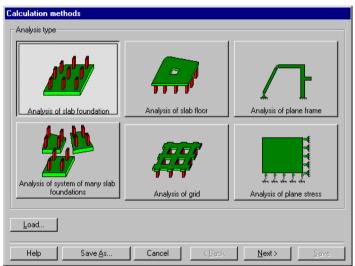


Figure 3-2 "Calculation method" wizard with "Analysis type" form

After clicking "Next" button the "Calculation methods" form appears, Figure 3-3.

To define the calculation method

- Select the calculation method "6-Modulus of Compressibility (Iteration)"
- Select subsoil model "Layered soil model"
- Click "Next" button to go to the next form

Calculation methods
Calculation methods:
O 1- Linear Contact Pressure
O 2/3- Constant/ Variable Modulus of Subgrade Reaction
O 4- Modification of Modulus of Subgrade Reaction by Iteration
○ 5- Isotropic Elastic Half Space
6- Modulus of Compressibility (Iteration)
O 7- Modulus of Compressibility (Elimination)
🔿 8- Rigid slab
O 9- Flexible foundation
Subsoil model:
O Half Space model
Layered soil model
Help         Save As         Cancel         < Back         Next >         Save

Figure 3-3 "Calculation methods" form

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

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

Calculation methods	
System symmetry:	
Unsymmetrical system	
Symmetrical system about x-axis	Double-symmetrical system
Symmetrical system about y-axis	Anti-symmetrical system in x-axis
Help Save <u>A</u> s Car	ncel < Back Next> Save

Figure 3-4 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 3-5. In this list *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other. Since no option will be considered in the analysis, click the "Save" button.

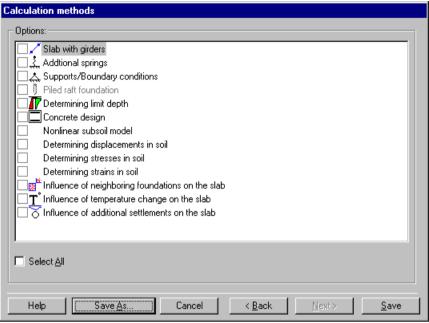


Figure 3-5 "Options" list

After clicking "Save" button the "Save as" dialog box appears, Figure 3-6.

In this dialog box

- Type a file name for the current project in the "File name" edit box. For example type "Raft 1". ELPLA will use automatically this file name in all reading and writing processes Click "Save" button

Save As				? ×
Save jn: 🔁	ELPLA PE 8.0	•	E 🗈 I	- 🖽
Example				
Raft 1				
File <u>n</u> ame:	Raft 1			Save
r lie <u>H</u> ame.				<u> </u>
Save as <u>t</u> ype:	Isolated slab foundation-files (*.P01	)	•	Cancel

Figure 3-6 "Save as" dialog box

ELPLA will activate the "Data" menu. Also the file name of the current project [Raft1] will be displayed instead of the word [Untitled] in the ELPLA-Data title bar, Figure 3-7.

In the "Data" menu the user can enter the remaining data of the project using the same sequence of commands in this menu. The first command in the menu is "Calculation methods", which has been already entered. Therefore *ELPLA* has put the sign " $\sqrt{}$ " beside this command (Figure 3-7). ELPLA puts this sign beside the commands those the user has entered so that the user can know which data were defined.

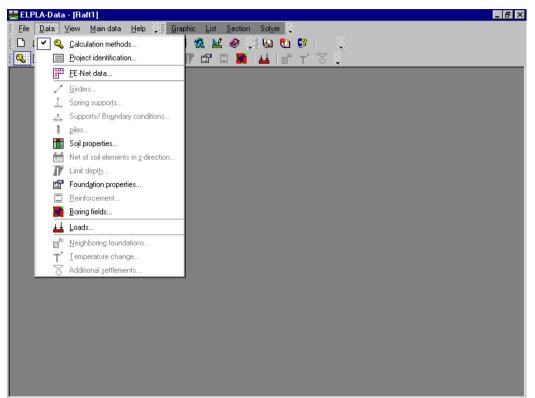


Figure 3-7 ELPLA-Data after defining the calculation method

### 2.2 **Project identification**

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data* window. The dialog box in Figure 3-8 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of system of two circular rafts"
- Type the date of the project in the "Date" edit box
- Type "Raft 1" in the "Project" edit box
- Click "Save" button

A	uftragsda	iten	×
	-Auftragsd	laten:	
	Auftrag	Analysis of system of two circular rafts	
	Datum	Mondy, 14. June 2003	
	Projekt	Raft 1	
	<u>S</u> peiche	ern <u>A</u> bbrechen <u>H</u> ilfe <u>L</u> aden Speichern <u>u</u> nter	

Figure 3-8 "Project identification" dialog box

#### 2.3 FE-Net data

Choose "FE-Net data" command from "Data" menu of *ELPLA-Data* window. The "FE-Net generation" wizard appears (Figure 3-9). This wizard will guide you through the steps required to generate the FE-Net. As shown in this Figure the first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These are used to generate standard nets that have irregular shapes. For the given problem the raft has a circular shape.

To generate the FE-Net

- In the "Slab type" options choose the circular slab option
- Type 22 in the "Diameter of circular slab" edit box
- Click "Next" button to go to the next form

FE-Net generation			
Slab type			
			0
Circular slab:			
Diameter of circular slab		D [m]	22
1			
Help	Cancel	< <u>B</u> ack. <u>N</u> ext >	Einish

Figure 3-9 "Slab type" form

The following "Generation type" form appears, Figure 3-10. *ELPLA* can deal with various types of generations with triangle and/ or rectangular elements. Choose the combination of triangle and rectangular elements option as the generation type. Click "Next" button to go to the next form.

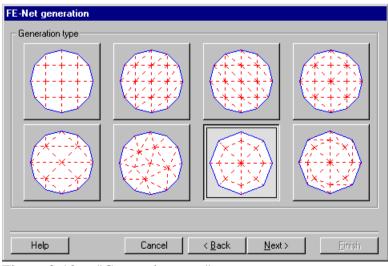


Figure 3-10 "Generation type" page

The next form of the "FE-Net generation" wizard is the "Generation parameters" form. In this form the default values of generation parameters appear, Figure 3-11. For the given problem mesh optimization options are not required.

In this form

- In the *x*-direction frame type 8 in the "No. of circular divisions" edit box
- Uncheck the "Smoothing mesh" option
- Uncheck the "Directing border elements" option

FE-Net generation					
Generation parameters					
No. of circular divisions				[-]	8
Mash antinination					
Mesh optimization					
Smoothing mesh					
Directing border elements					
Help	Cancel	< <u>B</u> ack	<u>N</u> ext >		<u>F</u> inish

Figure 3-11 "Generation parameters" form

Click "Finish" button in the "Generation parameters" form. *ELPLA* will generate a suitable FE-Net for the circular raft of 22 [m] diameter with a combination of triangle and rectangular elements, which have equal areas. The following embedded program in Figure 3-12 appears with the generated net.

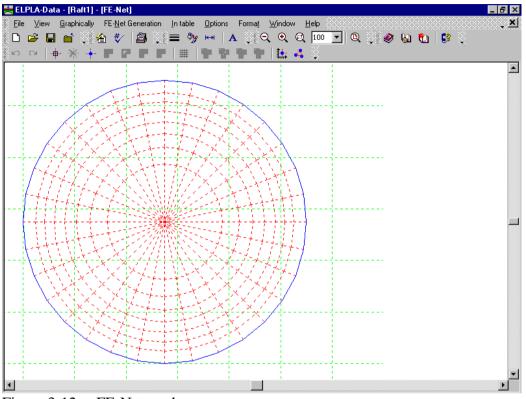


Figure 3-12 FE-Net on the screen

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

- Choose "Save FE-Net" command from "File" menu in Figure 3-12 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 3-12 to close "FE-Net" embedded program and to return to the main window of *ELPLA-Data*

### 2.4 Soil properties

To define the soil properties, choose "Soil properties" command from "Data" menu of *ELPLA-Data* window. The following sub program in Figure 3-13 appears with a default-boring log.

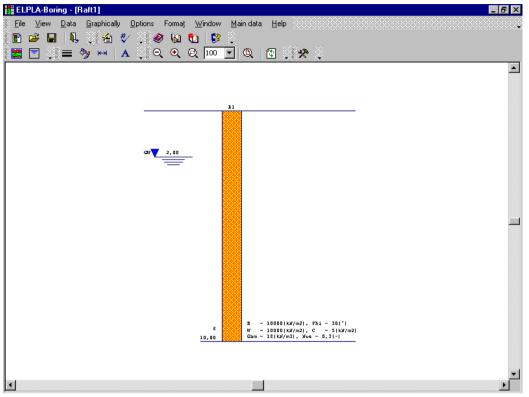


Figure 3-13 *ELPLA-Boring* sub program with a default-boring log

### Modifying data of boring log graphically

In *ELPLA* it is also possible to define or modify the boring log graphically which makes the definition of the boring log very easy. By double-clicking the left mouse button on a specified screen position, the user can define or modify the soil data and input parameters graphically.

To enter the geotechnical data of the soil layer

- Double-click on the geotechnical data of the soil layer. The corresponding dialog box to modify the geotechnical data of the soil layer appears (Figure 3-14)
- In the dialog group box "Geotechnical data of the layer" in Figure 3-14 define the geotechnical data of the soil layer as follows:

 $E_s = 9500$  [kN/m<sup>2</sup>]  $W_s = 9500$  [kN/m<sup>2</sup>]

The values *Es* and *Ws* are the same, because the effect of reloading on the soil is not required. The unit weight of the soil is used to determine the overburden pressure  $q_v$  [kN/m<sup>2</sup>] due to the removed soil, which is equal to  $\gamma_s * d_f$ . In the current example  $d_f = 0.0$ , which means the unit weight of the soil is not required. However, the unit weight of the soil under the foundation depth is entered by the default value. Also the angle of internal friction  $\varphi$  and the cohesion *c* of the soil are not required because the selected type of the analysis is linear analysis. Therefore the user can let the default values of the internal friction and the cohesion. Next, click "OK" button.

Soil data Boring log No. 1 fr Layer No. 1 from Geotechnical d				×
Es [kN/m2] Ws [kN/m2] Gam [kN/m3]	9500 9500 18	Fhi (*) c [kN/m2]	30 5	
	<u>C</u> ancel			

Figure 3-14 "Geotechnical data of the soil layer" dialog box

To define the soil type and color for the layer

- Double-click on the soil symbol of the soil layer. The corresponding dialog box to modify the soil symbols of that layer appears (Figure 3-15)
- Select "U, Silt" as the soil type in the "Main soil type 1" combo box in the dialog group box "Soil and rock symbols" in Figure 3-15. The color of the silt and a short text "U" according to the German specification code DIN 4023 will be automatically created
- Click "OK" button

Soil data		×
Boring log No. 1 from Layer No. 1 from 1 Soil and rock sym	layers:	
Main soil type 1	U, Silt	
Main soil type 2	-, No symbole	
Submain soil 1	-, No symbole	
Submain soil 2	-, No symbole 💌	
Color	-, No color	
Short text	U	
<u> </u>	<u>C</u> ancel	

Figure 3-15 "Soil and rock symbols" dialog box

To modify the groundwater depth under the ground surface

- Double-click on the groundwater level. The corresponding edit box to modify the groundwater depth under the ground surface appears, Figure 3-16. To neglect the uplift pressure on the raft, groundwater level is chosen at anywhere under the raft basement
- Type 15 in the "Groundwater depth under the ground surface" edit box
- Click "OK" button

Groundwater		×
Groundwater: —		
Groundwater de	pth under the ground surface (m) 15	
	part and de la groan de canade (m) [13	
<u>0</u> k	<u>C</u> ancel	

Figure 3-16 "Groundwater depth under the ground surface" edit box

To modify a layer depth

- Double-click on the layer depth. The corresponding dialog edit box to modify the layer depth under the ground surface appears, Figure 3-17
- Type 15 in the "Layer depth under the ground surface" edit box
- Click "OK" button

Soil data		×
Layer No. 1 from	rom 1 boring logs: n 1 layers: ler the ground surface [m]	5
<u>O</u> k	<u>C</u> ancel	

Figure 3-17 "Layer depth under the ground surface" edit box

To modify the label of a boring log

- Double-click on the label of the boring log. The corresponding edit box to modify the label of the boring log appears, Figure 3-18
- Type B1 in the edit box of Figure 3-18
- Press "Enter" key to consider the text

#### B2

Figure 3-18 "Label of the boring log" edit box

To enter the main soil data for the layer

- Choose "Main soil data" from "Data" menu in Figure 3-13. The following dialog group box in Figure 3-19 appears
- In this dialog group box enter the settlement reduction factor  $\alpha$  [-], *Poisson's* ratio of the soil  $v_s$  [-] and the groundwater depth under the ground surface  $G_w$  [m] as indicated in Figure 3-19. Note that the groundwater depth under the ground surface was typed in the corresponding edit box because this value has been already defined graphically
- Click "OK" button in the "Main soil data" dialog group box in Figure 3-19

Main soil data			×
Soil properties Calculation parameters of flexibility coefficients	Bearing capacity factor:	5	
Main soil data:			
Settlement reduction factor Alfa <= 1	Alfa	[·]	1
Groundwater depth under the ground surface	Gw	[m]	15,00
<u> </u>	Help		

"Main soil data" dialog group box Figure 3-19

After the user has completed the definition of all soil properties and parameters, the screen should look like the following Figure 3-20.

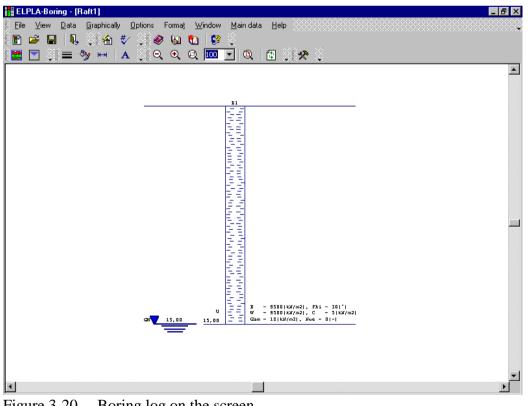


Figure 3-20 Boring log on the screen

After entering all data and parameters of boring log do the following two steps:

- Choose "Save boring logs" command from "File" menu in Figure 3-20 to save the data of the boring log
- Choose "Close boring logs" command from "File" menu in Figure 3-20 to close *ELPLA*-*Boring* and to return to the main window of *ELPLA*-*Data*

### 2.5 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu of *ELPLA-Data* window. The following embedded program in Figure 3-21 appears with default foundation properties. The data of foundation properties for the current example, which are required to define, are raft material and raft thickness. Any other data corresponding to foundation properties in the program menus are not required. Therefore the user can take these data from the default foundation properties.

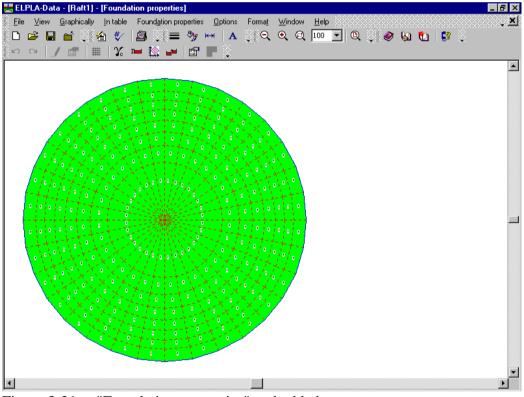


Figure 3-21 "Foundation properties" embedded program

To enter the raft material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 3-21. The following list box in Figure 3-22 with default data appears. To enter or modify a value in this list box, type that value in the corresponding cell, then press "Enter" key. In the list box of Figure 3-22 enter E-Modulus of the raft, *Poisson's* ratio of the raft and raft thickness
- Click "OK" button

Defining ele	ement groups (with	the same thickne	ess and slab material	×
Group No.	E-Modulus of slab [kN/m2]	Poisson's ratio of slab [-]	Slab thickness d [m]	<u>O</u> k Cancel
1	2,6E+07	0,15	0,65	
				Insert
				<u>С</u> ору
				<u>D</u> elete
				New
				<u>H</u> elp
				Excel

Figure 3-22 "Defining element groups" list box

To enter the unit weight of the raft

- Choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 3-21. The following dialog box in Figure 3-23 with a default unit weight of 25 [kN/m<sup>3</sup>] appears. To neglect the self-weight of the raft in the analysis, type 0 in the edit box "Unit weight of the foundation"
- Click "OK" button

Unit weight of the foundation	×	<
Unit weight of the foundation: Unit weight of the foundation	Gb [kN/m3] 0	
<u>D</u> k	<u>Cancel</u> <u>H</u> elp	

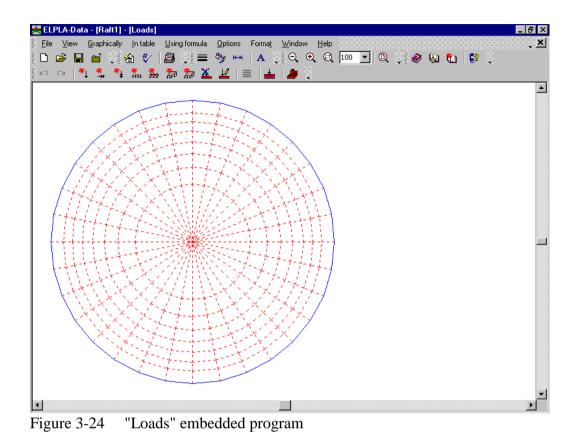
Figure 3-23 "Unit weight of the foundation" dialog box

After entering the foundation properties do the following two steps:

- Choose "Save foundation properties" command from "File" menu in Figure 3-21 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 3-21 to close the "Foundation properties" embedded program and to return to the main window of *ELPLA-Data*

### 2.6 Loads

To define the loads, choose "Loads" command from "Data" menu of *ELPLA-Data* window. The following embedded program in Figure 3-24 appears.



To enter loads

- Choose "Point loads" command from "In Table" menu in the window of Figure 3-24. The following list box in Figure 3-25 appears
- Enter the external point loads P [kN] and their corresponding coordinates (x, y) in the list box of Figure 3-25. This is done by typing the value in the corresponding cell and then press "Enter" key. The coordinates of the point load are related to the lower-left corner of the raft (local coordinates)
- Click "OK" button

No.   [•]	Column types I [·]	Load P [kN]	x-position x [m]	y-position y [m]		<u>O</u> k Cancel
1	1	1250,0	7,75	11,00		<u></u>
2	1	1250,0	14,25	11,00		Insert
3	1	1250,0	11,00	7,75		
4	1	1250,0	11,00	14,25		<u>С</u> ору
5	1	1250,0	8,70	8,70		2-65
6	1	1250,0	13,30	8,70		Delete
- 7	1	1250,0	8,70	13,30		
8	1	1250,0	13,30	13,30		New
9	1	1000,0	2,00	11,00		
10	1	1000,0	20,00	11,00		<u>H</u> elp
11	1	1000,0	11,00	2,00		
12	1	1000,0	11,00	20,00	F	Excel

Figure 3-25 "Point loads *P*" list box

ELPLA-Data - [Raif1] - [Loads] Fle View Graphically Intable Leing formula Options Format Window Help Come of the second 
After you have completed the definition of all load data, the screen should look like the following Figure 3-26.

Figure 3-26 Loads on the screen

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

- Choose "Save loads" command from "File menu in Figure 3-26 to save the load data
- Choose "Close loads" command from "File" menu in Figure 3-26 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

The project creation of raft 1 is now complete.

### **3** Creating the project of raft 2

The data of the two rafts are quite similar except the origin coordinates of the global system, which are chosen to be (0, 0) and (0, 22.5) for rafts 1 and 2 respectively. Project identification is entered here so that the user can distinguish between the two projects. The data of raft 2 are created by first saving the data of raft 1 under a new file name and then modifying the project identification and origin coordinates.

To save the data under a new file name

Choose "Save project as" command from "File" menu of *ELPLA-Data* window. The following "Save as" dialog box appears, Figure 3-27

In this dialog box

- Type a file name for the project of raft 2 in the file name edit box, for example "Raft 2"
- Click "Save" button

Save As		? X
Savejn: 🔁	ELPLA PE 8.0 💌 🖛 🗈 💣 💷 -	
Example		
Raft 1 Raft 2		
Γ		
File name:	D-0.2	-1
-	Raft 2 Save	_
Save as <u>t</u> ype:	Isolated slab foundation-files (*.PD1)	

Figure 3-27 "Save as" dialog box

### 3.1 Modifying the project identification

To modify the project identification of raft 2 choose "Project identification" command from "Data" menu of *ELPLA-Data* window. The dialog box in Figure 3-28 appears.

In this dialog box

- Type "Raft 2" in the "Project" edit box
- Click "Save" button

A	Auftragsdaten X			
	- Auftragso	laten:		
	Auftrag	Analysis of system of two circular rafts		
	Datum	Mondy, 14. June 2003		
	Projekt	Raft 2		
	<u>S</u> peich	ern <u>A</u> bbrechen <u>H</u> ilfe <u>L</u> aden Speichern <u>u</u> nter		

Figure 3-28 "Project identification" dialog box

### **3.2** Modifying origin coordinates

To modify the origin coordinates of raft 2, choose "Foundation properties" command from "Data" menu of *ELPLA-Data* window. The following embedded program in Figure 3-29 appears.

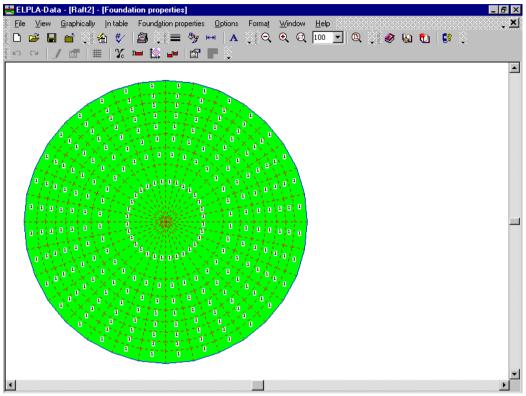


Figure 3-29 "Foundation properties" embedded program

In this "Foundation properties" embedded program

- Choose "Origin coordinates" command from "Foundation properties" menu in the window of Figure 3-29. The following dialog box in Figure 3-30 appears
- Type 22.5 in the "*x*-coordinate" edit box
- Click "OK" button

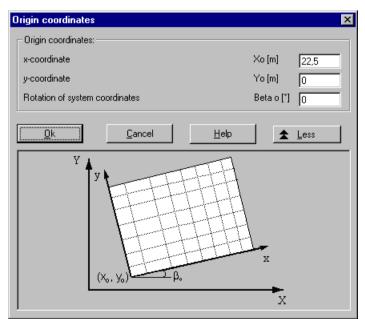


Figure 3-30 "Origin coordinates" dialog box

After entering the foundation properties do the following two steps:

- Choose "Save foundation properties" command from "File" menu in Figure 3-29 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 3-29 to close the "Foundation properties" embedded program and return to *ELPLA-Data*

Creating the projects of the two rafts 1 and 2 is now complete.

## 4 Creating the project of the system of rafts 1 and 2

Data of system of many slabs are defined through the "Data" menu in *ELPLA-Data*, which in this case contains the following two commands:

- "Filenames of slabs foundations" command
- "Project identification" command

### 4.1 Filenames of slab foundations

Choose "New project" command from the "File" menu of *ELPLA-Data* window. The following "Calculation method" wizard in Figure 3-31 appears.

Calculation methods		
Analysis type		
	<b>~</b>	
Analysis of slab foundation	Analysis of slab floor	Analysis of plane frame
Analysis of system of many slab	Analysis of grid	Analysis of plane stress
Load		
Help Save <u>A</u> s	Cancel < Back	<u>N</u> ext > <u>S</u> ave

Figure 3-31 "Calculation method" wizard

In this wizard

- Select "Analysis of system of many slab foundations", as the analysis type is a system of two rafts
- Click "Next" button

No.	Filename of Project	Slab type	

After clicking "Next" button "Filenames of slab foundations" list box in Figure 3-32 appears.

Figure 3-32 "Filenames of slab foundations" list box

To enter the file names of the rafts, which are required to be analyzed as a system of foundations, click "Add project". After clicking "Add project" the "Open" dialog box in Figure 3-33 appears.

In this dialog box

- Type the file name of raft 1 in the file name edit box
- Click "Open"

Open			? ×
Look jn: 🔁	ELPLA PE 8.0	- + 🗈	-111 *
Example			
Raft 1 Raft 2			
File <u>n</u> ame:			<u>O</u> pen
Files of <u>type</u> :	Isolated slab foundation-files (*.P01)	<b>_</b>	Cancel
r iles of type.	Isolated stab roundation-files (".PUT)		

Figure 3-33 "Open" dialog box

Repeat the previous steps and enter the file name of raft 2. After the user has completed the definition of file names of the projects, the list box should look like the following Figure 3-34.

		n methods b foundations:		
	No.	Filename of Project	Slab type	
	1	Raft1	elastic	
	2	Raft2	elastic	
ľ				
	A <u>d</u> d	project <u>R</u> emove project		<u>N</u> ew
	Help	Save <u>A</u> s Cancel	< <u>B</u> ack	Next > Save

Figure 3-34 "Filenames of slab foundations" list box after entering file names of the projects

Click "Save" button in the "List of filenames of slab foundations" list box in Figure 3-34. After clicking "Save" button the "Save as" dialog box in Figure 3-35 appears.

In this dialog box

- Type a file name for the project of a system of rafts in the file name edit box. For example type "Raft 1 + 2"
- Click "Save" button

Save As ? 🗙
Save jn: 🔄 ELPLA PE 8.0 💌 🖛 🛍 📸
gg <sup>fft</sup> Raft 1 + 2
File name: Raft 1 + 2
Save as type: System of many slab foundations-files (*.PO2 🔽 Cancel

Figure 3-35 "Save as" dialog box

## 4.2 **Project identification**

The project of a system of rafts is considered as an independent project. Therefore, like the single rafts, a new identification of the project of the system of rafts must be entered.

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data* window. The dialog box in Figure 3-36 appears.

In this dialog box

- Click "Load" button and open the project identification data of raft 1
- Modify "Raft 1" to "Raft 1 + 2" in the "Project" edit box
- Click "Save" button

P	roject ide	ntification				
	- Project id	entification:				
	Title	e Interaction of two circular slab foundations				
	Date	Mondy, 14. June 2003				
	Project	ect Raft 1 + 2				
	<u>S</u> ave	<u>C</u> ancel <u>H</u> elp <u>L</u> oad Save <u>A</u> s				

Figure 3-36 "Project identification" dialog box

Creating the project is now complete. The next step is analyzing the problem.

## 5 Carrying out the calculations

## 5.1 Starting *ELPLA-Solver*

To analyze the problem, leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data* window. Then *ELPLA-Solver* window appears, Figure 3-37.

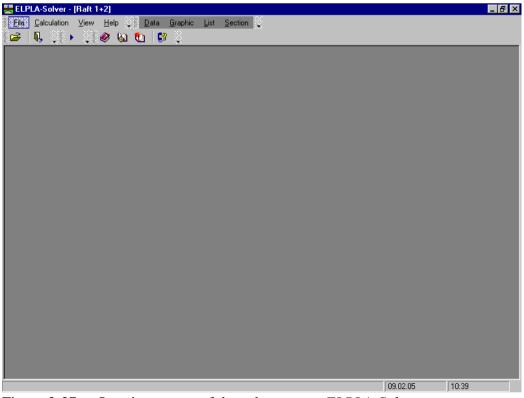


Figure 3-37 Opening screen of the sub program *ELPLA-Solver* 

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. The "Calculation" menu contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For the current example the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Determining flexibility coefficients of the soil for system of slabs
- Assembling the soil stiffness matrix
- Iteration process
- Determining deformation, internal forces, contact pressures

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

### 5.2 Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Calculation" menu in *ELPLA-Solver* Window. The following "Iteration parameters" option box in Figure 3-38 appears
- In "Iteration parameters" option box select the option of the iteration condition
- Click "OK" button

Iteration parameters				
Which option is endir	ng the iteration proces	:s?		
Accuracy [m]		0,0001		
C Iteration No.		10		
<u>k</u>	<u>C</u> ancel	<u>H</u> elp		

Figure 3-38 "Iteration parameters" option box

The progress of all computations according to the defined method will be carried out automatically with displaying information through menus and messages.

### **Analysis progress**

Analysis progress menu in Figure 3-39 appears, in which various phases of calculation are progressively reported as the program analyzes the problem. Also a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

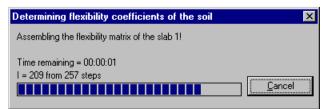


Figure 3-39 Analysis progress menu

#### **Iteration process**

Information about the convergence progress of the computations is displayed in the "Iteration process" list box in Figure 3-40 during the iteration process.

Iteration process				
Iteration No.	Accuracy [m]		<u>S</u> top	
1	0,01460557000			
2	0,00250047400		<u>P</u> ause	
3	0,00051294640			
			<u>H</u> elp	
Iteration cycles is ended at accuracy [m]<= 0,0001				
Computation time = 00:00:12 performing iteration!				

Figure 3-40 "Iteration process" list box

### Check of convergence

An accuracy number controls the convergence progress of the solution. The solution is considered convergent if the accuracy number of the step i + 1 is less than that of the previous step i. The maximum difference between the soil settlement and the slab deflection in [m] is considered as an accuracy number. A message box appears during the iteration process when the convergence becomes no longer convergent for example (Figure 3-41),

where the accuracy number of the iteration No. 5 = 0.00021766310 [m]

while the accuracy number of the iteration No. 4 = 0.00020922720 [m]

Check of convergence				
No convergence is reached at the last step!				
<u>D</u> k <u>H</u> elp				

Figure 3-41 "Check of convergence" message box

In this case the user can either stop the iteration process and save the results of iteration No. 5 or continue the iteration process until reaching the convergence.

Do the next steps to continue the iteration process:

- Click "OK" button in the "Check of convergence" message box in Figure 3-41. The "Iteration process" list box in Figure 3-42 appears. Note that the "Pause" button changes to "Continue" button
- Click "Continue" button

Iteration process					
Iteration No.	Accuracy [m]	<u>S</u> top			
1	0,01460557000				
2	0,00250047400	Continue			
3	0,00051294640	<u></u>			
		<u>H</u> elp			
Iteration cycles is ended at accuracy [m]<= 0,0001 Computation time = 00:00:12					

Figure 3-42 "Iteration process" list box

## Check of the solution

Once the analysis is complete, a check menu of the solution appears, Figure 3-43. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

Check of the solution		
		Slab 1
V - Load:		
Total load	[kN] =	26000,0
Sum of contact pressures	[kN] =	25903,0
X - Moment:		
Sum Mx from loads	[kN.m] =	1143,6
Sum Mx from contact pressures	[kN.m] =	1155,2
Y - Moment:		
Sum My from loads	[kN.m] =	-137,2
Sum My from contact pressures	[kN.m] =	
Ok <u>H</u> elp		<b>∢</b> ▶

Figure 3-43 Menu "Check of the solution"

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

## 6 Viewing data and results

*ELPLA* can view the data and results for each raft separately or for the system of rafts together. Individual data or results can be viewed in a manner similar to that in the previous examples. Here the user will learn to view the results of the system of rafts together through the following two samples.

## 6.1 Viewing result graphics

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

*ELPLA-Graphic* window appears, Figure 3-44. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

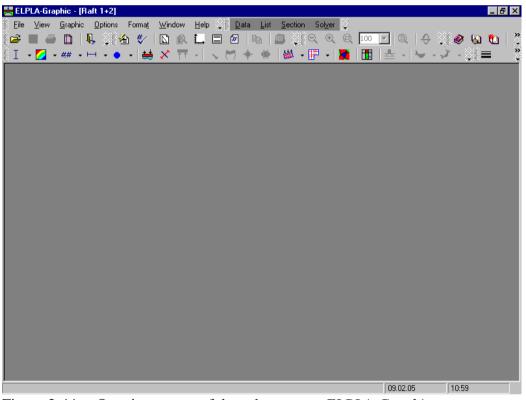


Figure 3-44 Opening screen of the sub program *ELPLA-Graphic* 

To view the results as contour lines

- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 3-45 appears
- In "Results as contour lines" option box select "Settlements *s*" as a sample for the results to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for the two rafts together as shown in Figure 3-46.

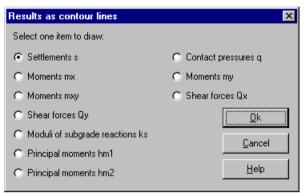


Figure 3-45 "Results as contour lines" option box

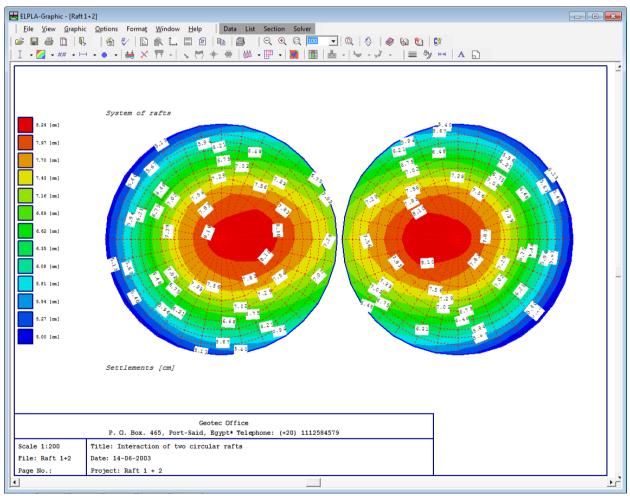


Figure 3-46 Settlements as contour lines

## 6.2 Plot a diagram of the results

To plot a diagram of the results, leave *ELPLA-Graphic* to *ELPLA-Section*. This is done by clicking on "Section" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Graphic* window.

*ELPLA-Section* window appears without file name in the window title bar, Figure 3-47. The disappearance of the file name in the window title bar indicates that *ELPLA-Section* cannot deal with the results of a system of rafts directly.

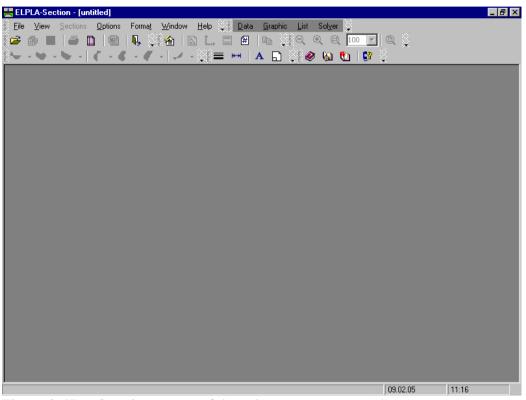


Figure 3-47 Opening screen of the sub program *ELPLA-Section* 

To consider the results for the system of rafts together

- Choose "Open" command from "File" menu of *ELPLA-Section*. Then open the project "Raft 1"
- Choose "Combination from many projects" command from "File" menu of *ELPLA*-*Section*. The following list box in Figure 3-48 appears. *ELPLA-Section* automatically considers the project "Raft 1" in the list of projects to be combined
- Click "Add project" button in "Combination from many projects" list box. Then open the project "Raft 2"
- Click "OK" button in the "Combination from many projects" list box

Co	mbinati	on from many projects		×
Γ	List of pr	ojects to be combined:		<u> </u>
	No.	File name of the project	Project identification	
	1	H:\Tutorial Manual\Englisch\Raft1	Raft 1	<u>C</u> ancel
				Add project
				<u>B</u> emove project
				New
				<u>H</u> elp

Figure 3-48 "Combination from many projects" list box

To plot a section in *x*-direction

- Choose "Section in *x*-direction" command from "Sections" menu of *ELPLA-Section*. The following option box in Figure 3-49 appears
- In "Section in *x*-direction" option box select "Settlements *s*" as a sample for the results to be plotted in a diagram
- Click "OK" button

×
C Contact pressures q
C Moments my
C Shear forces Qx
<u> </u>
Cancel
Help

Figure 3-49 "Section in *x*-direction" option box

The following option box in Figure 3-50 appears, to specify the section in x-direction that is required to be plotted.

In this dialog box

- Type 11 in the "Section at *y*-coordinate" edit box, to plot a diagram at the middle of the two rafts
- Click "OK" button

The settlements are now plotted in a diagram at the middle of the two rafts together as shown in Figure 3-51.

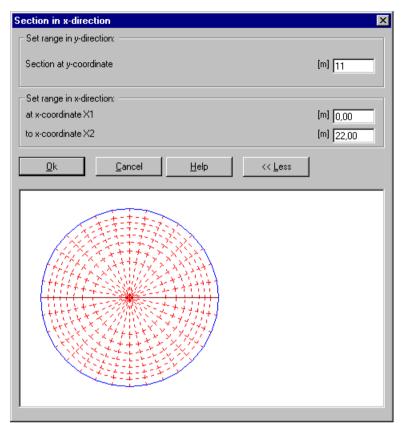


Figure 3-50 "Section in *x*-direction" dialog box

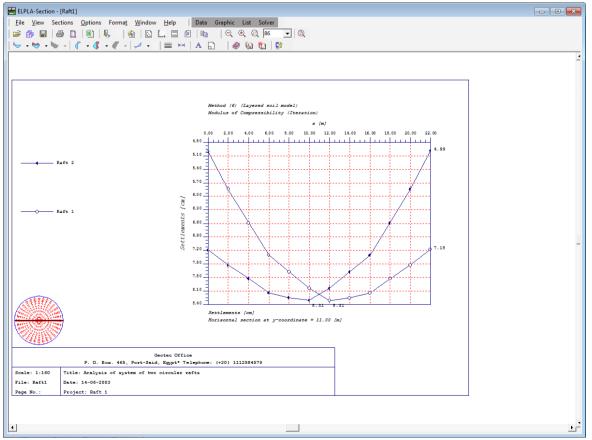


Figure 3-51 Diagram of settlements in x-direction at nodal row No. 14 for the two rafts

# 7 Index

# C

Calculation	
Calculation method	
Check of the solution	
Computation of all	
Contact pressures	
Contour lines	

# D

Deformation	26
Diagram	, 32

## F

FE-Net	9
Flexibility coefficients	
Foundation properties	16, 20

# G

Generation parameters
Graphic
1

# I

Internal forces	
Iteration process	. 26, 27

# L

Load vector	
Loads	

## M

```
Main soil data .....14
```

# 0

## P

Point loads	4, 18
Project identification	.8, 19, 23

## S

Save project as	
Slab type	
Soil	4
Soil properties	4, 11, 15
Soil stiffness matrix	
System symmetry	6

# U

Unit weight
-------------

Example 4

Analysis of a grid

## ELPLA-Tutorial

## Contents

# Page

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

## **1** Description of the problem

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

### 1.1 Loads and dimensions

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

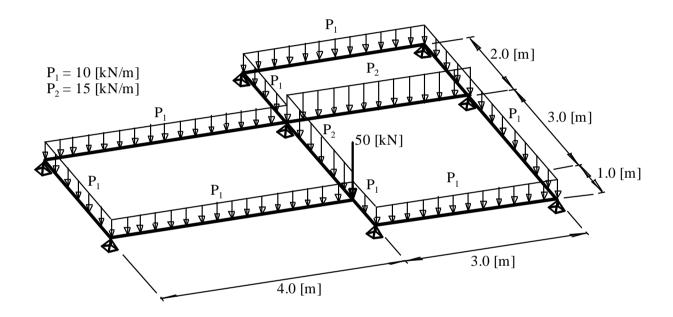


Figure 4-1 Geometry of the grid and loads

## 1.2 Grid material

Material of the grid has the following parameters:

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

## **1.3** Analysis of the grid

It is required to analyze the grid using 1.0 [m] beam element in both *x*- and *y*-directions and considering the supports to be rigid in *z*-direction only. This Tutorial Manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis a complete reference for numerical calculation methods are well documented in the User's Guide of *ELPLA*.

## 2 Creating the project

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

## 2.1 Calculation method

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

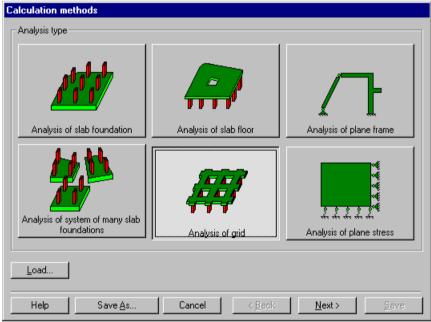


Figure 4-2 "Calculation method" wizard with "Analysis type" form

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

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

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

Calculation methods	
System symmetry:	
Unsymmetrical system	
Symmetrical system about x-axis	Double-symmetrical system
Symmetrical system about y-axis	Anti-symmetrical system in x-axis
Help Save <u>A</u> s Car	ancel < <u>B</u> ack <u>Next&gt;</u> <u>S</u> ave

Figure 4-3 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 4-4. In this list *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other.

In this list

- Check "Supports/ Boundary conditions" check box
- Click "Save" button

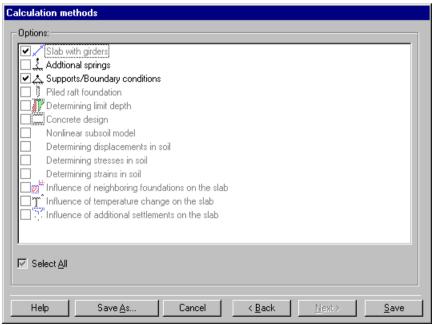


Figure 4-4 "Options" list

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

In this dialog box

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

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

Save As		? ×
Save jn: 🔁	ELPLA PE 8.0 🔽 🖛 🗈 🗃 🖽 -	
Floor Grid Raft 1 Raft 2		
File <u>n</u> ame:	Grid Save	
Save as <u>t</u> ype:	Isolated slab foundation-files (*.P01)	

Figure 4-5 "Save as" dialog box

### 2.2 **Project identification**

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

In this dialog box

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

Ρ	roject ide	ntification	×
	- Project id	entification:	
	Title	Analysis of a grid	
	Date	Mondy, 14. June 2003	
	Project	Grid	
	<u>S</u> ave	Cancel Help Load Save As	

Figure 4-6 "Project identification" dialog box

## 2.3 FE-Net data

To enter the grid geometry, an imaginary net must be first defined with suitable dimensions and then the grid is entered lately by connecting the corresponding nodes on the net that defines the grid shape. To define the net for the grid geometry, choose "FE-Net data" command from "Data" menu. The "FE-Net generation" wizard appears as shown in Figure 4-7. This wizard will guide you through the steps required to generate the FE-Net. The first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have constant size in both *x*- and *y*-directions.

To generate an imaginary net

- Choose the rectangular slab option to create an imaginary net of a rectangular area
- Type 7 in the "Length of rectangular slab" edit box
- Type 6 in the "Width of rectangular slab" edit box
- Click "Next" button to go to the next form

FE-Net generation	
Slab type	
Rectangular slab:	
Length of rectangular slab	L [m] 7
Width of rectangular slab	B [m] 6
Help Cancel < Back	

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

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

In this form

- Choose rectangular elements
- Click "Next"

FE-Net generation			
Generation type			
		× · · · · · · · · · · · · · · · · · · ·	······································
	Cancel	< Back Next >	Einish

Figure 4-8 "Generation type" form

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

FE-Net generation		
Grid definition		
Grids in x-direction		
Constant grid interval		
No. of grid intervals	7	
Grid interval Dx [m]	1,00	
Grids in y-direction		
Constant grid interval		
No. of grid intervals	6	
Grid interval Dy [m]	1,00	
Help	Cancel < <u>B</u> ack	Next > Einish

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

In this dialog box

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

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

📥 EL	PLA-D	ata - [Grid]	- [FE-N	let]												- 8 ×
Ele	 ⊻iew	<u>G</u> raphica	ally FE-	<u>N</u> et Gene	eration	<u>I</u> n table	<u>O</u> ptions	Forma <u>t</u>	<u>W</u> indow	<u>H</u> elp	_					• <del>x</del>
Ē	🖻 🖡	2 🖆 🕻	台	∜∕ ≜	<b>J</b> .!	= 🦻	₩ I	<b>↓</b> (	ર્ ભ્	Q 100	<u> </u>	. Ø	<b>6</b>	6	*	
ĮΩ.		• * ·						* *	·   🎎 (	4						
I				<u> </u>									_			
I .																
													_			
I .																
I .																
													-			
I .																
													-			
I .																
I .																
													-			
I .																
····													-			
····													-			<b>T</b>
•	1	1		1	1		1			1						

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

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

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

## 2.4 Girders

To define the girders, choose "Girders" command from "Data" menu of *ELPLA-Data* window. The following embedded program in Figure 4-11 appears.

	ta - [Grid] -		Forma <u>t</u> <u>W</u> i	adam Hala					× 8 -
					ବ୍ର୍ହ୍	100 💌 🔘	Q 📜 🤣 😡	i 🛍   📴 🖕	* <u>×</u>
	× * 🔨		oups 🦻						
		 		!					-
		 		+	   		4 1 1		
L	t	 i		<u>.</u>		L	1		•
	11	 1 11		ad maga					

Figure 4-11 "Girders" embedded program

To enter the cross section of the girders

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

Cross section definition	×
Cross section definition:	<u>0</u> k
	<u>C</u> ancel
<ul> <li>General cross section</li> <li>C Create a new element group as T/L-girder</li> </ul>	Help

Figure 4-12 "Cross section definition" option box

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

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

Girder gro	oups (with th	e same prop	oerties)			×
Group No.	E-Modulus of girder E [kN/m2]	G-Modul of girder G [kN/m2]	Height ofgirder h [m]	Width of girder b [m]	Girder weight pb [kN/m]	<u>O</u> k <u>C</u> ancel
1 2	3,2E+07 3,2E+07	1,3E+07 1,3E+07	0,60 0,60	0,15 0,15	10,0 15,0	<u>I</u> nsert
						<u>С</u> ору
						<u>D</u> elete
						New
						Help
						Excel

Figure 4-13 "Defining girder groups" list box

## Defining the girder locations on the net

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

- - Choose "Add girders" command from the "Graphically" menu in Figure 4-11. When "Add girders" command is chosen, the cursor will change from an arrow to a cross hair
  - Click the left mouse button on the start node of the first girder and drag the mouse until the end node of that girder (Figure 4-14). Then click on the end node. The "Girder elements" dialog box in Figure 4-15 appears

N   X ¥ X			] 🔍 🚅 🥔 😡	

Figure 4-14 Add girder by mouse

In this dialog box

- Select the group No
- Click "OK" button

Girder elements	×
Girder elements:	
Group No.	0 1 💌
Start from node No.	[·] 26
End at node No.	[·] [30
<u>Ok</u> <u>C</u> ancel	<u>H</u> elp

Figure 4-15 "Girder elements" dialog box

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

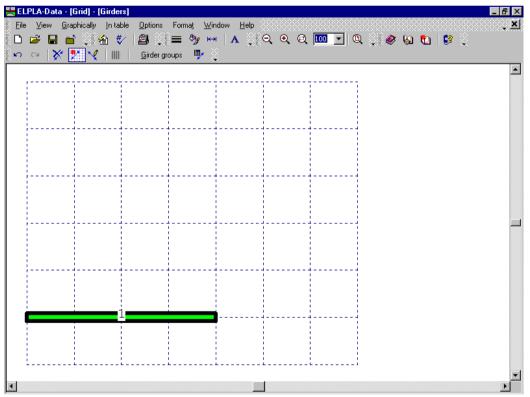


Figure 4-16 First girder on the screen

Repeat the previous steps to add the remaining girders on the net. After you have completed the definition of all girders, the screen should look like the following Figure 4-17.

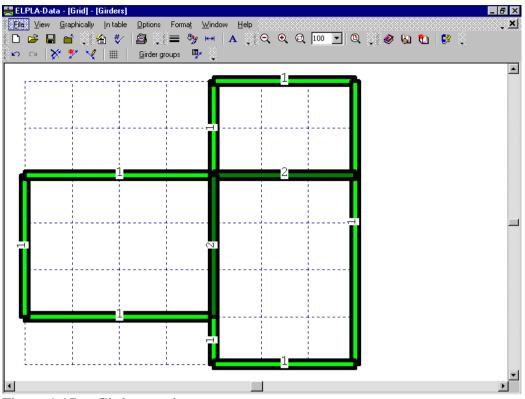


Figure 4-17 Girders on the screen

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

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

## 2.5 Supports/ Boundary conditions

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

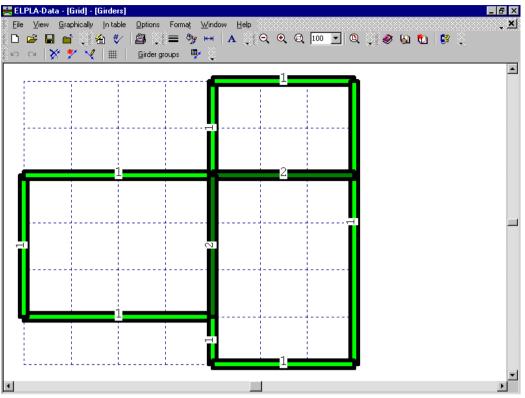


Figure 4-18 "Supports/ Boundary conditions" embedded program

*ELPLA* can display girders, supports, loads etc. in one view together. The advantage of this option is that the user can control easily locations of supports or loads on the net when entering the rest of the data. In the case of analysis of girder problems, *ELPLA* displays girders during input of other data. As shown in Figure 4-18, girders are drawn with the actual thickness.

To view girders as simple lines

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

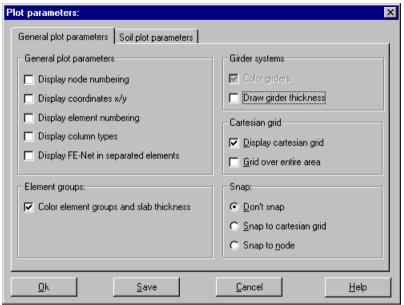


Figure 4-19 "Plot parameters" dialog box

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

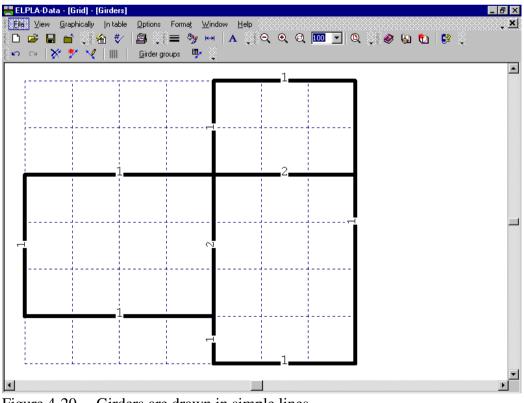


Figure 4-20 Girders are drawn in simple lines

### **Defining supports on the net**

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

To define supports on the net

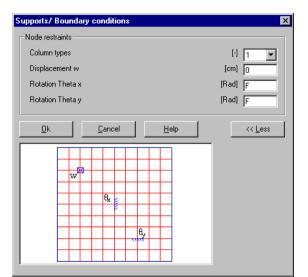
- Choose "Select nodes" command from the "Graphically" menu in Figure 4-20. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nodes that have supports as shown in Figure 4-21
- After selecting nodes of supports choose "Add boundaries" command from "Graphically" menu in Figure 4-21. "Supports/ Boundary conditions" dialog box in Figure 4-22 appears

<u>-</u> Eile <u>V</u> iew <u>G</u> raphically <u>I</u> n ta	orts/ Boundary conditions] ble Options Format Window			-   <del>0</del>
) ☞ 님 酋 ़ ⁄≙ : › ∝   ₩ ፟፟ ፟ ፟ ፟ ፟ ፟ ፟	१/   🖾 📜 ≡ 🦘 ⊷   ↓   🎒 🖳 .	<b>५</b>	<u>─</u>  ♥,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,	<b>e</b> a *
		<b></b> 1 <b></b>		
		2		
1	Ĩ			
			⊊⇒ ∐	
	2			
<b>Ğ</b>	1			
		]	<b>B</b>	

Figure 4-21 Selection of nodes that have supports

In this dialog box

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



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

ELPLA can calculate the punching stresses due to column reactions. In this example data corresponding to column dimensions are not required. Therefore the user can take these data from the default column dimensions and consider all supports have column type 1. After you have completed the definition of supports, the screen should look like the following Figure 4-23.

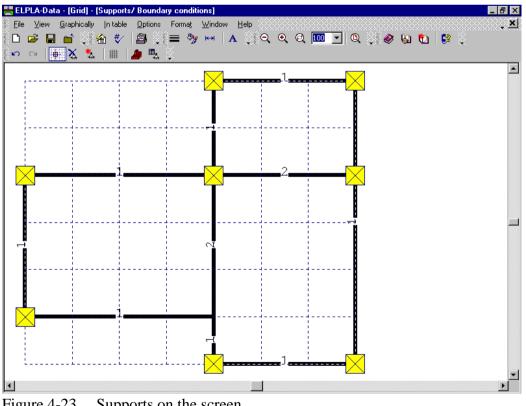


Figure 4-23 Supports on the screen

After entering supports, do the following two steps:

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

### 2.6 Loads

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

<u>F</u> ile	⊻ie	w <u>(</u>	<u>G</u> raphic	ally	<u>I</u> n tal	ole	<u>U</u> sing	form	ula	<u>O</u> pti	ions	For	maţ	∐ir	ndow	<u>H</u> e	lp									
																	⊇	100	•	©,	• • •	8	۵ 🐌	1   5	<mark>?</mark>	
0		**↓	*	*ŧ	*	<del>330</del>	24	27	<u> </u>	10	,   3		<b>*</b>		, ,											
											<b>f</b>					1	-			7						
1																	÷									
																2										
					-											_										
								!									1			L.						
Ì											2															
																	-+									
ļ			-			-				-							÷									
											-4															
																1										
											5						_									

Figure 4-24 "Loads" embedded program

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

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

In this dialog box

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

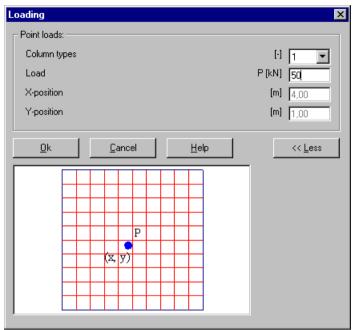
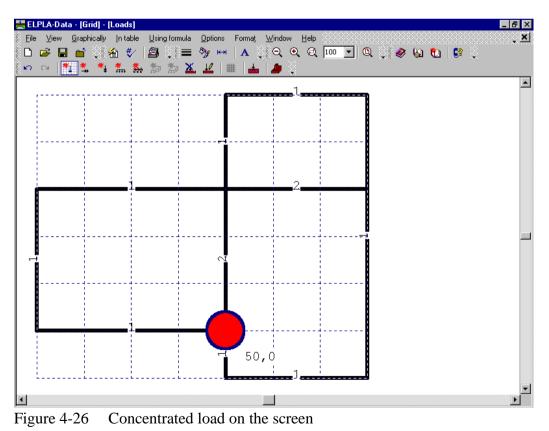


Figure 4-25 "Point loads *P*" dialog box

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



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

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

Creating the project of the grid is now complete. See how to use *ELPLA* for analyzing projects.

## **3** Carrying out the calculations

## 3.1 Starting *ELPLA-Solver*

To analyze the problem, switch to *ELPLA-Solver* by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data* window (Figure 4-27).

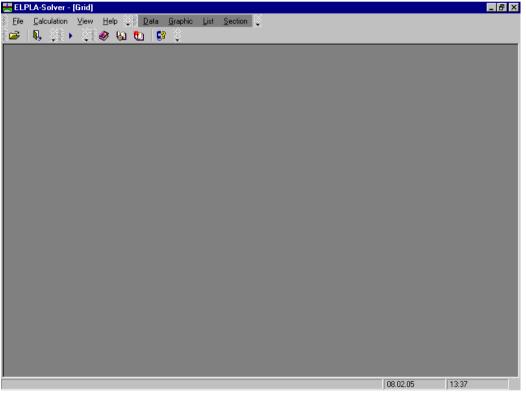


Figure 4-27 Opening screen of the sub program *ELPLA-Solver* 

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. Also *ELPLA* will activate the "Calculation" menu. This menu contains commands of all calculations, which depend on the analysis type. For the current example the items, which are required to be calculated, are:

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

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

#### **3.2** Carrying out all computations

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

#### Analysis progress

Analysis progress menu in Figure 4-28 appears, in which various phases of calculation are progressively reported as the program analyzes the problem. Also a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

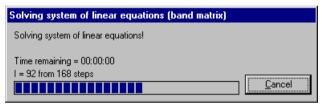


Figure 4-28 Analysis progress menu

#### Check of the solution

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

Check of the solution	
V - Load:	
Total load	[kN] = 400.0
Sum of reactions	
Sum or reactions	[kN] = 400,0
X - Moment:	
Sum Mx from loads	[kN.m] = -117,5
Sum Mx from reactions	[kN.m] = -117.5
Sum MX non reactions	forcini = 11175
Y - Moment:	
Sum My from loads	[kN.m] = 257,5
Sum My from reactions	[kN.m] = 257,5
	[]
[	
Ok 📗	<u>H</u> elp

Figure 4-29 Menu "Check of the solution"

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

## 4 Viewing data and result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window. *ELPLA-Graphic* window in Figure 4-30 appears. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

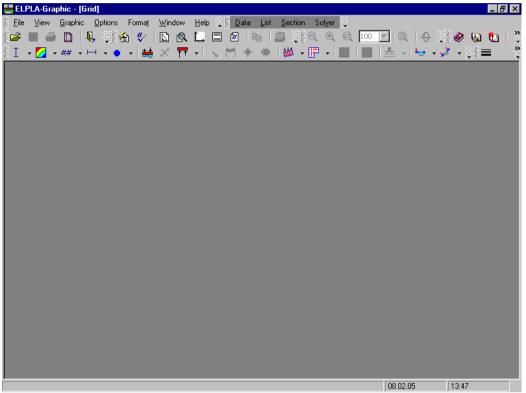


Figure 4-30 Opening screen of the sub program *ELPLA-Graphic* 

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

In this option box

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

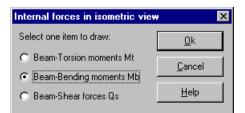


Figure 4-31 "Internal forces in isometric view" option box

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

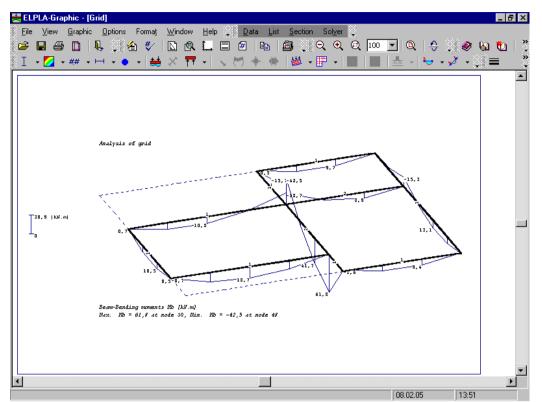


Figure 4-32 Beam-Bending moments *Mb* 

# 5 Index

# A

Analysis	3
Analysis type	

# B

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

# С

Calculation	
Calculation methods	
Check of the solution	
Computation of all	
Cross section	

# D

# F

E-Net generation7

# G

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

# Ι

Internal forces	22

# L

Load vector	20
Loads	3, 14, 18

# N

# P

Point loads	.18
Project identification	6

# S

Save as	6
Slab stiffness matrix	
Solver	
Supports	

Example 5

# Plane stress analysis of a wrench

### ELPLA-Tutorial

# Contents

# Page

1	Desc	cription of the problem	3
	1.1	Loads and dimensions	3
	1.2	Wrench material	3
	1.3	Analysis of the Wrench	3
	1.4	System Units	3
2	Crea	ting the project	4
	2.1	Calculation method	4
	2.2	Project identification	7
	2.3	FE-Net data	7
	2.4	Supports/ Boundary conditions	14
	2.5	Slab properties	17
	2.6	Loads	19
3	Carry	ying out the calculations	22
	3.1	Starting <i>ELPLA-Solver</i>	22
	3.2	Carrying out all computations	22
4	View	ving data and result graphics	23
5	Index	X	27

### **1** Description of the problem

An example for analyzing stresses and deformations of a wrench is selected to illustrate some features of *ELPLA* for analyzing plane stress problems.

### 1.1 Loads and dimensions

The wrench shown in Figure 5-1 is 10 [mm] thick and about 200 [mm] in length. The wrench end is loaded with a pressure of 2 [N/mm] along 100 [mm] of its handle as shown.

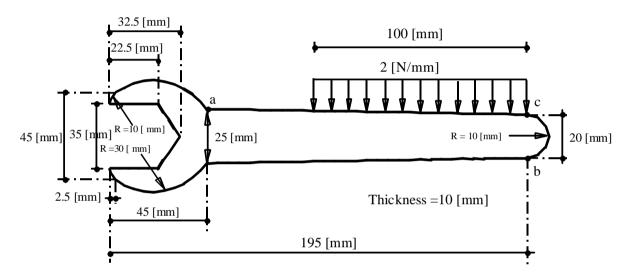


Figure 5-1 Wrench dimensions and loads

### 1.2 Wrench material

Material of the wrench has the following parameters:

Young's modulus	$E_b$	= 200000	$[N/mm^2]$
Poisson's ratio	$v_b$	= 0.3	[-]

### **1.3** Analysis of the wrench

It is required to analyze the stresses and deformations of the wrench when it is being used to tighten a bolt. Consider that there is no horizontal or vertical displacement along the lines where the wrench contacts the bolt. This Tutorial Manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis a complete reference for numerical calculation methods are well documented in the User's Guide of *ELPLA*.

### 1.4 System units

Since the wrench dimensions are relatively small compared with those of foundations, it is preferred to use small units for measurement. To change the system units, choose "System of units" command from "Main data" menu of *ELPLA-Data*. The dialog box in Figure 5-2 appears. In this dialog box change the units for lengths and forces to [mm] and [N] respectively and then click "Save" button.

9	system of units	×
	System of units:	
	Lengths (1): Depths, coordinates, dimensions, thickness	millimeter, [mm]
	Lengths (2): Reinforcement, concrete cover, settlements, eccentricity	millimeter, [mm]
	Forces (1): Loads, contact pressures, stresses	newton, [N]
	Forces (2): Punching shear stress, modulus of Compressibility, modulus of Elasticity	newton, [N]
	Temperature	Celsius (centigrade), [°C] 💌
	<u>Save</u>	<u>H</u> elp

Figure 5-2 "System of units" dialog box

Also *ELPLA* checks the element size when a mesh is generated. Since the element size of slab foundation is relatively large compared with those of a wrench, it is required to disable the element size check. To uncheck the element size when generating FE-Net, choose the "Preferences" command from "Main data" menu of *ELPLA-Data*. The dialog box in Figure 5-3 appears. In this dialog box uncheck the "Check element size" option then click "Save" button.

Preferences		×
FE-Net preferences:		
Check element overlaps		
Check element size		
Minimum distance between nodes	[mm]	50.00
Calculation preferences: The Internal forces are determined at: the element centers and then distributed to the the element nodes	e element no	des
<u>S</u> ave <u>C</u> ancel		<u>H</u> elp

Figure 5-3 "Preferences" dialog box

### 2 Creating the project

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

### 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation methods" wizard appears, Figure 5-4. The wizard will guide you through the steps required to create the system data of the project. As shown in this Figure, the first form of the wizard is the "Analysis type" form.

Calculation methods Analysis type:		
Analysis of slab foundation	Analysis of slab floor	Analysis of plane frame
Analysis of system of many slab foundations	Analysis of grid	Analysis of plane stress
Help Save As	Cancel < Back	<u>N</u> ext > <u>S</u> ave

Figure 5-4 "Calculation method" wizard with "Analysis type" form

In the "Analysis type" form in Figure 5-4, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a plane stress problem, select "Analysis of plane stress" option then click "Next" button to go to the next form.

The next form is the "System symmetry" (Figure 5-5). In this form choose "Unsymmetrical system" and click "Next" button.

The last form of the wizard assistant contains the "Option" list, Figure 5-6. In this list *ELPLA* displays some of the available options corresponding to the used numerical models, which differ from model to other.

In this list check "Supports/ Boundary conditions" check box then click "Save" button.

Calculation methods				
System symmetry:				
Unsymmetrical syst	em			
Symmetrical system about	ut x-axis	Double-symmet	rical system	
Symmetrical system about	ut y-axis	Anti-symmetrical s	ystem in x-axis	
Help Save <u>A</u> s	Cancel	< <u>B</u> ack	Next>	<u>S</u> ave

Figure 5-5 "System symmetry" form

Calculation methods
Options:
Slab with girders
🗌 🗌 🛴 Addtional springs
In International Conditions
Determining limit depth
Concrete design
Nonlinear subsoil model
Determining displacements in soil
Determining stresses in soil
Determining strains in soil
Influence of neighboring foundations on the slab
Influence of the temperature change on raft
Influence of additional settlements on raft
I Select All
Jan Jold All
Help         Save As         Cancel         < Back

Figure 5-6 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 5-7.

In this dialog box, type a file name for the current project in the file name edit box. For example type "Wrench". *ELPLA* will use automatically this file name in all reading and writing processes. Then click "Save" button to complete the definition of the calculation method and the file name of the project.

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

Save As					? ×
Savejn: 🔂	Examples	- 🗈	<u></u>	Ť	
Floor Example					
, File <u>n</u> ame:	Wrench				<u>S</u> ave
Save as <u>t</u> ype:	Isolated slab foundation-files (*.P	01)	•		Cancel

Figure 5-7 "Save as" dialog box

### 2.2 **Project identification**

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 5-8 appears.

In this dialog box

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

Ρ	roject ide	ntification	×					
[	Project ide	ntification:						
	Title Analysis of a Plane Stress Problem							
	Date	ate Monday, December 05, 2005						
	Project	Wrench						
	<u>S</u> ave	E <u>C</u> ancel <u>H</u> elp <u>L</u> oad Save <u>A</u> s						

Figure 5-8 "Project identification" dialog box

### 2.3 FE-Net data

To define the net for the wrench geometry, choose "FE-Net data" command from "Data" menu. The "FE-Net generation" wizard appears as shown in Figure 5-9. This wizard will guide you through the steps required to generate a FE-Net. The first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have constant size in both *x*- and *y*-directions.

To generate the FE-Net, choose the irregular slab option then click "Next" button to go to the next form.

FE-Net Generation				
				0
Rectangular slab: Length of rectangular slab Width of rectangular slab			L [mm] B [mm]	20.00
Help	<u>C</u> ancel	< <u>B</u> ack	<u>N</u> ext>	<u> </u>

Figure 5-9 "FE-Net generation" wizard with "Slab type" form

After clicking "Next" in the "FE-Net generation" wizard, the following "Generation type" form appears, Figure 5-10. *ELPLA* can deal with various types of generations with triangle and/ or rectangular elements. In this form, choose the triangle elements option then click "Next".

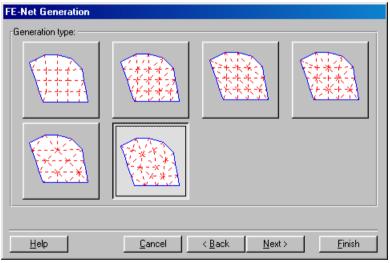


Figure 5-10 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "FE-Net generation" dialog box in Figure 5-11 appears with the default generation parameters. In this dialog box click "Finish" button.

FE-Net Generation				
Generation parameters:				
Min. angle			theta [*]	30
Element circumradius			r [mm]	1.00
Min. element circumradius			rmin (mm)	0.25
Mesh optimization:				
<u>H</u> elp	<u>C</u> ancel	< <u>B</u> ack	<u>N</u> ext >	<u> </u>

Figure 5-11 "FE-Net generation" dialog box

*ELPLA* will generate an imaginary net as shown in the following embedded program, Figure 5-12.

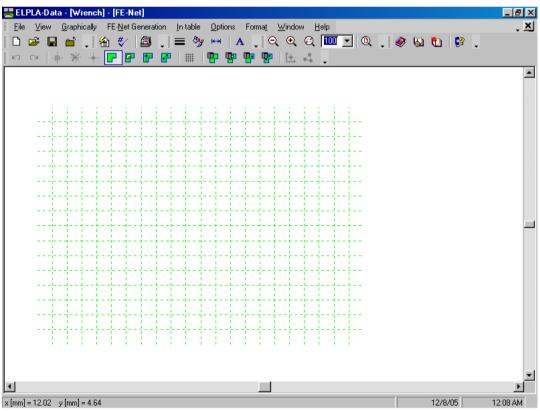


Figure 5-12 Imaginary net of a rectangular area on the screen

You can either draw the wrench directly on this imaginary net or input the wrench corner points in tabulated form. To input the FE-Net corner points, choose "Slab corners" command from "In Table" menu. The dialog box in Figure 5-13 appears.

To define the first segment of the wrench which connects points a and b (Figure 5-1)

- Type 45 in the "x1" edit box to define the x-coordinate of point a
- Type 32.5 in the "y1" edit box to define the y-coordinate of point a
- Type 190 in the "x2" edit box to define the x-coordinate of point b
- Type 30 in the "y2" edit box to define the y-coordinate of point b

Start poistion	x1	[mm] 45.00			
	y1	[mm] 32.50			
End position	x2	[mm] 190.00			
	у2	[mm] 30.00			
🗖 Use arc data			1		
Arc radius	R	[mm] 0.00			
Min. arc radius	Rmin	[mm] 0.00	Segment <u>c</u> opy		
🗖 Reverse rotatio	on direction	I	Segment insert		
🗖 Reverse radiu:	s position				

Figure 5-13 "Slab corners" dialog box

To define the second segment of the wrench which connects points b and c

- Use the Scroll bar to define segment No. 2
- Type 190 in the "x2" edit box to define the x-coordinate of point c
- Type 10 in the "y2" edit box to define the y-coordinate of point c
- Select the "Use arc data" option to convert the line segment to arc segment
- Type 10 in the "Arc radius" edit box to define the radius of the arc segment

In the "Slab corners" dialog box, *ELPLA* assumes that there are at least three segments with three corner points. Since the wrench drawing contains 11 segments, you can use the command "Segment insert" to insert the rest of the wrench segments. Use the corner points and arc information provided in Table 5-1 to complete the definition of the wrench corner points. Repeat the steps used for defining segment No. 1 to define any line segment and repeat those used for segment No. 2 to define any arc segment.

Segment	Start position		End position		Arc Radius
	$x_1$	<i>y</i> 1	$x_2$	<i>y</i> 2	
[-]	[mm]	[mm]	[mm]	[mm]	[mm]
1	45	32.5	190	30	
2	190	30	190	10	10
3	190	10	45	7.5	
4	45	7.5	2.5	0	30
5	2.5	0	0	5	10
6	0	5	22.5	5	
7	22.5	5	32.5	20	
8	32.5	20	22.5	35	
9	22.5	35	0	35	
10	0	35	2.5	40	10
11	2.5	40	45	32.5	30

Table 5-1Wrench Corner Points

After completing the definition of the wrench corner points, the slab corner dialog box should be like this in Figure 5-14 where a small sketch of the wrench appears in the dialog box picture. Click the "OK" button to see the drawing of the wrench outlines as shown in Figure 5-15.

ab corners			
ilab corners: Segment No. 11 from " Segment data: ——	11 segment	s:	
Start poistion	×1	[mm] 2.50	
	y1	[mm] 40.00	
End position	x2	[mm] 45.00	
	y2	[mm] 32.50	
🔽 Use arc data			
Arc radius	R	[mm] 30.00	
Min. arc radius	Rmin	[mm] 21.58	Segment copy
🗖 Reverse rotati	on direction		Segment insert
Reverse radiu:	s position		Segment delete
<u>0</u> k	<u>C</u> an	cel	<u>H</u> elp <u>N</u> ew <u>R</u> efresh

Figure 5-14 "Slab corners" dialog box after defining the wrench corner points

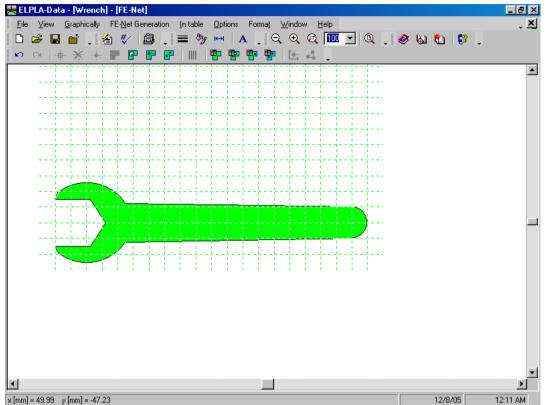


Figure 5-15 Wrench outlines

To complete the FE-Generation of the problem, choose "Generating FE-Net" command from "FE-Generation" menu. The following "Generation of FE-Net" dialog box appears.

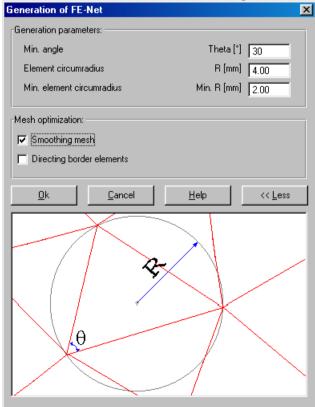


Figure 5-16 "Generation of FE-Net" dialog box

To enter the generation data

- Type 4 in the "Element circumradius" edit box to define the radius of circles containing elements
- Type 2 in the "Min. element circumradius" edit box to define the minimum radius of circles containing elements
- Check the "Smoothing mesh" option to optimize the dimension of FE-Net by making all elements having nearly the same area as possible as
- Click "OK" button

After clicking the "OK" button, FE-Net generation progress menu in Figure 5-17 appears in which various phases of the generation are progressively reported. After finishing the generation of the FE-Net and creating the wrench FE-Net as shown in Figure 5-18, do the following two steps:

- Choose "Save FE-Net" command from "File" menu in Figure 5-18 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 5-18 to close the "FE-Net" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "FE-Net data" command in the "Data" menu of *ELPLA-Data*.

The generation of FE-Net is carried out!	0
Please wait!	
	Abort

Figure 5-17 "FE-Net generation" progress menu

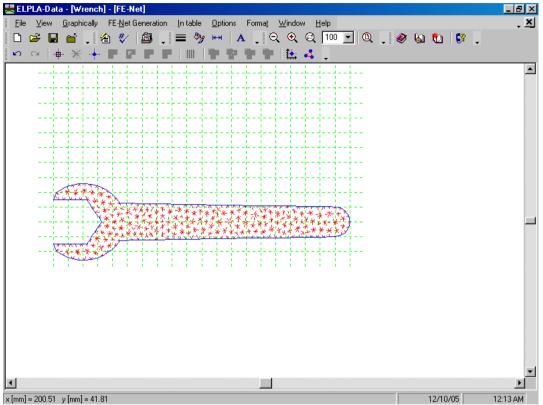


Figure 5-18 Final FE-Net of the wrench

### 2.4 Supports/ Boundary conditions

To define supports, choose "Supports/ Boundary conditions" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 5-19 appears.

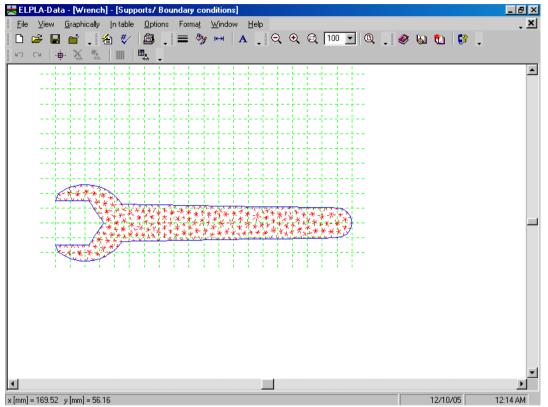


Figure 5-19 "Supports/ Boundary conditions" embedded program

### **Defining supports on the net**

Defining supports or boundary conditions on the net may be carried out either graphically or numerically (in a table). In the current example the user will learn how to define supports on the net graphically. Considering that there are no horizontal or vertical displacements along the lines where the wrench contacts the bolt, all constrains will be applied only to the upper and bottom jaws of the wrench.

To define supports on the net

- Choose the "Zoom window" command from the "Window" menu to zoom the wrench head
- Choose "Select nodes" command from the "Graphically" menu in Figure 5-20. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nods that have supports as shown in Figure 5-20
- After selecting nodes of supports, choose "Add boundaries" command from "Graphically" menu in Figure 5-20. "Supports/ Boundary conditions" dialog box in Figure 5-21 appears

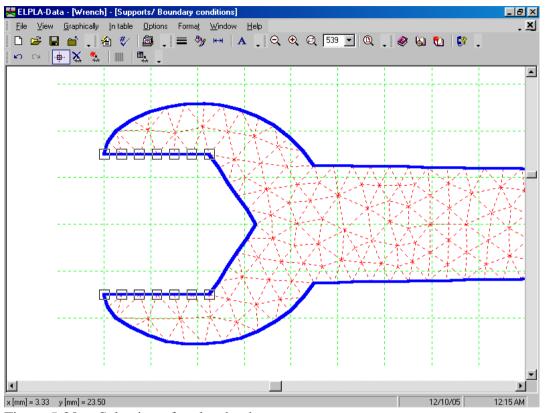


Figure 5-20 Selection of nodes that have supports

In this dialog box

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

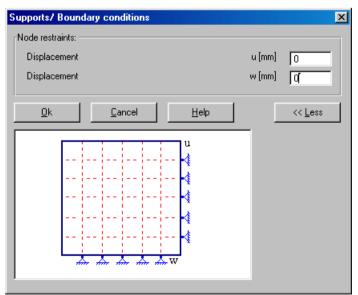


Figure 5-21 "Supports/ Boundary conditions" dialog box

After you have completed the definition of supports, the screen should look like the following Figure 5-22.

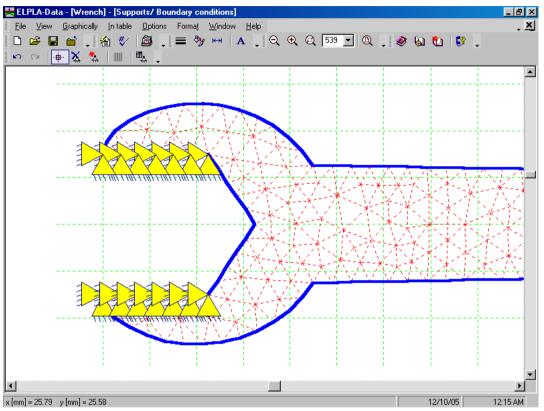


Figure 5-22 Supports on the screen

After entering supports, do the following two steps:

- Choose "Save supports/ Boundary conditions" command from "File" menu in Figure 5-22 to save the data of supports
- Choose "Close supports/ Boundary conditions" command from "File" menu in Figure 5-22 to close the "Supports/ Boundary conditions" embedded program and to return to the main window of *ELPLA-Data*

### 2.5 Slab properties

To define the material properties, choose "Slab properties" command from "Data" menu. The following embedded program in Figure 5-23 appears with default slab properties. The data of material properties for the current example, which are required to be defined, are wrench material and wrench thickness.

To enter the wrench material and thickness, choose "Element groups" command from "In Table" menu. The following list box in Figure 5-24 appears. In this list box, enter the E-Modulus and *Poisson's* ratio of the wrench material and wrench thickness. Then, click "OK" button to go to the next step.

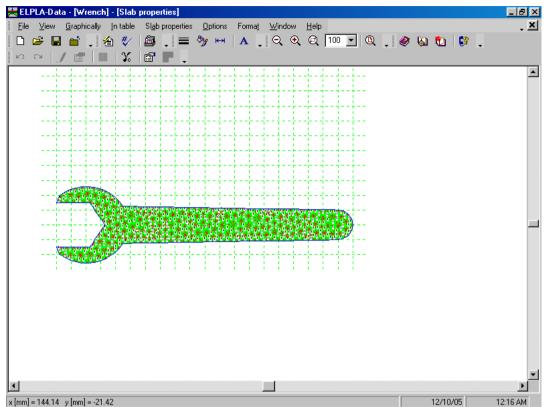


Figure 5-23 "Slab properties" embedded program

D	efining	element grou	ps (with the sam	e thickness and m	aterial) 🛛 🗙
	Group No.	E-Modulus of slab [N/mm2]	Poisson's ratio of slab [-]	Slab thickness d [mm]	<u><u> </u></u>
	1	200000	0.3	10	<u>C</u> ancel
					<u>I</u> nsert
					<u>С</u> ору
					<u>D</u> elete
					New
					<u>H</u> elp
					<u>E</u> xcel

Figure 5-24 "Defining element groups" list box

To enter the unit weight of the wrench material, choose "Unit weight of slab material" command from "Slab properties" menu in the window of Figure 5-23. The following dialog box in Figure 5-25 with a default unit weight of 25  $[kN/m^3]$  appears. To neglect the self-weight of the wrench in the analysis, type 0 in the "Unit weight of slab material" edit box then click "OK" button.

Unit weight of slab material	X
Unit weight of slab material	Gb [N/mm3] 0
<u>Ok</u> ew	<u>C</u> ancel <u>H</u> elp

Figure 5-25 "Unit weight of slab material" dialog box

After entering the wrench material properties, do the following two steps:

- Choose "Save Slab properties" command from "File" menu in Figure 5-23 to save the foundation properties
- Choose "Close Slab properties" command from "File" menu in Figure 5-23 to close the "Slab properties" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside "Slab properties" command in "Data" menu of *ELPLA-Data*.

### 2.6 Loads

To define the load applied to the wrench, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 5-26 appears.

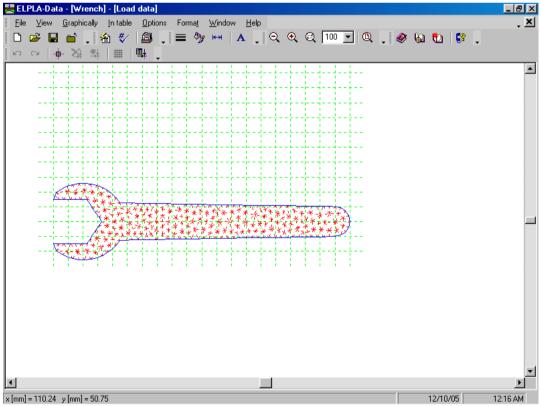


Figure 5-26 "Loads" embedded program

In plane stress problems, loads can be applied to nodes only. In this example the line load applied to the wrench will be converted to vertical concentrated loads applied to the wrench FE-Net nodes.

To enter the vertical concentrated load

- Choose the "Zoom window" command from the "Window" menu to zoom the wrench end as shown in Figure 5-27
- Choose "Select nodes" command from the "Graphically" menu in Figure 5-26. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nodes that would have loads. Since the total force is 200 [N] over a length of 100 [mm] and the nodes are generated at about [4] mm spacing, choose 25 nodes from the wrench end as shown in Figure 5-27
- After selecting nodes of loads, choose "Add nodal loads" command from "Graphically" menu in Figure 5-27. "Nodal loads" dialog box in Figure 5-28 appears

In this dialog box

- Type 8 in the "Load Pv" edit box to define the vertical nodal loads
- Click "OK" button

After you have completed the definition of the concentrated loads, the screen should look like Figure 5-29.

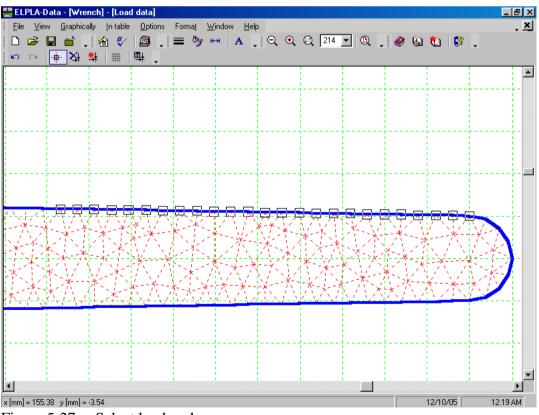


Figure 5-27 Select load nodes

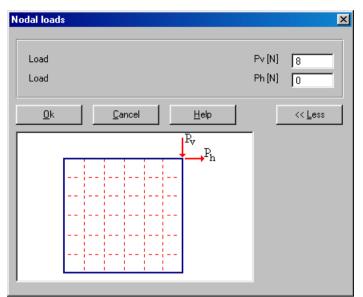


Figure 5-28 "Nodal loads" dialog box

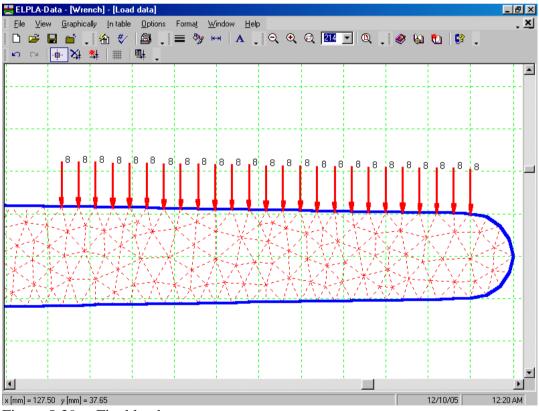


Figure 5-29 Final loads

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

- Choose "Save loads" command from "File" menu in Figure 5-29 to save the load data
- Choose "Close loads" command from "File" menu in Figure 5-29 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

Creating the project of the wrench is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

### **3** Carrying out the calculations

### 3.1 Starting *ELPLA-Solver*

To analyze the problem, switch to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window in Figure 5-30 appears.

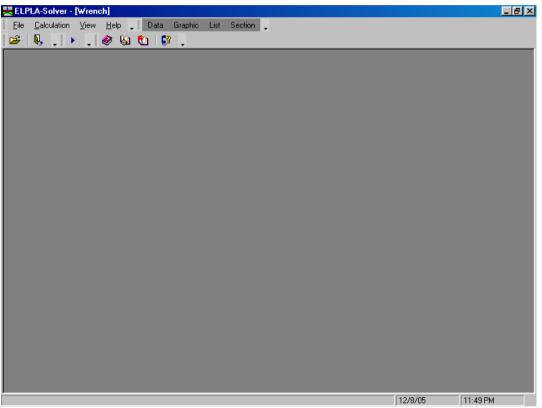


Figure 5-30 Opening screen of the sub program ELPLA-Solver

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. Also, *ELPLA* will active the "Calculation" menu. This menu contains commands of all calculations. Commands of the calculation depend on the analysis type. For the current example, the items, which are required to be calculated, are:

- Preparing the calculation
- Analysis of plane stress

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

### **3.2** Carrying out all computations

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

### Analysis progress

Analysis progress menu in Figure 5-31 appears in which various phases of the calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

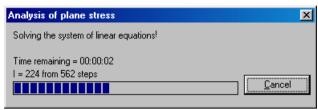


Figure 5-31 Analysis progress menu

### 4 Viewing data and result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window. *ELPLA-Graphic* window in Figure 5-32 appears. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

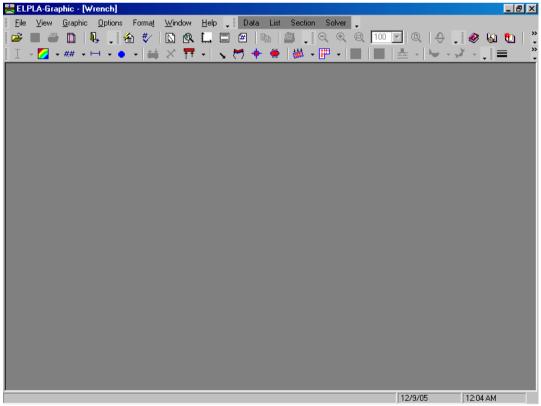


Figure 5-32 Opening screen of the sub program *ELPLA-Graphic* 

*ELPLA* can show solid elements in three dimension view. To view elements with real dimension in 3D-view, choose "Plot parameters" command from "Options" menu of *ELPLA-Graphic*. The following "Plot parameters" dialog box in Figure 5-33 appears. In the "Solid elements" tab, check the following check boxes: "draw solid elements", "draw element border" and "color solid element", then click "OK".

Plot parameters				2
General plot parameters:	Soil plot parameters	Solid elements:	Contour lines:	FE-Net:
Solid elements:			i <b></b>	_
🔽 Draw Solid element	s			
🔽 Draw element borde	er			
Color solid element				
Light position:				
Light position in x-dire	ction		[mm]	-10000. 🛫
Light position in y-dire	ction		[mm]	-10000. 🗧
Light position in z-dire	ction		[mm]	10000.0
<u>0</u> k	<u>S</u> ave	<u>C</u> a	ncel	<u>H</u> elp

Figure 5-33 "Plot parameters" option box

To view the wrench in 3D-view, choose "Data in isometric view" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 5-34 appears. In this option box, select "Slab thickness" option, then click "OK" button. The wrench is now displayed as solid elements in 3D-view as shown in Figure 5-35.

Data in isometric view	×
Select one item to draw	
Net numbering	○ Coordinates x/y
C Element groups	<u>k</u>
C Slab thickness	Cancel
C System of loading	
C Boundary conditions	<u>H</u> elp

Figure 5-34 "Data in isometric view" option box

### ELPLA-Tutorial

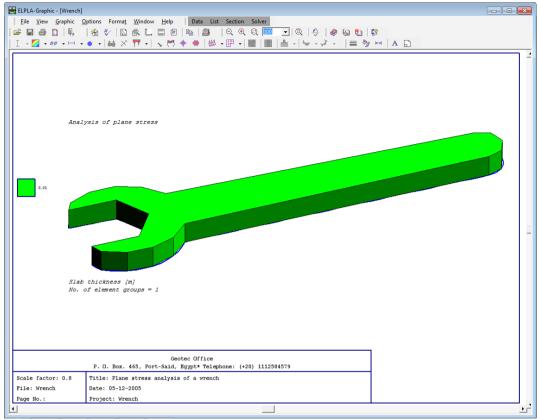


Figure 5-35 Wrench in isometric view

To view the results as contour lines

- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 5-36 appears
- In this option box, select "Maximum shear stresses max\_Sigma" as a sample for the results to be displayed
- Click "OK" button

The maximum shear stresses are now displayed as contour lines as shown in Figure 5-37.

Results as contour lines	×
Select one item to draw	
C Vertical displacements w	C Horizontal displacements u
C Support reactions Rh	C Support reactions Rv
🔿 x-Stresses Sigma_x	○ y-Stresses Sigma_y
C xy-Shear stresses Tau_xy	C Maximum shear stresses max_Tau
Maximum shear stresses max_Sigma	C Minimum stresses min_Sigma
🔿 x-Strains Epsilon_x	🔿 y-Strains Epsilon_y
○ xy-Shear strains Gamma_xy	<u>D</u> k
C Maximum shear strains max_Gamma	Cancel
C Maximum strains max_Epsilon	
C Minimum strains min_Epsilon	<u>H</u> elp

Figure 5-36 "Results as contour lines" option box

### ELPLA-Tutorial

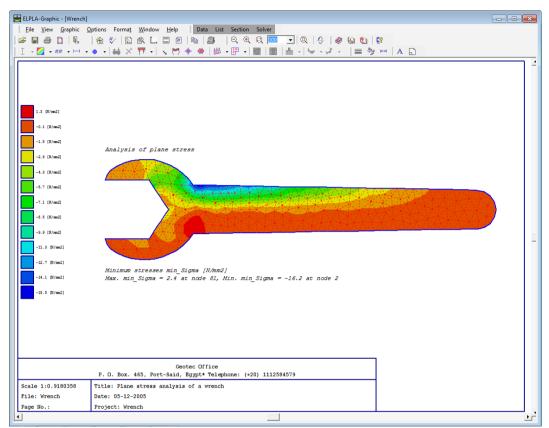


Figure 5-37 Maximum shear stresses max\_Sigma

### 5 Index

# A

Analysis	3
Analysis type	
JJF-	

# B

Boundary conditions	1	4
---------------------	---	---

# С

Calculation	
Calculation methods	
Computation of all	
Contour lines	

# F

FE-Net generation	7
Foundation properties	
1 1	

# G

Generation type	8
Graphic	

# L Loads 19, 20 N New project. 4 P Project identification. 7 S

Solver	22
Supports14,	15

# U

Unit weight of the raft 18	8
----------------------------	---

Example 6

# **Analysis of two-dimensional truss**

### ELPLA-Tutorial

# Contents

# Page

1	Desci	ription of the problem	3
	1.1	Loads and dimensions	3
	1.2	Grid material	3
	11.3	Analysis of the grid	3
2	Creat	ting the project	4
	2.1	Calculation method	4
	2.2	Project identification	6
	2.3	FE-Net data	6
	2.4	Girders	9
	2.5	Supports/ Boundary conditions	14
	2.6	Loads	19
3	Carry	ring out the calculations	23
	3.1	Starting ELPLA-Solver	23
	3.2	Carrying out all computations	23
4	View	ring data and result graphics	24
5	Index	ζ.	26

### **1** Description of the problem

Since truss structure is considered as a special case of plane frame problems, an example of a two-dimensional truss structure is selected to illustrate some features of *ELPLA* for analyzing plane frames.

### 1.1 Loads and dimensions

The truss consists of 4 equal spans. All truss members have a cross section area of  $0.01 \text{ [m}^2$ ] and moment of inertia of  $0.00001 \text{ [m}^4$ ]. Geometry of the truss and loads are shown in Figure 6-1.

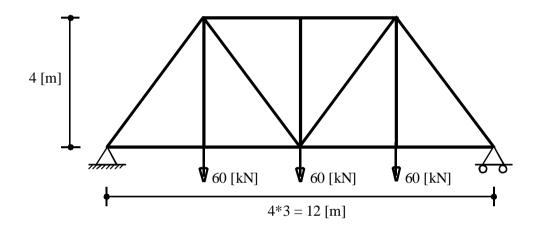


Figure 6-1 Geometry of the truss and loads

### **1.2** Truss properties

The properties of the truss are:

Young's modulus	$E_b$	$=2 * 10^8$	$[kN/m^2]$
Area of the member	A	= 0.01	[m <sup>2</sup> ]
Moment of inertia of the member	Ι	= 0.00001	[m <sup>4</sup> ]

### **1.3** Analysis of the grid

It is required to analyze the truss assuming that all members are pin connected. This Tutorial Manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for numerical calculation methods is well documented in the User's Guide of *ELPLA*.

### 2 Creating the project

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

### 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation methods" wizard appears, Figure 6-2. The wizard will guide you through the steps required to create the system data of the project. As shown in Figure 6-2, the first form of the wizard is the "Analysis type" form.

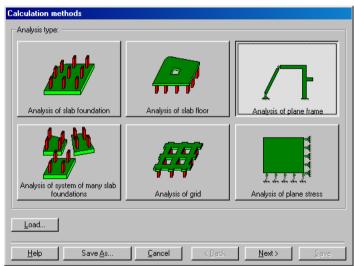


Figure 6-2 "Calculation method" wizard with "Analysis type" form

In the "Analysis type" form in Figure 6-2, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a truss problem, select "Analysis of plane frame". Then click "Next" button to go to the next form.

The next form is the "System symmetry", Figure 6-3. In this form

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

The last form of the wizard assistant contains the "Option" list, Figure 6-4. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other.

In this list

- Check "Supports/ Boundary conditions" check box
- Click "Save" button

Calculation methods	
System symmetry:	
Unsymmetrical system	
Symmetrical system about x-axis	Double-symmetrical system
Symmetrical system about y-axis	Anti-symmetrical system in x-axis
Help Save <u>A</u> s Ca	ncel < <u>B</u> ack <u>Next&gt;</u> Save

Figure 6-3 "System symmetry" form

Calculation methods	
Coptions:	
Slab with girders	
🗌 🔲 🛴 Addtional springs	
🗌 🗹 📥 Supports/Boundary conditions	
l Died raft foundation	
Determining limit depth	
Concrete design	
Nonlinear subsoil model	
Determining displacements in soil	
Determining stresses in soil	
Determining strains in soil	
Influence of neighboring foundations on the slab	
Influence of temperature change on the slab	
Influence of additional settlements on the slab	
Select All	
Help         Save As         Cancel         < Back	

Figure 6-4 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 6-5.

In this dialog box

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

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

Save As					? ×
Save in: 🔂	Examples	▼ 🗈	<u></u>	<del>d</del> *	<b></b>
Floor Grid Example					
File <u>n</u> ame:	Truss				<u>S</u> ave
Save as <u>t</u> ype:	Isolated slab foundation-files (	(*.P01)	•		Cancel

Figure 6-5 "Save as" dialog box

### 2.2 **Project identification**

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

In this dialog box

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

P	roject Ide	entification	X
	- Project Io	dentification:	
	Title	Analysis of a truss	
	Date	8.06.2006	•
	Project	Truss	
Ľ			
	<u>S</u> ave	e <u>C</u> ancel <u>H</u> elp <u>L</u> oad Save	e <u>A</u> s

Figure 6-6 "Project identification" dialog box

### 2.3 FE-Net data

To enter the truss geometry, an imaginary net must be first defined with suitable dimensions and then the truss is entered lately by connecting the corresponding nodes on the net that define the truss shape. To define the net for the truss geometry, choose "FE-Net data" command from "Data" menu. The "FE-Net generation" wizard appears as shown in Figure 6-7. This wizard will guide you through the steps required to generate the FE-Net. The first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets (Figure 6-7).

These net templates are used to generate standard nets that have constant size in both x- and y-directions.

To generate an imaginary net

- In the "Slab type" options, choose the rectangular slab option to create an imaginary net of a rectangular area
- Type 12 in the "Length of rectangular slab" edit box
- Type 4 in the "Width of rectangular slab" edit box
- Click "Next" button to go to the next form

FE-Net Generation	
Rectangular slab:	
Length of rectangular slab	L [m] 12
Width of rectangular slab	B [m] 4
	< <u>B</u> ack <u>N</u> ext > <u>F</u> inish

Figure 6-7 "FE-Net generation" wizard with "Slab type" form

After clicking "Next" in the "FE-Net generation" wizard, the following "Generation type" form appears, Figure 6-8. *ELPLA* can deal with various types of generations with triangle and/ or rectangular elements.

In this form

- Choose rectangular elements
- Click "Next"

FE-Net generation			
Generation type			
		× · · · · · · · · · · · · · · · · · · ·	× × × × × × × × × × × × × × × × × × ×
	Cancel	< Back Next >	• <u>Finish</u>

Figure 6-8 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "Grid definition" dialog box in Figure 6-9 appears with default values of constant element size.

In this "Grid definition" dialog box

- Type 12 in the "No. of grid intervals" edit box for grids in x-direction
- Type 4 in the "No. of grid intervals" edit box for grids in y-direction
- Type 1 in the "Grid interval Dx" edit box
- Type 1 in the "Grid interval *Dy*" edit box
- Click "Finish"

*ELPLA* will generate an imaginary net for a rectangular area of 12 [m] length and 4 [m] width with square elements of 1.0 [m] each side. The following embedded program in Figure 6-10 appears with the generated imaginary net.

-Net Generation Grid definition:			
Grids in x-direction:			
🔽 Constant grid interval			
No. of grid intervals	12		
Grid interval Dx [m]	1,00		
Grids in y-direction:			
🔽 Constant grid interval			
No. of grid intervals	4		
Grid interval Dy [m]	1,00		
Help	Cancel   < Back	Next >	Finish

Figure 6-9 "FE-Net generation" dialog box

### ELPLA-Tutorial

EL	PLA-	Data	n - [Tri	· [221	[FE-	Net]																		_ 8 ×
<u>F</u> ile	_								<u>I</u> n table		ptions			<u>W</u> indo			-							- ×
D					<b>a</b> (	\$∕	<u>i</u>	•		9 <del>K</del>	×	Α	• e			100 💌	0	• •	۲	6	t)	3	•	
K)	Cil	··•	- *			Ρ.			#					🔛	4	•								
																								<b>_</b>
-						ļ																		
-						<u>†</u>									+		·							
											_													
-																	4							
1		Ì.				1																		
																								-
∢  ∰βSt			Doroco			1@	Eucl		Tutori-I	<b>  F</b>	<u>ਡ</u> ੀ ਸ∠-	المنامع	Eupera	L I E	ലപം	ıng-Beis;	iale 1	<b>—</b> -		D	_			▶ I 19:45
ឮ១០	art		Persor	naiin	ansiat.	· <u> </u>	Expl	orer -	i utoriai			orials	-Examp	ie 🦉	JUebu	ing-Beisj	ыењ	12 F	LPLA	A-Dat	a		3412	<b>-</b> 19:45

Figure 6-10 Imaginary net of a rectangular area on the screen

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

- Choose "Save FE-Net" command from "File" menu in Figure 6-10 to save the data of the imaginary net
- Choose "Close FE-Net" command from "File" menu in Figure 6-10 to close "FE-Net" embedded program and to return to the main window of *ELPLA-Data*

### 2.4 Girders

To define the truss members, choose "Girders" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 6-11 appears.

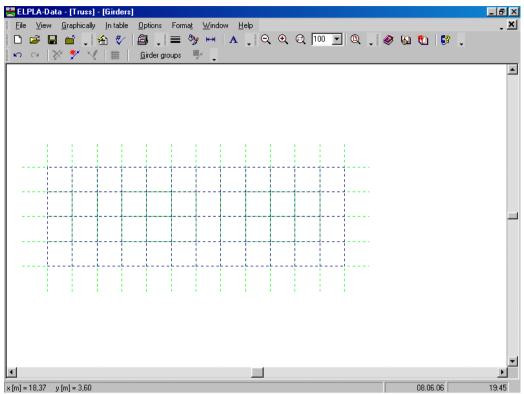


Figure 6-11 "Girders" embedded program

To enter the cross section of the girders

- Choose "Girder groups" command from "In Table" menu in Figure 6-11. The following option box in Figure 6-12 appears
- In this option box select "Rectangular cross section"
- Click "OK" button

Cross section definition	×
Cross section definition:	
C Rectangular cross section	<u>0</u> k
<ul> <li>General cross section</li> </ul>	Cancel
🔿 Create a new element group as T/L-girder	
	<u>H</u> elp

Figure 6-12 "Cross section definition" option box

After clicking "OK" button in the "Cross section definition" option box, the following list box in Figure 6-13 appears.

In this list box

- Enter the material properties of the girder, cross section dimensions and the girder weight as indicated in Figure 6-13. This is done by entering the value in the corresponding cell and press "Enter" button
- Click "OK"

irder gro	oups				X
Group No.	E-Modulus of girder E [kN/m2]	Moment of inertia I [m4]	Area A [m2]	Girder weight pb [kN/m]	<u>O</u> k <u>C</u> ancel
1	2E+08	0,000010	0,01	0,00	<u>I</u> nsert
					Сору
					Delete
					New
					<u>H</u> elp
					<u>E</u> xcel

Figure 6-13 "Defining girder groups" list box

### Defining the girder locations on the net

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

To define the girder locations on the net graphically

- Choose "Add girders" command from the "Graphically" menu in Figure 6-11. When "Add girders" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the start node of the first girder and drag the mouse until the end node of that girder (Figure 6-14). Then, click on the end node. The "Girder elements" dialog box in Figure 6-15 appears

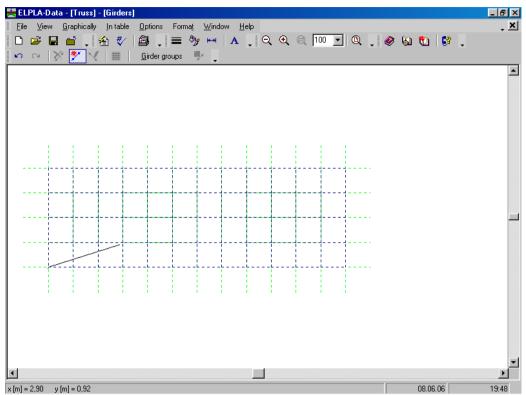


Figure 6-14 Add girder by mouse

In this dialog box

- Select the group No.
- Click "OK"

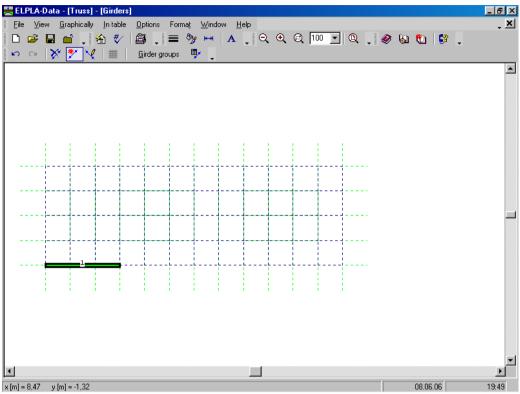


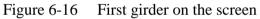
Figure 6-15 "Girder elements" dialog box

Now, the first girder is defined as shown in Figure 6-16. Note that *ELPLA* has typed automatically the girder type on it indicating the No. of girder group.

Repeat the previous steps to add the remaining girders on the net. After you have completed the definition of all girders, the screen should look like the following Figure 6-17.

### ELPLA-Tutorial





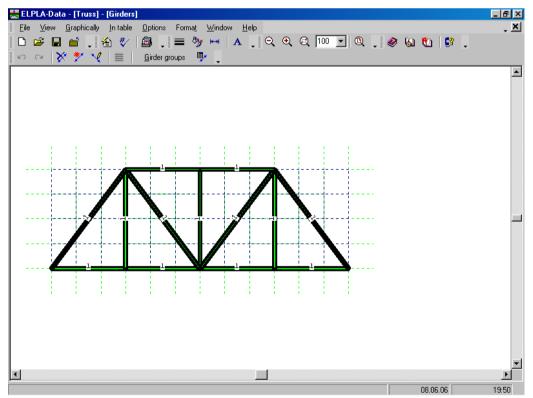


Figure 6-17 Girders on the screen

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

- Choose "Save girders" command from "File" menu in Figure 6-17 to save the data of girders
- Choose "Close girders" command from "File" menu in Figure 6-17 to close the "Girders" embedded program and to return to the main window of *ELPLA-Data*

### 2.5 Supports/ Boundary conditions

To define supports, choose "Supports/ Boundary conditions" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 6-18 appears.

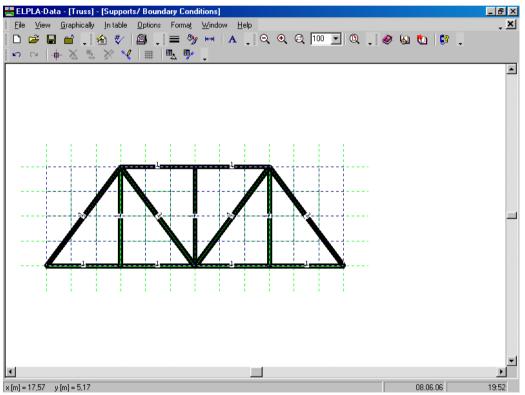


Figure 6-18 "Supports/ Boundary conditions" embedded program

*ELPLA* can display girders, supports, loads, etc. in one view together. The advantage of this option is that the user can control easily locations of supports or loads on the net when entering the rest of the data. In the case of analysis of girder problems, *ELPLA* displays girders during input of other data. As shown in Figure 6-18, girders are drawn with the actual thickness.

To view girders as simple lines

- Choose "Plot parameters" command from "Options" menu. The "Plot parameters" dialog box in Figure 6-19 appears
- In the "General plot parameters" tab, uncheck the "Draw thickness" check box in the "Girder systems" dialog box
- Click "OK" button

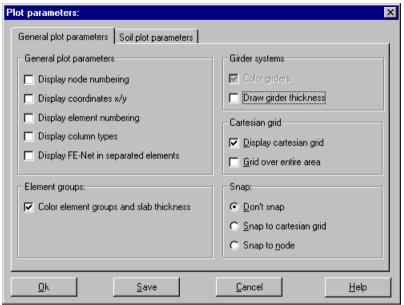


Figure 6-19 "Plot parameters" dialog box

After clicking "OK" in the "Plot parameters" dialog box, the girders are drawn as simple lines as shown in Figure 6-20.

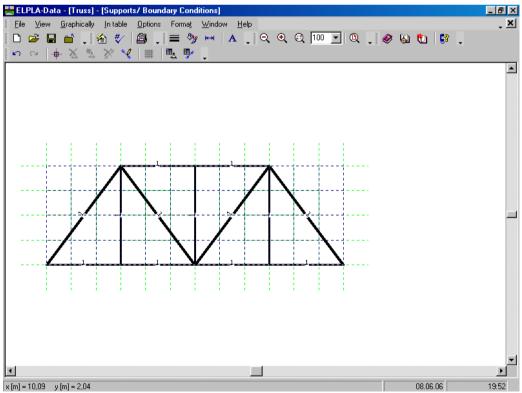


Figure 6-20 Girders are drawn in simple lines

### **Defining supports on the net**

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

To define the hinged support

- Choose "Select nodes" command from the "Graphically" menu in Figure 6-20. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nodes that have hinged supports as shown in Figure 6-21
- Choose "Add boundaries" command from "Graphically" menu in Figure 6-21. "Supports/ Boundary conditions" dialog box in Figure 6-22 appears

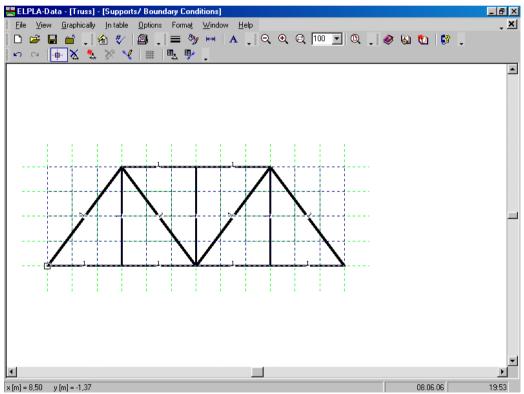


Figure 6-21 Selection of hinged support node

In this dialog box

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

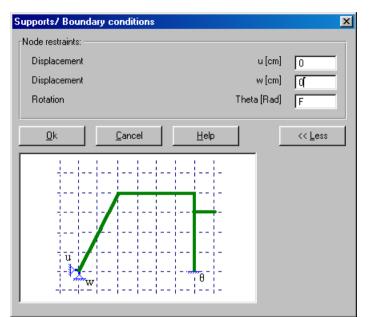


Figure 6-22 "Supports/ Boundary conditions" dialog box

To define the roller support, repeat the steps used to define the hinged support except that the displacement in *x*-direction is free. To make the displacement of the roller support is free in *x*-direction, type "F" in the "Displacement u" instead of typing "0".

After you have completed the definition of supports, the screen should look like the following Figure 6-23.

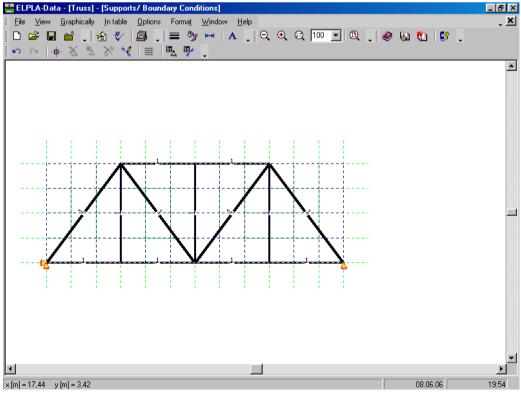


Figure 6-23 Supports on the screen

### **Defining hinges for truss members**

Assuming all truss members are pin connected, these members will be defined as hinged-hinged elements. Defining element hinges on the net may be carried out either graphically or numerically (in a table). In the current example the user will learn how to define element hinges on the net graphically.

To define the element hinged, choose "Edit hinges" command from the "Graphically" menu in Figure 6-20. When "Edit hinges" command is chosen, the cursor will change from an arrow to a cross hair. Double click on the element that has the hinges. "Edit hinges" dialog box in Figure 6-24 appears.

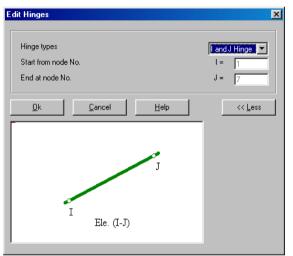


Figure 6-24 "Edit hinges" dialog box

In this dialog box, choose "I and J Hinge" from the hinge types list to define both ends of the element as hinges. Then click "OK" button.

After you have completed the definition of element hinges, the screen should look like the following Figure 6-25.

### ELPLA-Tutorial

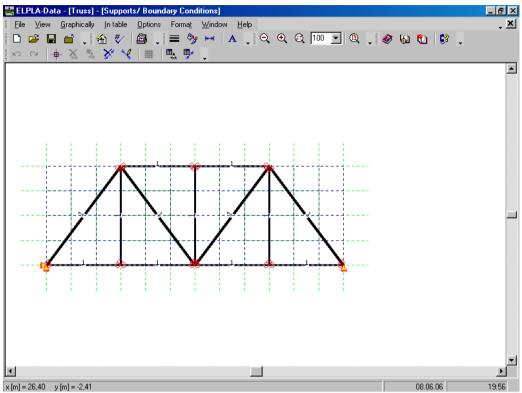


Figure 6-25 Supports and hinges on the screen

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

- Choose "Save supports/ boundary conditions" command from "File" menu in Figure 6-23 to save the data of supports
- Choose "Close supports/ boundary conditions" command from "File" menu in Figure 6-23 to close the "Supports/ Boundary conditions" embedded program and to return to the main window of *ELPLA-Data*

### 2.6 Loads

To define the concentrated load, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 6-26 appears with girders on the net.

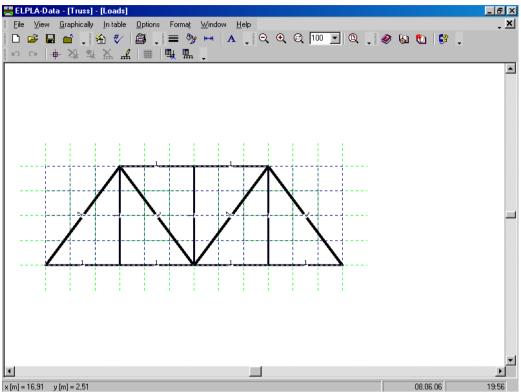


Figure 6-26 "Loads" embedded program

Although the user can define loads on both girder element and girder nodes in plane frame problem, in the truss problem we will define the loads at nodes only.

To enter the concentrated load at nodes choose "Select nodes" command from "Graphically" menu in the window of Figure 6-26. When "Select nodes" command is chosen, the cursor is changed from an arrow to a cross hair. Choose nodes to be loaded as shown in Figure 6-27. Then choose "Add nodal loads" command from "Graphically" menu, the "Nodal loads" dialog box shown in Figure 6-28 appears. In this dialog box, type 60 in the "Load Pv" edit box to define the vertical nodal load then click "OK" button.

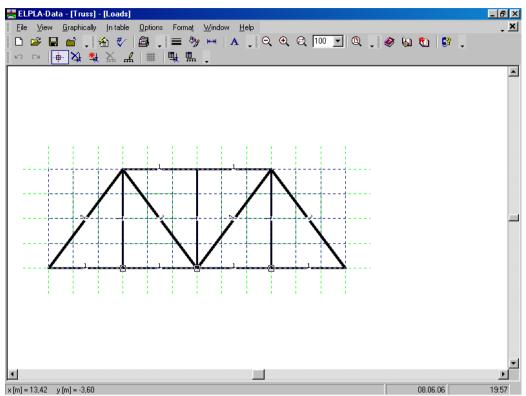


Figure 6-27 Selection of loaded nodes

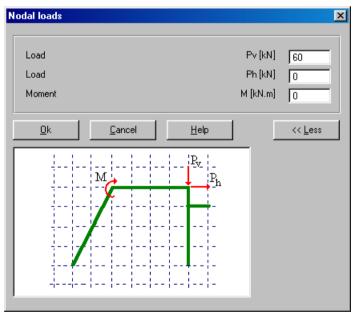


Figure 6-28 "Nodal loads" dialog box

After you have completed the definition of the nodal loads, the screen should look like the following Figure 6-29.

### ELPLA-Tutorial

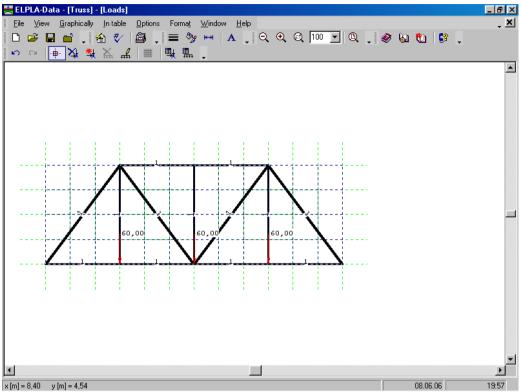


Figure 6-29 Truss load on the screen

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

- Choose "Save loads" command from "File" menu in Figure 6-29 to save the load data
- Choose "Close loads" command from "File" menu in Figure 6-29 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

Creating the project of the grid is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

### **3** Carrying out the calculations

### 3.1 Starting *ELPLA-Solver*

To analyze the problem, switch to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window in Figure 6-30 appears.

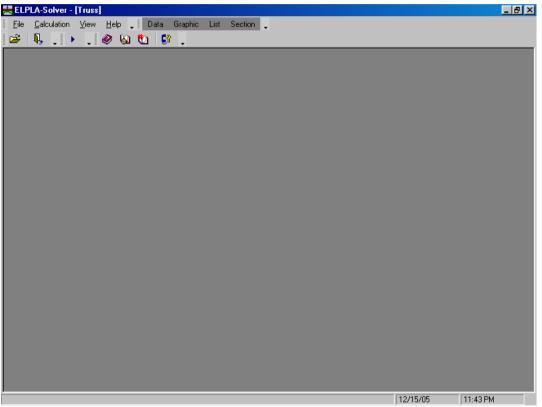


Figure 6-30 Opening screen of the sub program ELPLA-Solver

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. Also, *ELPLA* will active the "Calculation" menu. This menu contains commands of all calculations. Commands of the calculation depend on the analysis type. For the current example, the items, which are required to be calculated, are:

- Preparing the calculation
- Analysis of plane frame

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

### **3.2** Carrying out all computations

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

### Analysis progress

Analysis progress menu in Figure 6-31 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

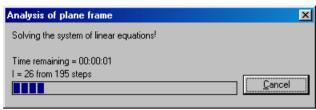


Figure 6-31 Analysis progress menu

### 4 Viewing data and result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window. *ELPLA-Graphic* window in Figure 6-32 appears. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

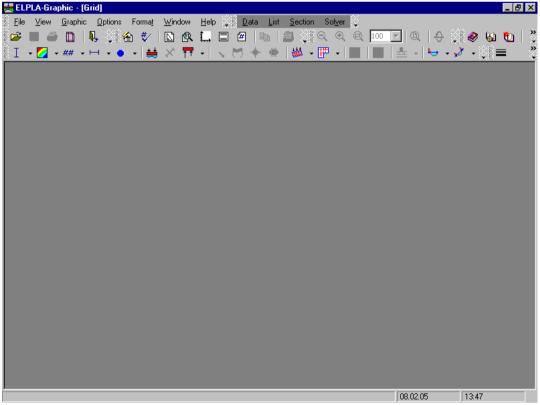


Figure 6-32 Opening screen of the sub program *ELPLA-Graphic* 

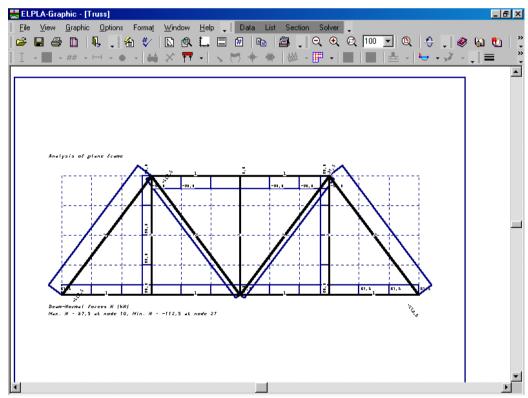
To view the results of girders, choose "Beam results" command and then "Distribution of internal forces (in plane)" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 6-33 appears.

In this option box

- Select "Beam-Normal forces *N*" as an example for the results to be displayed
- Click "OK" button

Distribution of Internal Force	s (in plane) 🛛 🔀
Select one item to draw	
Beam-Normal forces N	C Beam-Bending moments Mb
C Beam-Shear forces Qs	
C Vertical displacements w	<u> </u>
C Horizontal displacements u	Cancel
C Rotation	
C Frame deformation Delta	<u>H</u> elp

Figure 6-33 "Distribution of internal forces (in plane)" option box



The normal forces are now displayed for the truss as shown in Figure 6-34.

Figure 6-34 Truss normal forces

#### 5 Index

# A

Analysis	3
Analysis type	

# B

Boundary conditions
---------------------

# С

Calculation	
Calculation methods	
Computation of all	
Cross section	

# F

FE-Net generation	5
-------------------	---

# G

Generation type	7
Geometry	
Girder locations	
Girders	
Girders as simple lines	
Graphic	
•	

# 

Project identification6
-------------------------

# S

P

L

N

Solver	
Supports	4, 14, 16, 19

Example 7

# Analysis of plane frame

### ELPLA-Tutorial

### Contents

# Page

1	Desc	ription of the problem	3
	1.1	Loads and dimensions	3
	1.2	Frame material	3
	1.3	Analysis of the frame	3
2	Creat	ting the project	4
	2.1	Calculation method	4
	2.2	Project identification	6
	2.3	FE-Net data	7
	2.4	Girders	9
	2.5	Supports/ Boundary conditions	14
	2.6	Loads	19
3	Carry	ying out the calculations	25
	3.1	Starting <i>ELPLA-Solver</i>	25
	3.2	Carrying out all computations	26
4	View	ving data and result graphics	26
5	Gene	eral remarks about two-dimensional frame problems	28
	5.1	Introduction	28
	5.2	Coordinate systems	28
	5.3		29
	5.4	Graphical output	30
6	Index	X	31

### **1** Description of the problem

An example for analyzing a plane frame is selected to illustrate some features of *ELPLA* for analyzing frames.

### 1.1 Loads and dimensions

The frame is shown in Figure 7-1. All frame members have cross section dimensions of 0.2 [m] \* 0.5 [m]. Geometry of the frame and loads are shown in Figure 7-1.

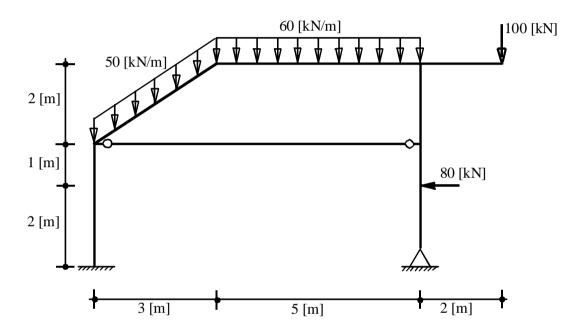


Figure 7-1 Geometry of the frame and loads

### **1.2** Frame material

Material of the frame has the following parameter:

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

### **1.3** Analysis of the frame

It is required to analyze the plane frame. This Tutorial Manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis a complete reference for numerical calculation methods are well documented in the User's Guide of *ELPLA*.

### 2 Creating the project

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

### 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation methods" wizard appears, Figure 7-2. The wizard will guide you through the steps required to create the system data of the project. As shown in Figure 7-2, the first form of the wizard is the "Analysis type" form.

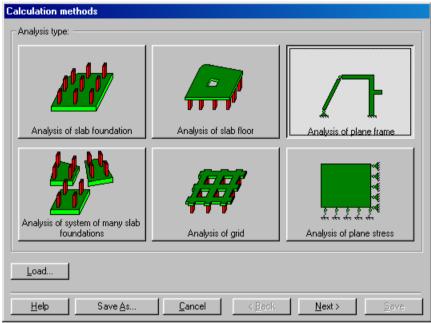


Figure 7-2 "Calculation method" wizard with "Analysis type" form

In the "Analysis type" form in Figure 7-2, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a plane frame problem, select "Analysis of plane frame". Then click "Next" button to go to the next form.

The next form is the "System symmetry" (Figure 7-3). In this form

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

The last form of the wizard assistant contains the "Option" list, Figure 7-4. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other.

In this list

- Check "Supports/ Boundary conditions" check box Click "Save" button \_
- -

Calculation methods	
Unsymmetrical system	
Symmetrical system about x-axis	Double-symmetrical system
Help Save As Cancel	
	New Years

"System symmetry" form Figure 7-3

✓ Slab with girders ↓ Additional springs	
✓ ▲ Supports/Boundary conditions	
Determining limit depth	
Concrete design	
Nonlinear subsoil model	
<ul> <li>Determining displacements in soil</li> </ul>	
Determining stresses in soil	
Determining strains in soil	
Influence of neighboring foundations on the slab	
nfluence of temperature change on the slab	
Influence of additional settlements on the slab	
V Select All	
⊠ Select <u>A</u> ll	
⊠ Select <u>A</u> ll	

After clicking "Save" button, the "Save as" dialog box appears, Figure 7-5.

In this dialog box

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

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

Save As						? ×
Save jn: 🔂	Examples	•	£		<u>r</u>	<b></b>
Wrench Example Rait1 Rait2 Truss Grid	Here Floor					
File <u>n</u> ame:	Frame					<u>S</u> ave
Save as <u>t</u> ype:	Isolated slab foundation	files (*.P01)		•		Cancel

Figure 7-5 "Save as" dialog box

### 2.2 **Project identification**

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 7-6 appears.

In this dialog box

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

P	roject ide	ntification X
Г	Project id	entification:
	Title	Analysis of a plane frame
	Date	Dienstag, 31. Januar 2006
	Project	frame
L		
[	<u>S</u> ave	Eancel Help Load Save As

Figure 7-6 "Project identification" dialog box

### 2.3 FE-Net data

To enter the frame geometry, an imaginary net must be first defined with suitable dimensions and then the frame is entered lately by connecting the corresponding nodes on the net that define the frame shape. To define the net for the frame geometry, choose "FE-Net data" command from "Data" menu. The "FE-Net generation" wizard appears as shown in Figure 7-7. This wizard will guide you through the steps required to generate the FE-Net. The first form of the wizard is the "Slab type" form, which contains a group of templates of different shapes of nets (Figure 7-7). These net templates are used to generate standard nets that have constant size in both x- and ydirections.

To generate an imaginary net

- In the "Slab type" options, choose the rectangular slab option to create an imaginary net of a rectangular area
- Type 10 in the "Length of rectangular slab" edit box
- Type 5 in the "Width of rectangular slab" edit box
- Click "Next" button to go to the next form

FE-Net generation		
Slab type		
		0
Rectangular slab:		
Length of rectangular slab	L [m]	10
Width of rectangular slab	B [m]	5
Help Cancel	< <u>B</u> ack <u>N</u> ext >	Einish

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

After clicking "Next" in the "FE-Net generation" wizard, the following "Generation type" form appears, Figure 7-8. *ELPLA* can deal with various types of generations with triangle and/ or rectangular elements.

In this form

- Choose rectangular elements
- Click "Next"

FE-Net generation			
Generation type			
		× · · · · · · · · · · · · · · · · · · ·	
	Cancel	< Back <u>N</u> ext >	> <u>Finish</u>

Figure 7-8 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "Grid definition" dialog box in Figure 7-9 appears with default values of constant element size.

In this "Grid definition" dialog box

- Type 10 in the "No. of grid intervals" edit box for grids in x-direction
- Type 5 in the "No. of grid intervals" edit box for grids in y-direction
- Type 1 in the "Grid interval Dx" edit box
- Type 1 in the "Grid interval Dy" edit box
- Click "Finish" button

*ELPLA* will generate an imaginary net for a rectangular area of 10 [m] length and 5 [m] width with square elements of 1.0 [m] each side. The following embedded program in Figure 7-10 appears with the generated imaginary net.

irid definition:		
Grids in x-direction:		
🔽 Constant grid interval		
No. of grid intervals	10 +	
Grid interval Dx [m]	1,00	
Grids in y-direction:		
🔽 Constant grid interval		
No. of grid intervals	<u>व</u>	
Grid interval Dy [m]	1,00	

Figure 7-9 "FE-Net generation" dialog box

### ELPLA-Tutorial

			- [FE-Nel													- 8
<u>F</u> ile ⊻ie	ew <u>G</u>	araphically	FE- <u>N</u> et (	Generation	In table	<u>O</u> ption	s Form	a <u>t W</u> ino	low <u>H</u> e	elp			<u> </u>			- 2
Ľ 🖻			쉽 ∜∕	۵.	= 🤊	9 <del>KN</del>	Α ͺ	⊖, e	્યા	100 💌	Q	• 🧶	61	0   0	•	
0 0I	···	* +			#				; 4	•						
																-
												-				
	1											-				
												-				
	1											-				
1																Ð
n] = 14,98	8 y[r	m] = -1,07											0	9.06.06		21:36

Figure 7-10 Imaginary net of a rectangular area on the screen

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

- Choose "Save FE-Net" command from "File" menu in Figure 7-10 to save the data of the imaginary net
- Choose "Close FE-Net" command from "File" menu in Figure 7-10 to close "FE-Net" embedded program and to return to the main window of *ELPLA-Data*

### 2.4 Girders

To define the frame members, choose "Girders" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 7-11 appears.

		- [Frame] - iraphically			Format	Window	v Help								- 8
ነ 🖻	÷ 🖬 i	∎	🖆 😻	ê .	= 🤌	9 <del>K N</del>		<b>₹</b> @	100 💌	Q	. 🤌	<b>6</b> 2 <b>(</b>	2   💱	•	
	1		1												
								 			-				
											-				
								 			-				
								 			-				
				L				 			_				
			1					1	;						
	07	n] = 3,88											.06.06		21:37

Figure 7-11 "Girders" embedded program

To enter the cross section of the girders

- Choose "Girder groups" command from "In table" menu in Figure 7-11. The following option box in Figure 7-12 appears
- In this option box, select "Rectangular cross section"
- Click "OK" button

Cross section definition	×
Cross section definition:	<u>0</u> k
Rectangular cross section	<u>C</u> ancel
C General cross section	
C Create a new element group as T/L-girder	<u>H</u> elp

Figure 7-12 "Cross section definition" option box

After clicking "OK" button in the "Cross section definition" option box, the following list box in Figure 7-13 appears.

In this list box

- Enter the material properties of the girder, cross-section dimensions and the girder weight as indicated in Figure 7-13. This is done by entering the value in the corresponding cell and press "Enter" button
- Click "OK" button

irder gro	oups				X
Group No.	E-Modulus of girder E [kN/m2]	Height of girder h [m]	Width of girder b [m]	Girder weight pb [kN/m]	<u>O</u> k <u>C</u> ancel
1	2E+07	0,50	0,20	0,00	Insert
					Сору
					Delete
					New
					<u>H</u> elp
					<u>E</u> xcel

Figure 7-13 "Defining girder groups" list box

### Defining the girder locations on the net

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

To define the girder locations on the net graphically

- Choose "Add girders" command from the "Graphically" menu in Figure 7-14. When "Add girders" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the start node of the first girder and drag the mouse until the end node of that girder (Figure 7-14). Then, click on the end node. The "Girder elements" dialog box in Figure 7-15 appears

### ELPLA-Tutorial

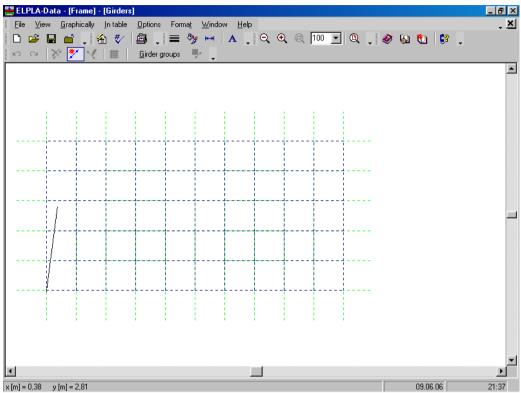


Figure 7-14 Add girder by mouse

In this dialog box

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

2
[.] 1 🔽
0 1
[·] 28

Figure 7-15 "Girder elements" dialog box

Now, the first girder is defined as shown in Figure 7-16. Note that *ELPLA* has typed automatically the girder type on it indicating the No. of girder group.

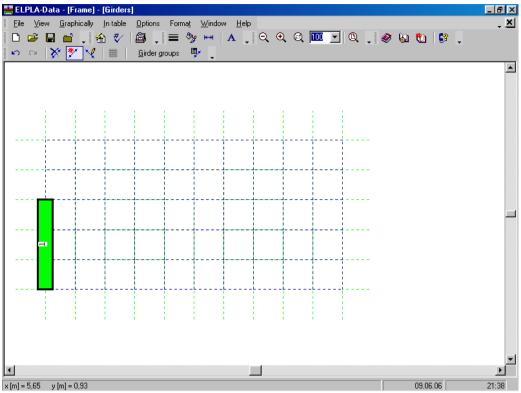


Figure 7-16 First girder on the screen

Repeat the previous steps to add the remaining girders on the net. After you have completed the definition of all girders, the screen should look like the following Figure 7-17.

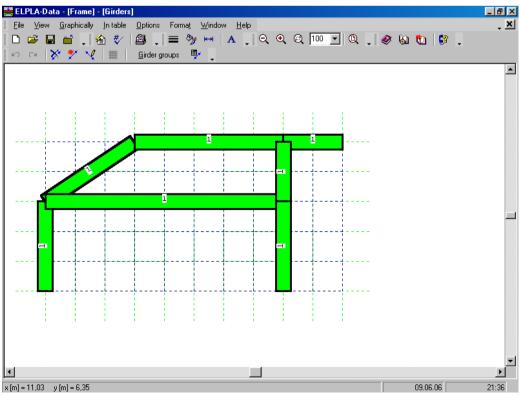


Figure 7-17 Girders on the screen

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

- Choose "Save girders" command from "File" menu in Figure 7-17 to save the data of girders
- Choose "Close girders" command from "File" menu in Figure 7-17 to close the "Girders" embedded program and to return to the main window of *ELPLA-Data*

### 2.5 Supports/ Boundary conditions

To define supports, choose "Supports/ Boundary conditions" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 7-18 appears.

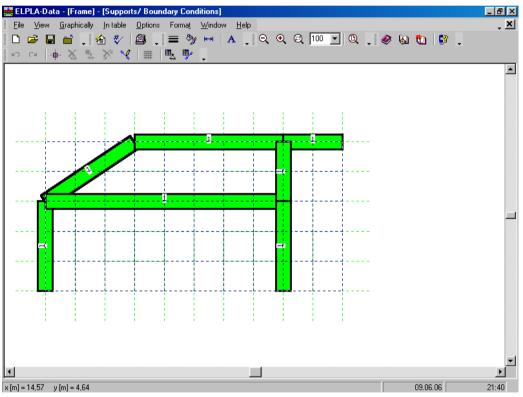


Figure 7-18 "Supports/ Boundary conditions" embedded program

*ELPLA* can display girders, supports, loads etc. in one view together. The advantage of this option is that the user can control easily locations of supports or loads on the net when entering the rest of the data. In the case of analysis of girder problems, *ELPLA* displays girders during input of other data. As shown in Figure 7-18, girders are drawn with the actual thickness.

To view girders as simple lines

- Choose "Plot parameters" command from "Options" menu. The "Plot parameters" dialog box in Figure 7-19 appears
- In the "General plot parameters" tab, uncheck the "Draw thickness" check box in the "Girder systems" dialog box
- Click "OK" button

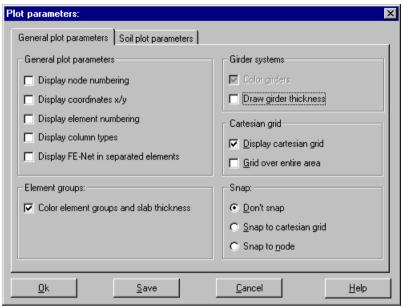


Figure 7-19 "Plot parameters" dialog box

After clicking "OK" in the "Plot parameters" dialog box, the girders are drawn as simple lines as shown in Figure 7-20.

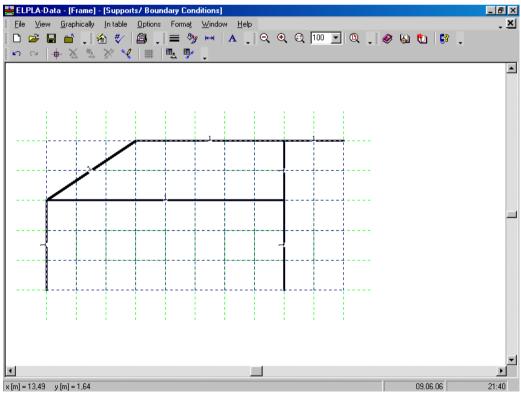


Figure 7-20 Girders are drawn as simple lines

### **Defining supports on the net**

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

To define the fixed support

- Choose "Select nodes" command from the "Graphically" menu in Figure 7-20. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nods that have the fixed supports as shown in Figure 7-21
- Choose "Add boundaries" command from "Graphically" menu in Figure 7-21. "Supports/ Boundary conditions" dialog box in Figure 7-22 appears

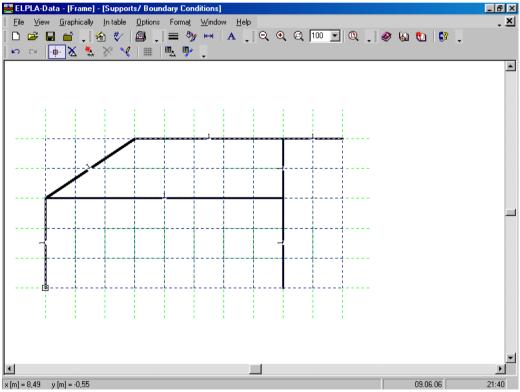


Figure 7-21 Selection of fixed support node

In this dialog box

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

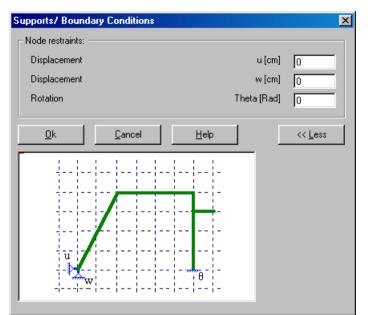


Figure 7-22 "Supports/ Boundary conditions" dialog box

To define the hinged support, repeat the steps used to define the fixed support except that the support is free to rotate. To make the rotation of the hinged support is free, type "F" in the "Rotation Theta" instead of typing "0".

After you have completed the definition of supports, the screen should look like the following Figure 7-23.

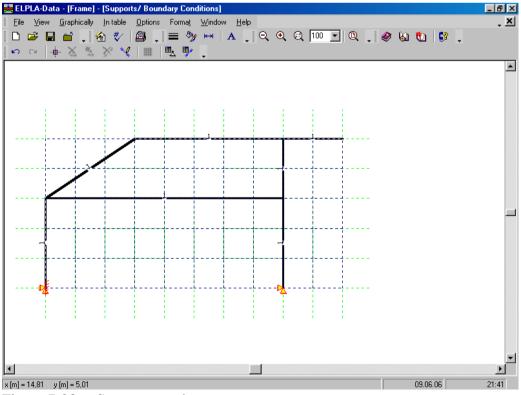


Figure 7-23 Supports on the screen

### **Defining hinges for frame members**

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

To define the element hinged, choose "Edit hinges" command from the "Graphically" menu in Figure 7-20. When "Edit hinges" command is chosen, the cursor will change from an arrow to a cross hair. Double click on the element that has hinges. "Edit hinges" dialog box in Figure 7-24 appears.

dit Hinges	X
Hinge types	and J Hinge
Start from node No. End at node No.	l = 28 J = 56
<u>O</u> k <u>C</u> ancel <u>H</u> elp	<< <u>L</u> ess
	-
J	
I Fh. (I.D.	
Ele. (I-J)	

Figure 7-24 "Edit hinges" dialog box

In this dialog box, choose "I and J Hinge" from the hinge types list to define both ends of the element as hinges. Then click "OK" button.

After you have completed the definition of element hinges, the screen should look like the following Figure 7-25.

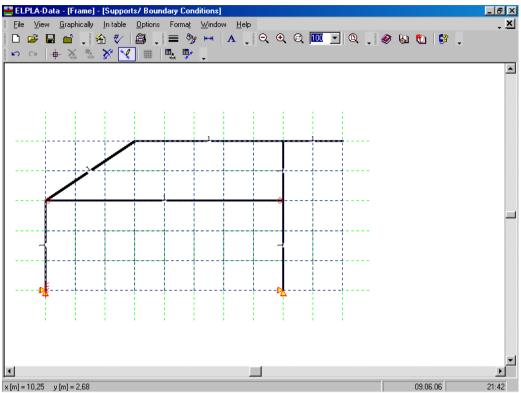


Figure 7-25 Supports and hinges on the screen

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

- Choose "Save supports/ Boundary conditions" command from "File" menu in Figure 7-23 to save the data of supports
- Choose "Close supports/ Boundary conditions" command from "File" menu in Figure 7-23 to close the "Supports/ Boundary conditions" embedded program and to return to the main window of *ELPLA-Data*

# 2.6 Loads

To define the concentrated load, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 7-26 appears with girders on the net.

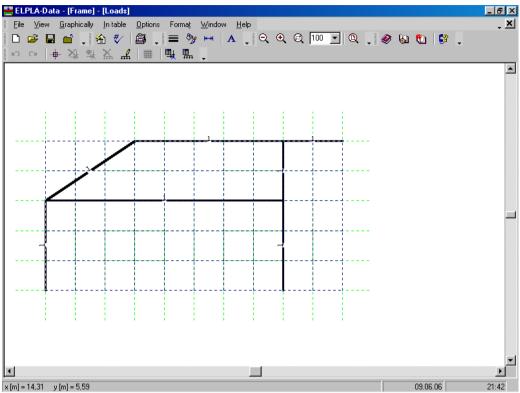


Figure 7-26 "Loads" embedded program

To enter the concentrated load at nodes, choose "Select nodes" command from "Graphically" menu in the window of Figure 7-26. When "Select nodes" command is chosen, the cursor is changed from an arrow to a cross hair. Choose a node to be loaded as shown in Figure 7-27.

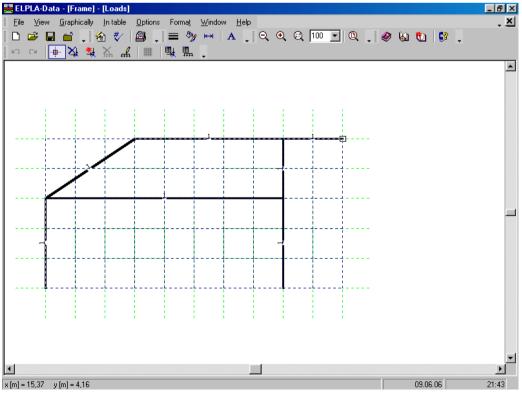


Figure 7-27 Selection of loaded node

Then choose "Add nodal loads" command from "Graphically" menu, the "Nodal loads" dialog box shown in Figure 7-28 appears. In this dialog box, type 100 in the "Load Pv" edit box to define the vertical nodal load and then click "OK" button. The node should be loaded as shown in Figure 7-29.

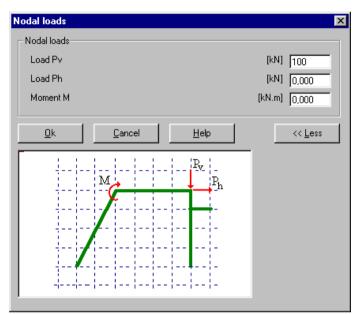
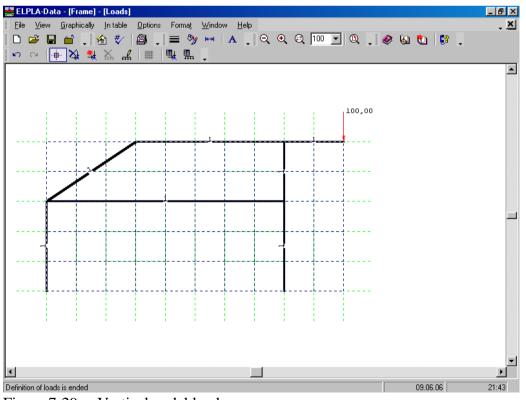
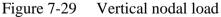


Figure 7-28 "Nodal loads" dialog box





Another node to be loaded is chosen as shown in Figure 7-30. Choose "Add nodal loads" command from "Graphically" menu, the "Nodal loads" dialog box shown in Figure 7-31 appears.

## ELPLA-Tutorial

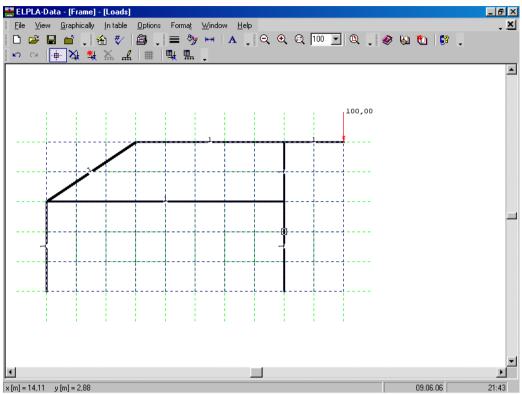


Figure 7-30 Selection of loaded node

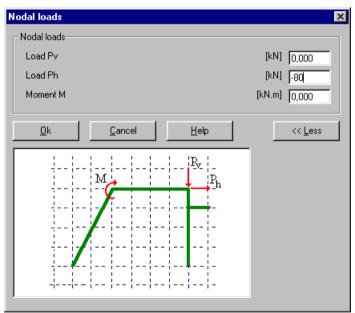


Figure 7-31 "Nodal loads" dialog box

In this dialog box, type -80 in the "Load *Ph*" edit box to define the horizontal nodal load then click "OK" button. The loaded node is shown in Figure 7-32.

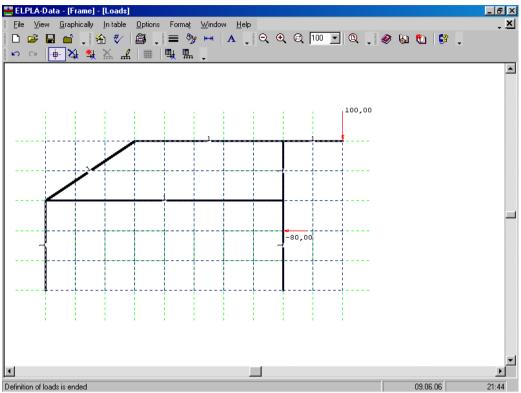


Figure 7-32 Horizontal nodal load

To input the uniform distributed load, choose "Edit member load" command from "Graphically" menu. When "Edit member load" command is chosen, the cursor will change from an arrow to a cross hair. Double click on the element that has the uniform distributed load. "Edit member load" dialog box in Figure 7-33 appears. The loaded member is shown in Figure 7-34.

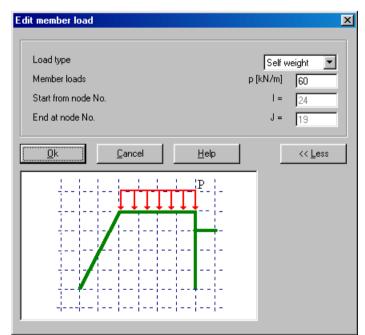


Figure 7-33 "Edit member load" dialog box

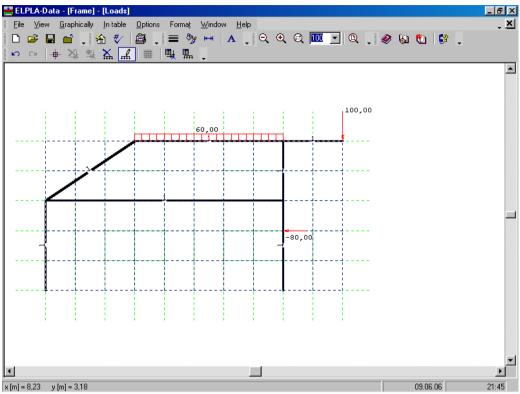


Figure 7-34 Loaded member

Use the same steps to input the uniform distributed load on the inclined member. After you have completed the definition of the load data, the screen should look like the following Figure 7-35.

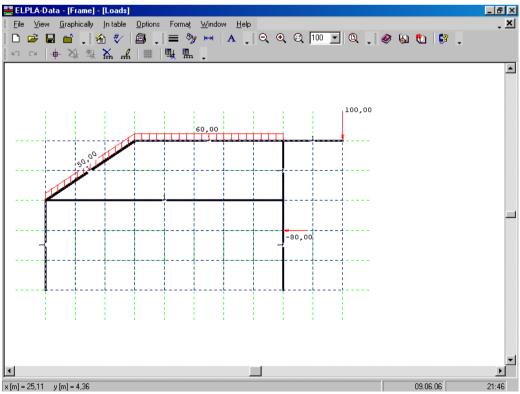


Figure 7-35 Frame load on the screen

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

- Choose "Save loads" command from "File" menu in Figure 7-35 to save the load data
- Choose "Close loads" command from "File" menu in Figure 7-35 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

Creating the project of the frame is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

# **3** Carrying out the calculations

#### 3.1 Starting *ELPLA-Solver*

To analyze the problem, switch to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window in Figure 7-36 appears.

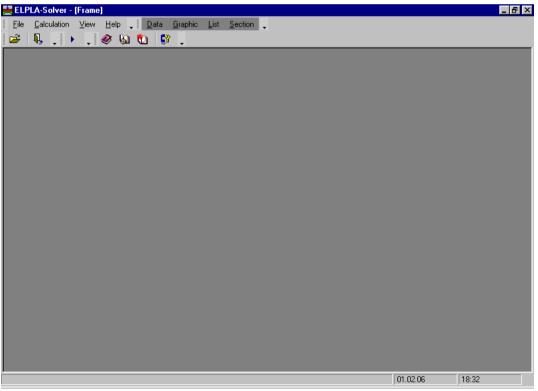


Figure 7-36 Opening screen of the sub program ELPLA-Solver

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. Also, *ELPLA* will active the "Calculation" menu. This menu contains commands of all calculations. Commands of the calculation depend on the analysis type. For the current example, the items, which are required to be calculated, are:

- Preparing the calculation
- Analysis of plane frame

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

# **3.2** Carrying out all computations

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

#### Analysis progress

Analysis progress menu in Figure 7-37 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

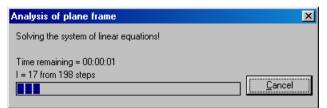


Figure 7-37 Analysis progress menu

# 4 Viewing data and result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window. *ELPLA-Graphic* window in Figure 7-38 appears. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

🗮 ELPLA-Graphic - [Frame]		_ 8 ×
File View Graphic Options Format Window Help Data List Section Solver		• (a ••   )>
┃ I - 💋 - ## - H - ● -   🚢 × 🃅 -   🔪 🧑 🔶   👹 - 🕮 -   🔤   🔳		•••
	01.02.06	:33

Figure 7-38 Opening screen of the sub program *ELPLA-Graphic* 

To view the results of girders, choose "Beam results" command and then "Distribution of internal forces (in plane)" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 7-39 appears.

In this option box

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

Distribution of Internal Forces (in plane)				
Select one item to draw				
C Beam-Normal forces N	Beam-Bending moments Mb			
C Beam-Shear forces Qs				
C Vertical displacements w	<u>0</u> k			
C Horizontal displacements u	Cancel			
C Rotation				
C Frame deformation Delta	Help			

Figure 7-39 Distribution of internal forces (in plane)" option box

The bending moments are now displayed for the frame as shown in Figure 7-40.

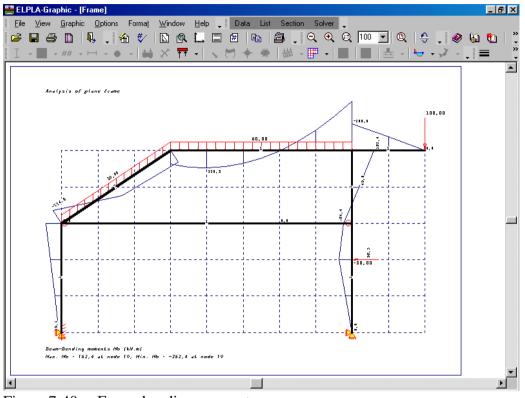


Figure 7-40 Frame bending moments

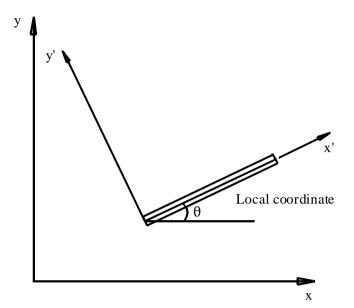
# 5 General remarks about two-dimensional frame problems

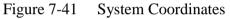
# 5.1 Introduction

This section describes directions of coordinate systems and singes of input actions such as element loads and nodal loads or sings of results such as reactions, displacements and internal forces, which are used when modeling two-dimensional frame problems by *ELPLA*.

# 5.2 Coordinate systems

There are two different coordinates for two-dimensional frame problems, global coordinate system and local coordinate system (Figure 7-41). Each of these coordinate systems is used to describe certain data such as the location of nodes or the direction of loads, displacements, internal forces and reactions. Understanding these different coordinate systems is essential to the user to being able to properly define the problem.





# 5.3 Element Loads

As shown in Figure 7-42, *ELPLA* uses a different vertical direction for defining loads. The positive value of load means that it is a downward load. Nodal loads are applied on global coordinates where element loads are applied in three different cases as follows:

#### Self weight

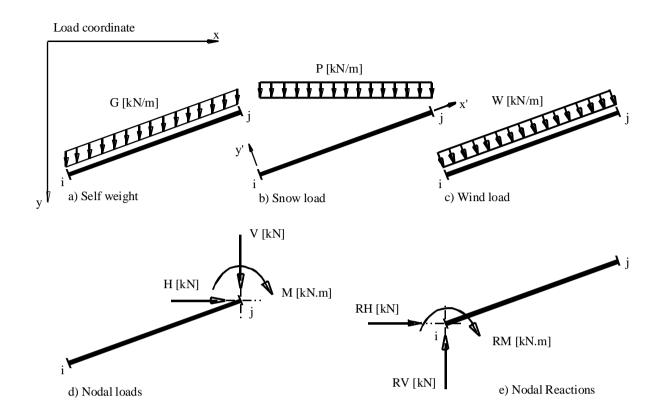
A vertical uniform load distributed along the length of the element

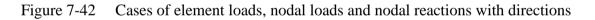
#### Snow load

A vertical uniform load distributed along the horizontal projection of the element

#### Wind load

A uniform load distributed along the length of the element with a direction perpendicular to the element (local x' axis)





# 5.4 Graphical output

Graphical outputs of results such as displacements, rotations and internal forces (bending moments, shear forces and normal forces) are drawn in locale coordinate.

#### 6 Index

# A

Analysis	
Analysis type	

# B

Boundary conditions14
-----------------------

# С

Calculation	
Calculation methods	
Computation of all	
Cross section	

# F

E-Net generation7
-------------------

# G

Generation type	7
Geometry	
Girder locations	
Girders	9, 13
Girders as simple lines	
Graphic	

# 

# P

N

L

oject identification
----------------------

# S

Save as	
Solver	
Supports	5, 14, 16, 17, 19

Example 8

Analysis of beam

# ELPLA-Tutorial

# Contents

# Page

1	Desc	escription of the problem 3	
	1.1	Loads and dimensions	3
	1.2	Beam material	3
	1.3	Analysis of the beam	3
2	Crea	ting the project	3
	2.1	Calculation method	3
	2.2	Project identification	5
	2.3	FE-Net data	6
	2.4	Girders	9
	2.5	Supports/ Boundary conditions	14
	2.6	Loads	20
3	Carry	ying out the calculations	25
	3.1	Starting <i>ELPLA-Solver</i>	25
	3.2	Carrying out all computations	26
4	View	ving data and result graphics	27
5	Index		29

## **1** Description of the problem

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

#### 1.1 Loads and dimensions

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

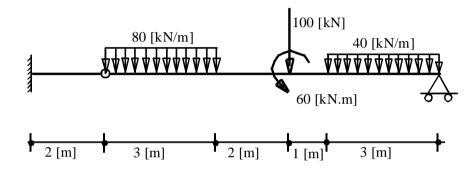


Figure 8-1 Geometry of the beam and loads

# 1.2 Beam material

Material of the beam is:

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

# **1.3** Analysis of the beam

To analyze the beam, the beam is divided in beam elements, each has a length of 1.0 [m]. This Tutorial Manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for numerical calculation methods is well documented in the User's Guide of *ELPLA*.

# 2 Creating the project

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

# 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose "New project" command from the "File" menu. The "Calculation methods" wizard appears, Figure 8-2. The wizard will guide you through the steps required to create the system data of the project. As shown in Figure 8-2, the first form of the wizard is the "Analysis type" form.

Calculation methods Analysis type:		
Analysis of slab foundation	Analysis of slab floor	Analysis of plane frame
Analysis of system of many slab foundations	Analysis of grid	Analysis of plane stress
Load		
Help Save As	Cancel< Back	<u>N</u> ext> Save

Figure 8-2 "Calculation method" wizard with "Analysis type" form

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

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

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

The last form of the wizard assistant contains the "Option" list, Figure 8-4. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other.

In this list

- Check "Supports/ Boundary conditions" check box
- Click "Save" button

Calculation methods		
System symmetry:		
Unsymmetrical system		
Symmetrical system about x-axis	Double-symmetrical system	
Symmetrical system about y-axis	Anti-symmetrical system in x-axis	
Help Save <u>A</u> s Cancel	< <u>B</u> ack <u>Next&gt;</u> Save	

Figure 8-3 "System symmetry" form

Calculation	methods
C Options:	
	b with girders
	dtional springs
	oports/Boundary conditions
/	d raft foundation
	ermining limit depth
Cor	norete design
Nor	nlinear subsoil model
Det	ermining displacements in soil
Det	ermining stresses in soil
	ermining strains in soil
	uence of neighboring foundations on the slab
	uence of temperature change on the slab
🔄 🔲 📜 Influ	uence of additional settlements on the slab
<u> </u>	
Colored	A II
Select /	2
Help	

Figure 8-4 "Options" list

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

In this dialog box

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

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

Save As					? ×
Save jn: 🔁	Examples	<b>-</b>		<u> –</u>	III 📰
Wrench Example Raft1 Raft2 Truss Grid	∰ <sup>11</sup> Floor				
File <u>n</u> ame:	Beam			9	<u>à</u> ave
Save as <u>t</u> ype:	Isolated slab foundation	n-files (*.PO1)	•	Ca	ancel

Figure 8-5 "Save as" dialog box

# 2.2 **Project identification**

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

In this dialog box

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

Project ide	entification	×
Project ide	entification:	
Title	Analysis of a plane beam	
Date	08/2/2006	<u> </u>
Project	beam	
<u>Save</u>		Save <u>A</u> s

Figure 8-6 "Project identification" dialog box

# 2.3 FE-Net data

To enter the beam geometry, an imaginary net must be first defined with suitable dimensions and then the beam is entered lately by connecting the corresponding nodes on the net that define the beam shape. To define the net for the beam geometry, choose "FE-Net data" command from "Data" menu. The "FE-Net generation" wizard appears as shown in Figure 8-7. This wizard will guide you through the steps required to generate the FE-Net. The first form of the wizard is the "Slab type" form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have constant size in both *x*- and *y*-directions.

To generate an imaginary net

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

FE-Net generation			
			0
Rectangular slab: Length of rectangular slab Width of rectangular slab		L [m] B [m]	11
(Help) Cancel	< <u>B</u> ack	<u>N</u> ext >	Einish

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

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

In this form

- Choose rectangular elements
- Click "Next"

FE-Net generation			
Generation type			
		× × × × × × × × × × × × × × × × × × ×	······································
Help	Cancel	< <u>B</u> ack <u>N</u> ext>	<u> </u>

Figure 8-8 "Generation type" form

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

In this dialog box

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

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

FE-Net Generation	
Grid definition:	
Grids in x-direction:	
🔽 Constant grid interval	
No. of grid intervals	11 =
Grid interval Dx [m]	1.00
Grids in y-direction:	
🔽 Constant grid interval	
No. of grid intervals	2
Grid interval Dy [m]	1,00
<u>H</u> elp	<u>C</u> ancel < <u>B</u> ack <u>N</u> ext > <u>F</u> inish

Figure 8-9 "FE-Net generation" dialog box

					[FE-N					0 r.			1. P. 1									
													 		elp 100 💌	l n		e fra	*	100		- <u>×</u>
⊔ ∽							<b>,</b>		<b>~</b> ≱ ⊯			•		य । 4			• •	> 61		<b>6</b> 8	•	
- /		<u>п</u>	- M	T					##				1	~	•							-
																						-
		+	† -		†	†	+-			• † •	- † -	†			†	+	-					
																	-					
					L											Ļ	_					
 	10.00		[m] = -	1 64				_		_	_								07.08	2.00		16:01

Figure 8-10 Imaginary net of a rectangular area on the screen

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

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

# 2.4 Girders

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

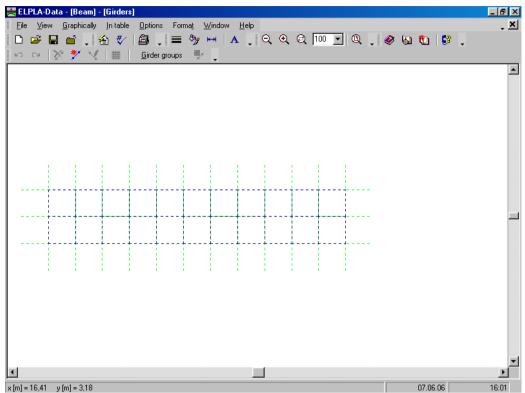


Figure 8-11 "Girders" embedded program

To enter the cross section of the girders

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

Cross section definition	×
Cross section definition:	<u>0</u> k
Rectangular cross section	Cancel
C General cross section	
C Create a new element group as T/L-girder	<u>H</u> elp

Figure 8-12 "Cross section definition" option box

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

In this list box

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

irder gro	oups					x
Group No.	E-Modulus of girder E [kN/m2]	Height of girder h [m]	Width of girder b [m]	Girder weight pb [kN/m]	k 	
1	2E+07	0,50	0,20	0,00	<u>I</u> nsert	
					Copy	
					Delete	
					New	
					<u>H</u> elp	1
					<u>E</u> xcel	

Figure 8-13 "Defining girder groups" list box

# Defining the girder locations on the net

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

To define the girder locations on the net graphically

- Choose "Add girders" command from the "Graphically" menu in Figure 8-14. When "Add girders" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the start node of the first girder and drag the mouse until the end node of that girder (Figure 8-14). Then, click on the end node. The "Girder elements" dialog box in Figure 8-15 appears

In this dialog box

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

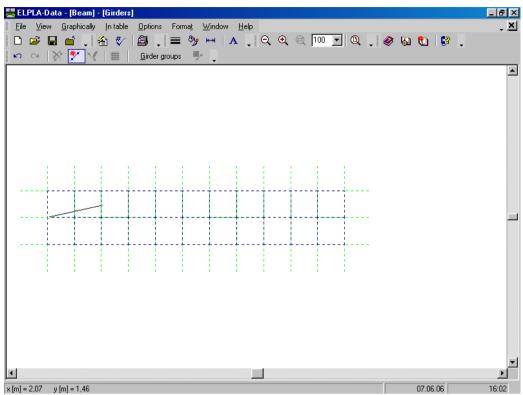


Figure 8-14 Add girder by mouse

Girder elements	×
Group No. Start from node No. End at node No.	[·] 1 [·] 26 [·] 28
kCancel	<u>H</u> elp

Figure 8-15 "Girder elements" dialog box

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

One note should be taken into consideration that girder ends are at exchanged nodes, for example supported nodes, hinged nodes and loaded nodes, so we should have five girders in this example.

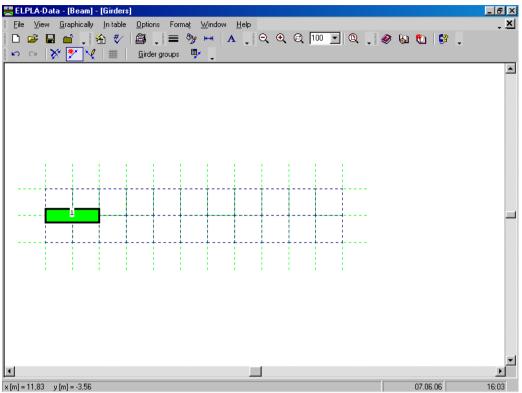


Figure 8-16 First girder on the screen

Repeat the previous steps to add the remaining girders on the net. After you have completed the definition of all girders, the screen should look like the following Figure 8-17.

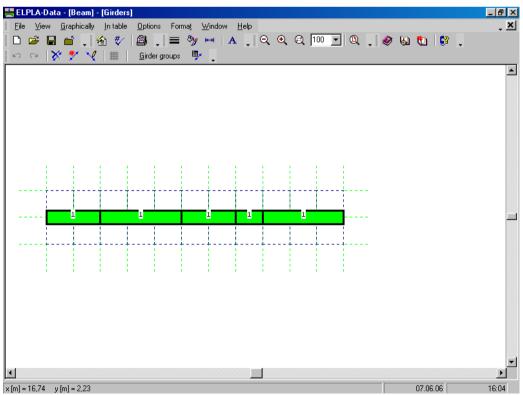


Figure 8-17 Girders on the screen

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

- Choose "Save girders" command from "File" menu in Figure 8-17 to save the data of girders
- Choose "Close girders" command from "File" menu in Figure 8-17 to close the "Girders" embedded program and to return to the main window of *ELPLA-Data*

# 2.5 Supports/ Boundary conditions

To define supports, choose "Supports/ Boundary conditions" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 8-18 appears.

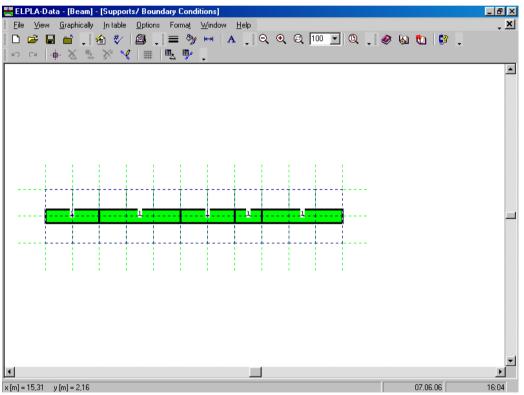


Figure 8-18 "Supports/ Boundary conditions" embedded program

*ELPLA* can display girders, supports, loads etc. in one view together. The advantage of this option is that the user can control easily locations of supports or loads on the net when entering the rest of the data. In the case of analysis of girder problems, *ELPLA* displays girders during input of other data. As shown in Figure 8-18, girders are drawn with the actual thickness.

To view girders as simple lines

- Choose "Plot parameters" command from "Options" menu.
- The "Plot parameters" dialog box in Figure 8-19 appears
- In the "General plot parameters" tab, uncheck the "Draw girder thickness" check box in the "Girder systems" dialog box
- Click "OK" button

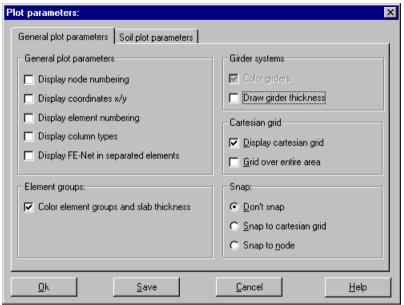


Figure 8-19 "Plot parameters" dialog box

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

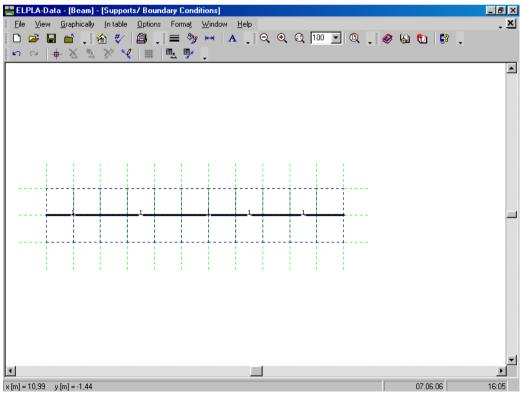


Figure 8-20 Girders are drawn as simple lines

#### **Defining supports on the net**

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

To define the fixed support

- Choose "Select nodes" command from the "Graphically" menu in Figure 8-20. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nodes that have the fixed supports as shown in Figure 8-21
- Choose "Add boundaries" command from "Graphically" menu in Figure 8-21
- "Supports/ Boundary conditions" dialog box in Figure 8-22 appears

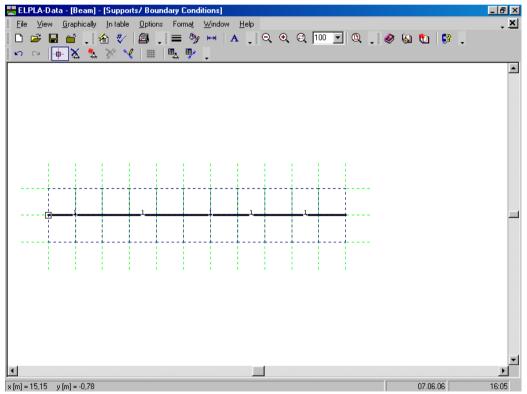


Figure 8-21 Selection of fixed support node

In this dialog box

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

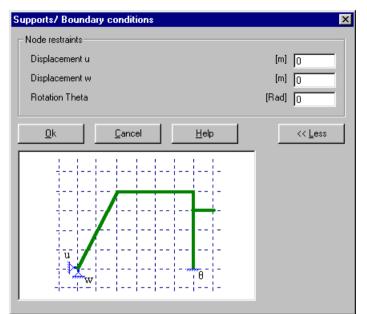


Figure 8-22 "Supports/ Boundary conditions" dialog box

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

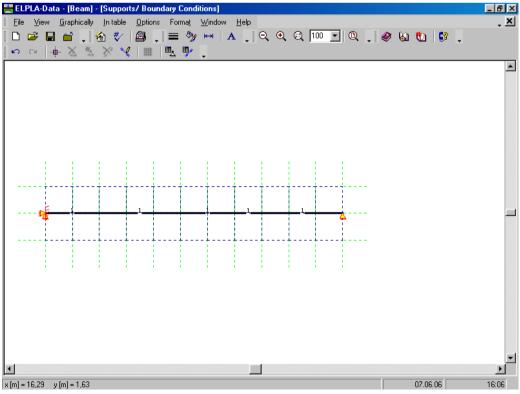


Figure 8-23 Supports on the screen

#### **Defining hinges for beam members**

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

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

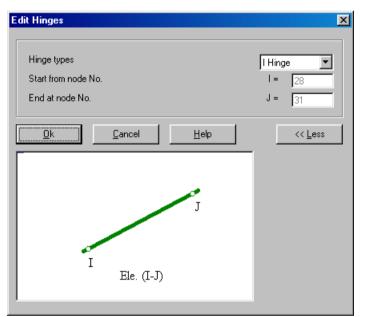


Figure 8-24 "Edit hinges" dialog box

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

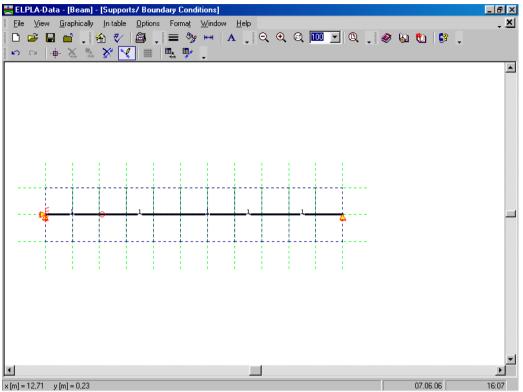


Figure 8-25 Hinged end of the element

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

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

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

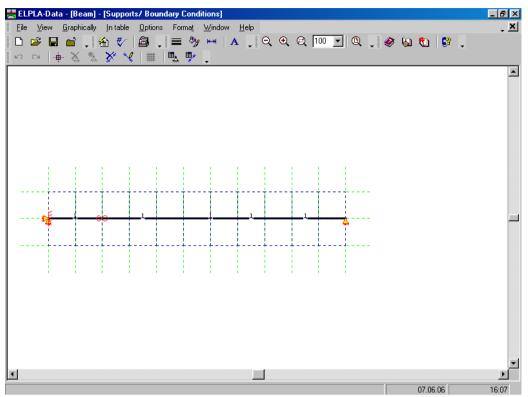


Figure 8-26 Supports and hinges on the screen

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

- Choose "Save supports/ Boundary conditions" command from "File" menu in Figure 8-23 to save the data of supports
- Choose "Close supports/ Boundary conditions" command from "File" menu in Figure 8-23 to close the "Supports/ Boundary conditions" embedded program and to return to the main window of *ELPLA-Data*

# 2.6 Loads

To define the concentrated load, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 8-27 appears with girders on the net.

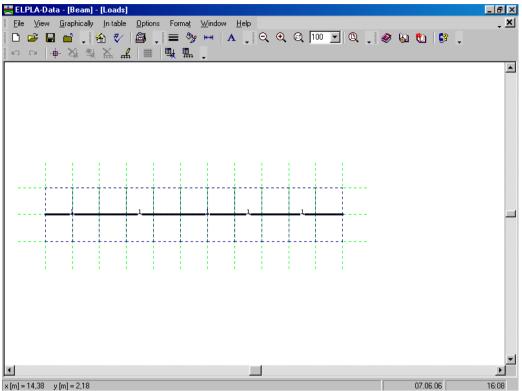


Figure 8-27 "Loads" embedded program

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

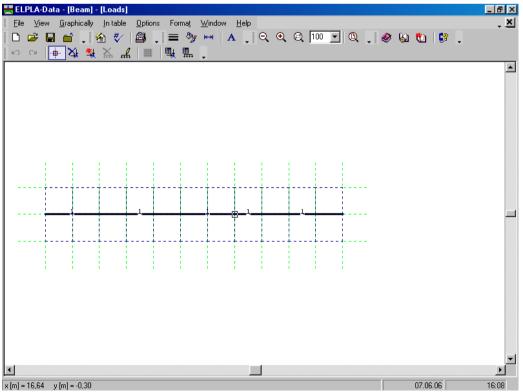


Figure 8-28 Selection of loaded node

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

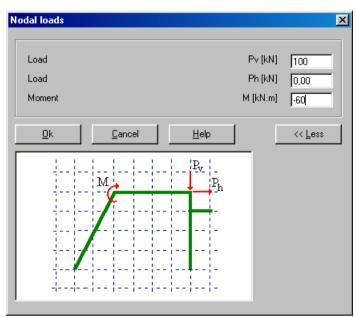


Figure 8-29 "Nodal loads" dialog box

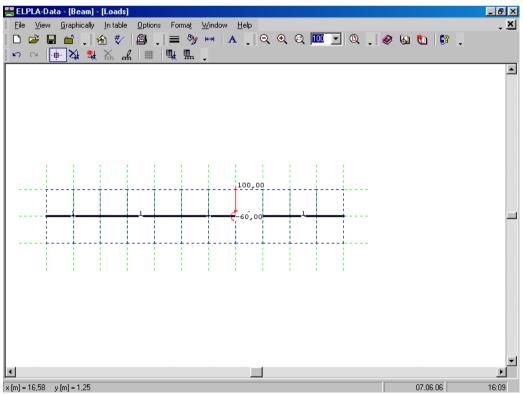


Figure 8-30 Nodal loads

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

lit member load	
Load type	Self weight 💌
Member loads	p [kN/m] 80
Start from node No.	l = 28
End at node No.	J = 31
<u>O</u> k <u>C</u> ancel <u>H</u> elp	<< <u>L</u> ess

Figure 8-31 "Edit member load" dialog box

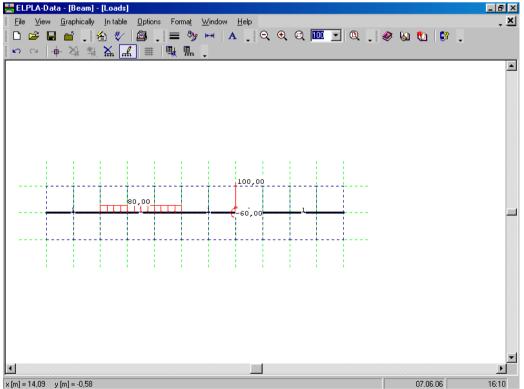


Figure 8-32 Loaded member

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

ELPLA-Data - [Beam] - [Loads]	
Eile ⊻iew Graphically Intable Options Format Window Help   D 😅 🖬 💼 , 🏠 💞   👼 ,   ☰ 🦻 ⊷   A , .   Q, Q, Q, 100 🔽   Q,	
▋D ☞ ■ ≝ , 월 ∜ 圖, ■ 物 ⊷ ▲ , Q Q Q 100 로 Q ,   ▷ ○ ▲ ☆ ☆ Ă ∡   ▦   Щ 鰛 ,	ବ ଭାଷା 🖬 🕇
100,00	
	_
x [m] = 21,30 y [m] = 5,95	07.06.06 16:10

Figure 8-33 Beam loads on the screen

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

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

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

View grouping	×
Select items to be displayed:	
✓ Supports/ Boundary conditions ✓ Girders	<u>O</u> k
	<u>C</u> ancel
	<u>H</u> elp
	🔽 Select <u>A</u> ll

Figure 8-34 View grouping dialog box

Check the "Supports/Boundary conditions" check box then click the "OK" button. The screen should look like Figure 8-35.

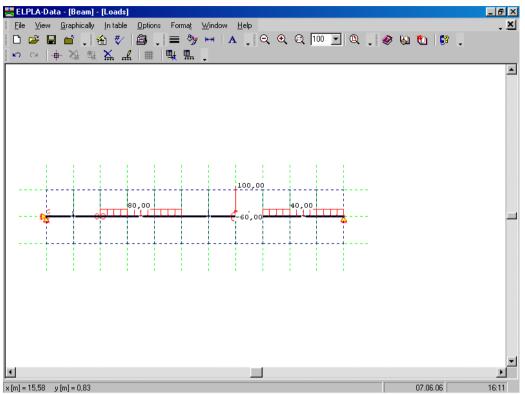


Figure 8-35 Beam supports and system of loading on the screen

Creating the project of the beam is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

#### **3** Carrying out the calculations

#### 3.1 Starting *ELPLA-Solver*

To analyze the problem, switch to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window in Figure 8-36 appears.

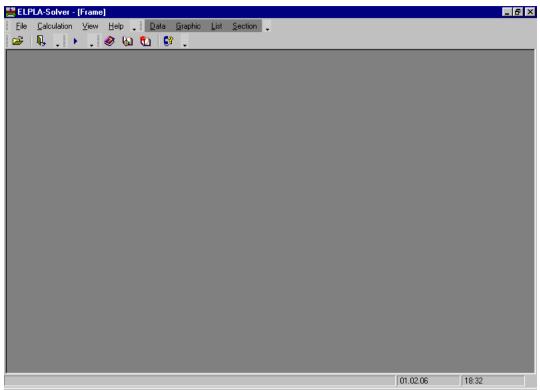


Figure 8-36 Opening screen of the sub program *ELPLA-Solver* 

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. Also, *ELPLA* will active the "Calculation" menu. This menu contains commands of all calculations. Commands of the calculation depend on the analysis type. For the current example, the items, which are required to be calculated, are:

- Preparing the calculation
- Analysis of plane beam

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

#### **3.2** Carrying out all computations

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

#### Analysis progress

Analysis progress menu in Figure 8-37 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

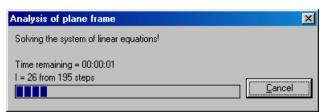


Figure 8-37 Analysis progress menu

#### 4 Viewing data and result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window. *ELPLA-Graphic* window in Figure 8-38 appears. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

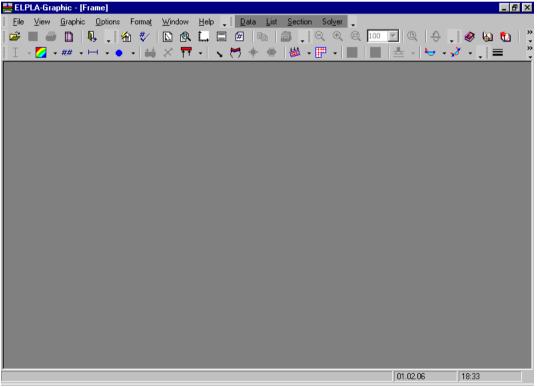


Figure 8-38 Opening screen of the sub program *ELPLA-Graphic* 

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

In this option box

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

Distribution of Internal Forces (in plane)			
Select one item to draw			
C Beam-Normal forces N	Beam-Bending moments Mb		
C Beam-Shear forces Qs			
C Vertical displacements w	<u> </u>		
C Horizontal displacements u	Cancel		
C Rotation			
C Frame deformation Delta	<u>H</u> elp		

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

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

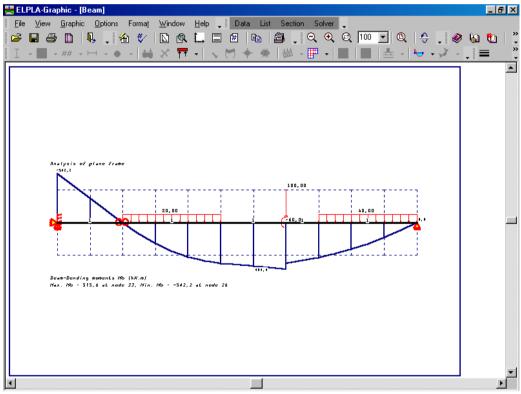


Figure 8-40 Beam bending moments

### 5 Index

## A

Analysis	
Analysis type	

### B

Boundary conditions		14,	16
---------------------	--	-----	----

### С

Calculation	
Calculation method	
Computation of all	
Cross section	

### F

FE-Net g	generation	6
----------	------------	---

### G

Generation type	7
Geometry	
Girder locations	
Girders	
Girders as simple lines	
Graphic	

Loads	
N	

### P

L

### S

Save as	5
Solver	
Supports	

Example 9

# Influence of a new neighboring building II on an old one I

### ELPLA-Tutorial

### Contents

1	Desc	cription of the problem	3
-	1.1	Loads and dimensions	3
	1.2	Foundation material and thickness	5
	1.3	Soil properties	5
	1.4	Mathematical model	5
	1.5	Analysis	5
2	Crea	ting the project of the new building II	6
	2.1	Calculation method	6
	2.2	Project identification	9
	2.3	FE-Net data	10
	2.4	Soil properties	14
	2.5	Foundation properties	18
	2.6	Loads	21
3	Crea	ting the project of the Old Building I	23
	3.1	Modifying the calculation methods options	23
	3.2	Modifying the project identification	24
	3.3	Modifying origin coordinates	24
	3.4	Neighboring foundations	26
4	Carry	ying out the calculations	26
	4.1	Starting ELPLA-Solver	26
	4.2	Carrying out all computations	28
5	View	ving data and results	29
	5.1	Viewing result graphics	29
	5.2	Plot a diagram of results	31
6	Index	X	35

#### **1** Description of the problem

An example of two neighboring buildings is selected to illustrate some of the essential features of *ELPLA* for studying the influence of a new building on an existing old one.

#### 1.1 Loads and dimensions

Figure 9-1 shows plan and section of a new building II beside a similar old one I. The building I was constructed for a long time, while the building II will be constructed close to the first one. The two buildings have the same construction geometry and loads. In addition, every building is symmetrical about both *x*- and *y*-axes.

Each foundation has a side of 10.08 [m] and thickness of 1.0 [m]. Loading on each foundation consists of 9 column loads as shown in Figure 9-1 and in Table 9-1. In spite of the two buildings are closed to each other, a small distance of 20 [cm] is assumed between them to avoid overlapping their nodes. The origin coordinates for the building I in the global system are (10.28, 0.0) while those for the building II are (0.0, 0.0).

Table 9-1	Point loads P		
Load No.	Load value	X-coord.	Y-coord.
I	P	Х	У
[-]	[kN]	[m]	[m]
1	500	0.72	0.72
2	500	0.72	9.36
3	500	9.36	9.36
4	500	9.36	0.72
5	1000	5.04	9.36
6	1000	5.04	0.72
7	1000	9.36	5.04
8	1000	0.72	5.04
9	2000	5.04	5.04

#### ELPLA-Tutorial



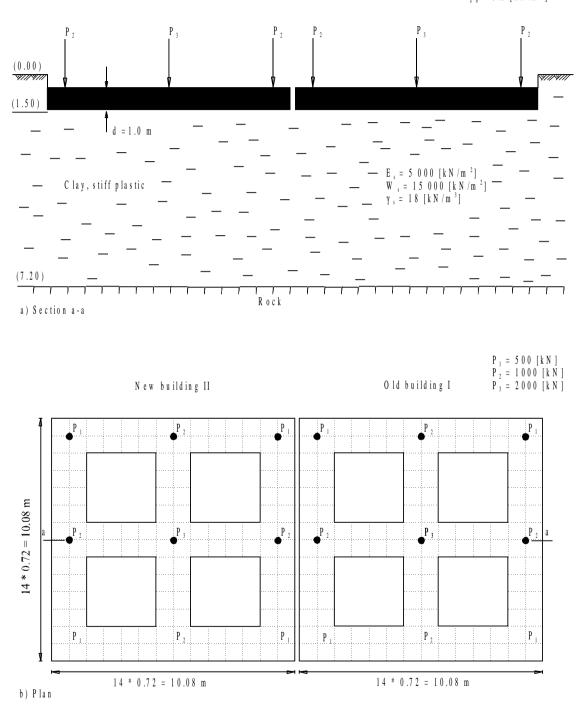


Figure 9-1 Action of new building II on the old building I

#### **1.2** Foundation material and thickness

The foundation material and thickness are supposed to have the following parameters:

Foundation thickness	d	= 1.0	[m]
Young's modulus	$E_b$	$=2 * 10^{7}$	$[kN/m^2]$
Unit weight	$\gamma_b$	= 0.0	$[kN/m^3]$
Poisson's ratio	$v_b$	= 0.15	[-]

Unit weight of the foundation material is assumed to be zero to neglect its own weight in the analysis.

#### **1.3** Soil properties

The subsoil under the buildings consists of a layer of stiff plastic clay with 5.70 [m] thickness, overlying a rigid base (Figure 9-1a). The soil is supposed to have the following parameters:

Modulus of compressibility for loading	$E_s$	= 5000	$[kN/m^2]$
Modulus of compressibility for reloading	$W_s$	= 15000	$[kN/m^2]$
Unit weight	$\gamma_s$	= 18	[kN/m <sup>3</sup> ]

#### **1.4** Mathematical model

The influence of surrounding structures and external loads can be taken into consideration only for the Continuum model (methods 4 to 9). The Continuum model based on, the settlement at any node is affected by the forces at all the other nodes. In this example, the Modulus of compressibility method (method 7) is chosen to analyze both of the two buildings.

This tutorial manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for the soil models and numerical calculation methods are well documented in the User's Guide of *ELPLA*.

#### 1.5 Analysis

To analyze foundations, each foundation is subdivided into elements with 132 nodes as shown in Figure 9-1b. Two independent names define the data of the two buildings are chosen. The data are quite similar for the two buildings except the origin coordinates, which are chosen to be  $(x_0, y_0) = (10.28, 0.0)$  and (0.0, 0.0) for buildings I and II, respectively. In spite of the two buildings are closed to each other, a small distance of 20 [cm] is assumed between them to avoid overlapping their nodes.

The analysis of the new building II is carried out first to obtain the contact pressures under it. Due to these contact pressures, settlements will occur not only under the building II but also outside under the building I. Then, under the assumption that left beside the old building a new building will be constructed, the contact pressures, settlements and internal forces of the old building are determined.

#### 2 Creating the project of the new building II

In this section, the user will learn how to create a project for analyzing system of two foundations. Thus is done by first entering the data of the two foundations individually in the same manner of the previous foundation example.

#### 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation method" wizard appears, Figure 9-2. This wizard will guide you through the steps required to create the project. As shown in Figure 9-2, the first form of the wizard is the "Analysis type" form. In this form, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a foundation problem, select "Analysis of slab foundation" then click "Next" button to go to the next page.

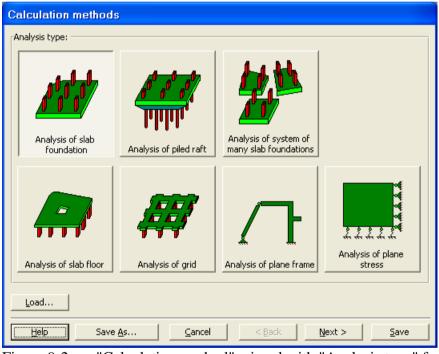


Figure 9-2 "Calculation method" wizard with "Analysis type" form

After clicking "Next" button, the "Calculation methods" form appears, Figure 9-3.

To define the calculation method

- Select the calculation method "7-Modulus of Compressibility (Elimination)"
- Click "Next" button to go to the next form

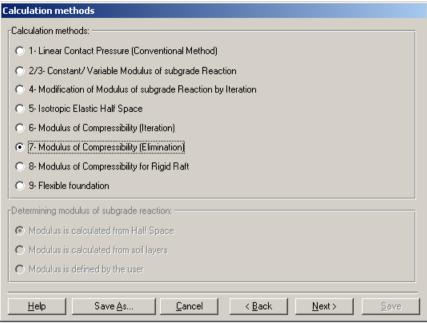


Figure 9-3 "Calculation methods" form

The next form is the "System symmetry" (Figure 9-4). In this form choose "Unsymmetrical system" and then click "Next" button.

Calculation methods	
System symmetry:	
Unsymmetrical system	
Symmetrical system about x-axis	Double-symmetrical system
Symmetrical system about y-axis	Anti-symmetrical system about x-axis
Help Save As Cancel	< Back Next> Save

Figure 9-4 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 9-5. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other. There isn't any requested choices, so click the "Save" button.

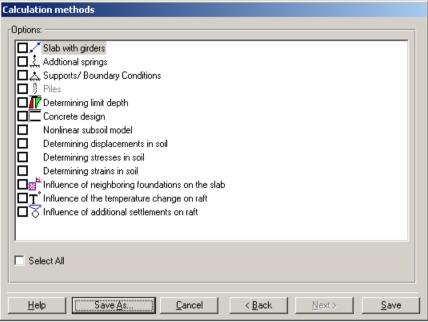


Figure 9-5 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 9-6.

In this dialog box

- Type a file name for the current project in the "File name" edit box. For example type "New Building II". *ELPLA* will use automatically this file name in all reading and writing processes
- Click "Save" button

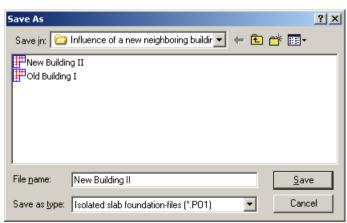


Figure 9-6 "Save as" dialog box

*ELPLA* will activate the "Data" menu. Also the file name of the current project [New Building II] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 9-7.

In the "Data" menu, the user can enter the remaining data of the project using the same sequence of commands in this menu. The first command in the menu is "Calculation methods", which has been already entered. Therefore, *ELPLA* has put the sign " $\sqrt{}$ " beside this command, Figure 9-7.

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

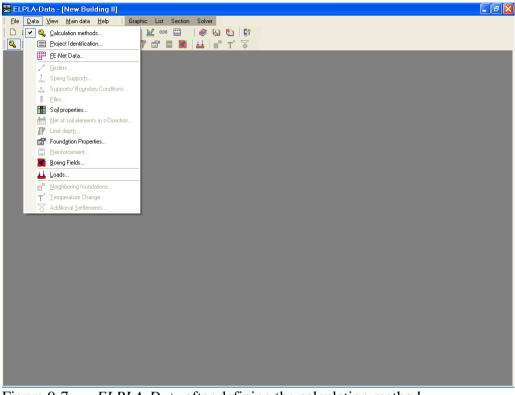


Figure 9-7 ELPLA-Data after defining the calculation method

#### 2.2 **Project identification**

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 9-8 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Influence of a new neighboring building II on an old one I"
- Type the date of the project in the "Date" edit box
- Type "New Building II" in the "Project" edit box
- Click "Save" button

Р	roject Ide	ntification X			
	Project Ide	ntification:			
	Title	Influence of a new neighboring building II on an old one I			
	Date	17.07.1998			
	Project	New Building II			
	<u>S</u> ave	<u>Cancel</u> <u>H</u> elp <u>L</u> oadSave <u>A</u> s			

Figure 9-8 "Project identification" dialog box

#### 2.3 FE-Net data

Choose "FE-Net data" command from "Data" menu of *ELPLA-Data*. The "FE-Net generation" wizard appears as shown in Figure 9-9. This wizard will guide you through the steps required to generate the FE-Net. The first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have regular shapes. For the given problem, the foundation has a square shape.

To generate the FE-Net

- In the "Slab type" options, choose the square slab option
- Type 10.08 in the "Length/ width of square slab" edit box
- Click "Next" button to go to the next form

FE-Net Generation
Slab type:
Square slab: Length/ width of square slab L/ B [m] 10.08
Help     Cancel     Back     Next >

Figure 9-9 "Slab type" form

After clicking "Next" in "Slab type" form, the "Generation type" form appears, Figure 9-10. *ELPLA* can deal with various type of generations with rectangular elements. Choose the rectangular elements option as generation type. Then click "Next" button to go to the next form.

FE-Net Generation			
Generation type:		× × × × × × × × × × × × × × × × × × ×	· · · · · · · · · · · · · · · · · · ·
Help	<u>C</u> ancel	< <u>B</u> ack <u>N</u> ext >	Einish

Figure 9-10 "Generation type" form

The next form of the "FE-Net generation" wizard is the "Grid definition" dialogue box. In this dialogue box, the default values of constant element size appear, Figure 9-11.

In this "Grid definition" dialog box

- Type 14 in the "No. of grid intervals" edit box for grids in x-direction
- Type 14 in the "No. of grid intervals" edit box for grids in y-direction
- Type 0.72 in the "Grid interval  $D_x$  [m] " edit box for grids in x-direction
- Type 0.72 in the "Grid interval  $D_y$  [m] " edit box for grids in y-direction
- Click "Finish" button

FE-Net Generation	
Grid definition:	
Grids in x-direction:	
Constant grid interval	
No. of grid intervals	14 -
Grid interval Dx [m]	0.72
Grids in y-direction:	
Constant grid interval	
No. of grid intervals	14
Grid interval Dy [m]	0.72
Help	Cancel < <u>B</u> ack <u>N</u> ext> <u>F</u> inish

Figure 9-11 "Fe-Net generation" form

Click "Finish" button in the "Generation parameters" form. *ELPLA* will generate a suitable FE-Net for the square foundation of 10.08 [m] length with rectangular elements, which have equal areas. The following embedded program in Figure 9-12 appears with the generated net.

#### ELPLA-Tutorial

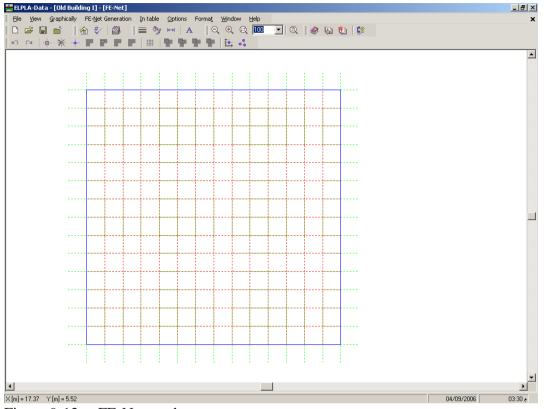


Figure 9-12 FE-Net on the screen

#### **Deleting nodes from the FE-Net**

To select the unnecessary nodes, that are required to be removed from the net, first choose "Select nodes" command from the "Graphically" menu in Figure 9-12. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair. The command "Remove nodes" in the menu "Graphically" will be enabled, indicating the mode in which is being operated. Next, select the required nodes by clicking on each node individually or selecting a group of nodes as shown in Figure 9-13. A group of nodes can be selected by holding the left mouse button down at the corner of the region. Then, dragging the mouse until a rectangle encompasses the required group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

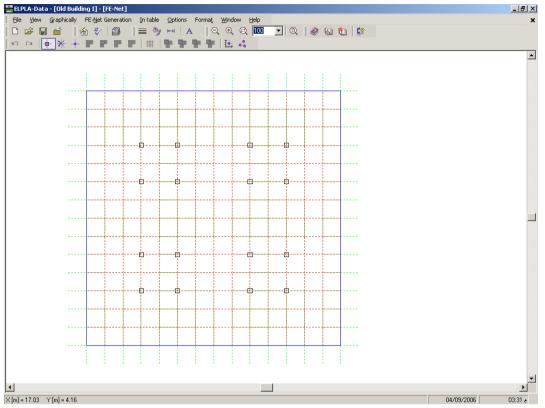


Figure 9-13 Generated FE-Net after selecting the unnecessary nodes

To remove the selected nodes, choose "Remove nodes" command from "Graphically" menu. The action of this command is indicated in Figure 9-14. To leave the graphic mode, press "Esc" key.

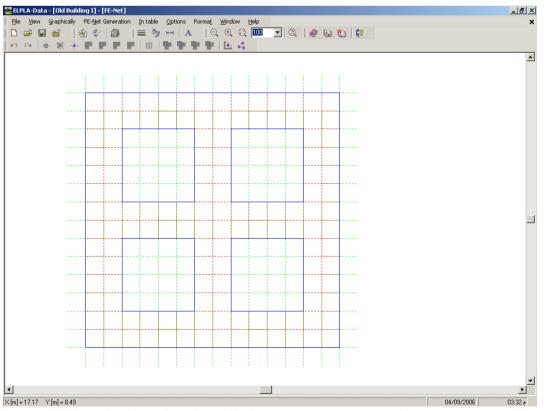


Figure 9-14 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save FE-Net" command from "File" menu in Figure 9-14 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 9-14 to close "FE-Net" embedded program and to return to the main window of *ELPLA-Data*

#### 2.4 Soil properties

To define the soil properties, choose "Soil properties" command from "Data" menu of *ELPLA-Data*. The following sub program in Figure 9-15 appears with a default-boring log.

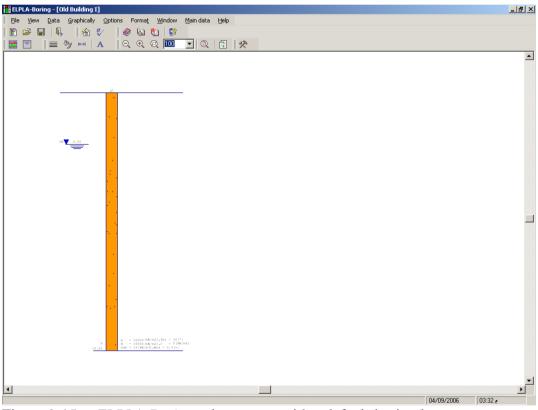


Figure 9-15 *ELPLA-Boring* sub program with a default-boring log

#### Modifying data of boring log graphically

In *ELPLA*, the boring log can be defined or modified graphically, which makes the definition of the boring log very easy. By double-clicking the left mouse button on a specified screen position, the user can define or modify the soil data and input parameters graphically.

To enter the geotechnical data of the soil layer

- Double-click on the geotechnical data of the soil layer. The corresponding dialog box appears allowing you to modify the geotechnical data of the soil layer, Figure 9-16
  In the dialog group box "Geotechnical data of the layer" in Figure 9-16, define the
- geotechnical data of the soil layer as follows:

 $\begin{array}{ll} E_s &= 50000 & [kN/m^2] \\ W_s &= 15000 & [kN/m^2] \\ \gamma_s &= 18 & [kN/m^3] \end{array}$ 

The unit weight of the soil is used to determine the overburden pressure  $q_v$  [kN/m<sup>2</sup>] due to the removed soil, which is equal to  $\gamma_s * d_f$ . In the current example  $d_f = 1.5$ . The angle of internal friction  $\varphi$  and the cohesion *c* of the soil are not required because the selected type of the analysis is linear analysis. Therefore, the user can let the default values of the internal friction and the cohesion. Next, click "OK" button.

Soil data				
Layer No.	o. 1 from 1 boring logs: 1 from 1 layers:			
Soil pro	perties are defined by Mod	ulus of Comp	ressibility Es (1/mv)	-
Es	[kN/m2] 5000	Fhi	[°] 0	_
Ws	[kN/m2] 15000	с	[kN/m2] 0	
Gam	[kN/m3] 18			
<u>k</u>	Cancel			

Figure 9-16 "Geotechnical data of the soil layer" dialog box

To define the soil type and color for the layer

- Double-click on the soil symbol of the soil layer. The corresponding dialog box appears allowing you to modify the soil symbols of that layer, Figure 9-17
- Select "T, Clay" as the soil type in the "Main soil type 1" combo box in the dialog group box "Soil and rock symbols" in Figure 9-17. The color of the clay and a short text "T" according to the German specification code DIN 4023 will be automatically created
- Click "OK" button

Soil Data Boring log No. 1 from Layer No. 1 from 1 k Soil and rock symb	ayers:	×
Main soil type 1	T, Clay	
Main soil type 2	-, No symbole 💌	
submain soil 1	-, No symbole	
submain soil 2	-, No symbole 💌	
Color	-, No symbole 💌	
Short text	Т	
	1	
<u>k</u>	<u>C</u> ancel	

Figure 9-17 "Soil and rock symbols" dialog box

To modify the groundwater depth under the ground surface

- Double-click on the groundwater level. The corresponding edit box appears allowing you to modify the groundwater depth under the ground surface, Figure 9-18. To neglect the uplift pressure on the foundation, groundwater level is chosen at anywhere under the foundation basement
- Type 7.20 in the "Groundwater depth under the ground surface" edit box
- Click "OK" button

Groundwater		X
Groundwater:	h under the ground surface [m] $\left  \vec{p}, 20 \right $	
<u>O</u> k	Cancel	

Figure 9-18 "Groundwater depth under the ground surface" edit box

To modify a layer depth

- Double-click on the layer depth. The corresponding edit box appears allowing you to modify the layer depth under the ground surface, Figure 9-19
- Type 7.20 in the "Layer depth under the ground surface" edit box
- Click "OK" button

Soil Data	×
Boring log No. 1 from 1 boring logs: Layer No. 1 from 1 layers: Layer depth under the ground surface [m] 7.20	

Figure 9-19 "Layer depth under the ground surface" edit box

To modify the label of a boring log

- Double-click on the label of the boring log. The corresponding edit box appears allowing you to modify the label of the boring log, Figure 9-20
- Type "BPN1" in the edit box of Figure 9-20
- Press "Enter" key to consider the text

BPN1

Figure 9-20 "Label of the boring log" edit box

To enter the main soil data for the layer

- Choose "Main soil data" from "Data" menu in Figure 9-15. The following dialog group box in Figure 9-21 appears
- In this dialog group box, enter the settlement reduction factor  $\alpha$  and the groundwater depth under the ground surface  $G_w$  [m] as indicated in Figure 9-21. Note that the groundwater depth under the ground surface was typed in the corresponding edit box because this value has been already defined graphically
- Click "OK" button in the "Main soil data" dialog group box in Figure 9-21

Bearing capacity fact	ors	
Alfa	[-]	1
Gw	[m]	7.20
Help		
	Alfa Gw	Gw [m]

Figure 9-21 "Main soil data" dialog group box

After the user has completed the definition of all soil properties and parameters, the screen should look like the following Figure 9-22.

ELPLA-Boring - [New Building II] ile: View Data Graphically Options: Format Window Main d ] 22 届 1 見   名 参   梁 ⑤ ① 1 83	a <u>H</u> elp	
0.745		
토립		
$ \begin{array}{c} \pi \\ \pi $		
—		
		<u>•</u>

Figure 9-22 Boring log on the screen

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

- Choose "Save boring logs" command from "File" menu in Figure 9-22 to save the data of boring log
- Choose "Close boring logs" command from "File" menu in Figure 9-22 to close the *ELPLA-Boring* sub program and to return to the main window of *ELPLA-Data*

#### 2.5 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 9-23 appears with default foundation properties. The data of foundation properties for the current example, which are required to define, are foundation material, foundation thickness and foundation level. Any other data corresponding to foundation properties in the program menus are not required. Therefore, the user can take these data from the default foundation properties.

-Data - [Ok View Graph	ically						Options	; For	mat '	Window	/ Heli	)							
												_	0	1 🔊	ίλη i	to 🕼			
									-, -					~	00				
	1	1		11117	. 1 .														
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		1	1					1	1					1	1				
						ļ													
		1	1					1	1					1	1				
		1	1					1	1					1	1				
		1	1					1	1					1	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		1	1	-				4	1					4	1				
		1	±.					1	1					1					
		1	1					1	1					1	1				
		1	1					1	1					1	1				
			<u></u>																
		1	1					1	1					1	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		-	-	-	-	-	-	-	-	-	-	-	-	-	-				
		1	1	1	1	1	1	1	1	1	1	1	1		1	·			
54 Y[m]=																		/09/2006	

Figure 9-23 "Foundation properties" embedded program

To enter the foundation material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 9-23. The following list box in Figure 9-24 with default data appears. To enter or modify a value in this list box, type that value in the corresponding cell then press "Enter" key. In the list box of Figure 9-24, enter E-Modulus of the foundation, *Poisson's* ratio of the foundation and foundation thickness
- Click "OK" button

D	efining (	element grou	ips (with the sam	ne thickness and n	naterial) 🔀
	Group No.	E-Modulus of slab [kN/m2]	Poisson's ratio of slab [-]	Slab thickness d [m]	<u>D</u> k Cancel
	1	2E+07	0.15	1	
					Insert
					Сору
					<u>D</u> elete
					New
					<u>H</u> elp
					<u>E</u> xcel

Figure 9-24 "Defining element groups" list box

To enter the unit weight of the foundation

- Choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 9-23. The following dialog box in Figure 9-25 with a default unit weight of 25 [kN/m<sup>3</sup>] appears. To neglect the self-weight of the foundation in the analysis, type 0 in the edit box "Unit weight of the foundation"
- Click "OK" button

Unit weight of the foundation		x
Unit weight of the foundation	Gb [kN/m3] 0	
<u>k</u> ew	<u>C</u> ancel <u>H</u> elp	

Figure 9-25 "Unit weight of the foundation" dialog box

To enter the foundation level

- Choose "Foundation depth" command from "Foundation properties" menu in the window of Figure 9-23. The following dialog box in Figure 9-26 appears
- Type 1.5 in the "Foundation depth under ground surface (a)/ (b)" edit box
- Click "OK" button

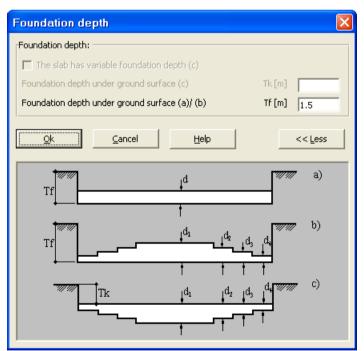


Figure 9-26 "Foundation depth" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 9-23 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 9-23 to close the "Foundation properties" embedded program and to return to the main window of *ELPLA-Data*

#### 2.6 Loads

To define the loads, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 9-27 appears.

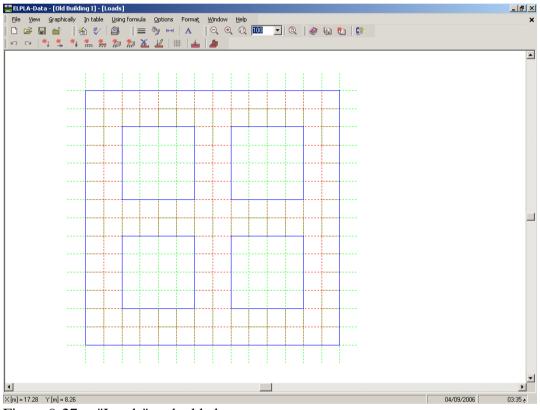


Figure 9-27 "Loads" embedded program

To enter loads

- Choose "Point loads" command from "In Table" menu in the window of Figure 9-27. The following list box in Figure 9-28 appears
- Enter the external point loads P [kN] and their corresponding coordinates (x, y) in the list box of Figure 9-28. This is done by typing the value in the corresponding cell and then press "Enter" key. The coordinates of the point load are related to the lower-left corner of the foundation (local coordinates)
- Click "OK" button

Point I	oads				×
No.   [•]	Column types I [-]	Load P [kN]	X-position x [m]	Y-position y [m]	<u>Ok</u> Cancel
1	1	500.0	0.72	0.72	
2	1	500.0	0.72	9.36	Insert
3	1	500.0 500.0	9.36 9.36	9.36 0.72	
5	1	1000.0	5.04	9.36	<u>C</u> opy
6	1	1000.0	5.04	0.72	Delete
7	1	1000.0	9.36	5.04	
8	1	1000.0	0.72	5.04	New
9	1	2000.0	5.04	5.04	
					<u>H</u> elp
					<u>Excel</u>

Figure 9-28 "Point loads *P*" list box

After you have completed the definition of all load data, the screen should look like the following Figure 9-29.

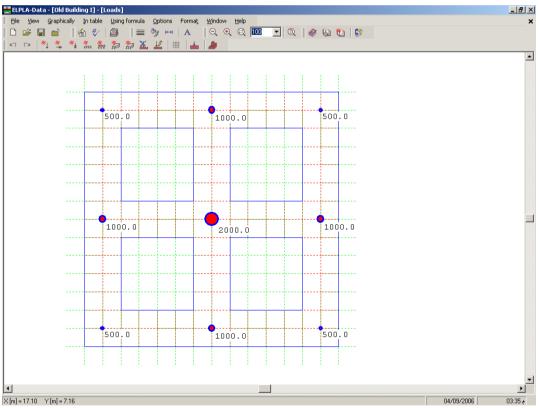


Figure 9-29 Loads on the screen

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

- Choose "Save loads" command from "File" menu in Figure 9-29 to save the load data
- Choose "Close loads" command from "File" menu in Figure 9-29 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

The project creation of the New Building II is now completed.

#### **3** Creating the project of the Old Building I

The data of the two foundations are quite similar except the origin coordinates of the global system, which are chosen to be (0.0, 0.0) and (10.28, 0.0) for buildings I and II respectively. Project identification is entered so that you can distinguish between the two projects. The data of Old Building I are created by first saving the data of the New Building II under a new file name and then modifying the project identification and origin coordinates.

To save the data under a new file name

- Choose "Save project as" command from "File" menu of *ELPLA-Data*. The following "Save as" dialog box appears, Figure 9-30

In this dialog box

- Type a file name for the project of the Old Building I in the file name edit box. For example type "Old Building I"
- Click "Save" button

Save As		<u>? ×</u>
Savejn: 隘	Influence of a new neighboring buildir 💌 🖛 🗈 📸 🎹	
New Buildin	-	
Old Building	gI	
J		
File <u>n</u> ame:	Old Building I Sav	e
Save as <u>t</u> ype:	Isolated slab foundation-files (*.P01)	:el

Figure 9-30 "Save as" dialog box

#### 3.1 Modifying the calculation methods options

To add the option of neighboring foundations do the following steps:

- Choose "Calculation methods" command from "Data" menu
- The first form of the wizard "Calculation method" is the "Analysis type" form, Figure 9-2
- In this form, click "Next" button to go to the next page
- After clicking "Next" button, the "Calculation methods" form appears, Figure 9-3. Click "Next" button to go to the next form
- The next form is the "System symmetry", Figure 9-4. In this form click "Next" button
- The last form of the wizard assistant contains the "Option" list, Figure 9-31. In this list, check the option "Influence of neighboring foundations on the slab", then click the "Save" button



Figure 9-31 "Options" list

#### **3.2** Modifying the project identification

To modify the project identification of building II, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 9-32 appears. In this dialog, type "Old building I" in the "Project" edit box then click "Save" button.

Ρ	roject Ide	ntification X								
	Project Ide	ntification:								
	Title	Influence of a new neighboring building II on an old one I								
	Date	17.07.1998								
	Project	Old Building I								
	<u>S</u> ave	Cancel Help Load Save As								

Figure 9-32 "Project identification" dialog box

#### 3.3 Modifying origin coordinates

To modify the origin coordinate of building II, choose "Foundation properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 9-33 appears.

<u>V</u> iew <u>G</u> raphi				_		erties	) Options	Eor	mat 1	Window	, Hel	n							
	4		1										0	1 🔊	۱. Ka	tu   💱			
	1		1							, -,	-			1~	0				
	1	1		1997															
		1	1	-	4	- 1	1	- 1	- 1	-	1	- 1	1	-	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		1	1	i —				1	1	i –				1	1				
		1	1					1	1					1	1				
		1	1					1	1					1	1				
												ļ							
		1	1					1	1					1	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
				-	-	-	-			-	-	-	-		L				
		1	1					1	1					1	1				
		1	1					1	1					1	1				
		1	1					1	1					1	1				
															L				
		1	1					1	1					1	1				
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
		1			-1	-	1	4			-	-		-					
		1	1	1	1	1	1	1	1	1	1	1	1	1	1				
						1													

Figure 9-33 "Foundation properties" embedded program

To enter the origin coordinates

- Choose "Origin coordinates" command from "Foundation properties" menu in the window of Figure 9-23. The following dialog box in Figure 9-34 appears
- In this dialog box type 10.28 in the "x-coordinate" edit box
- Click "OK" button

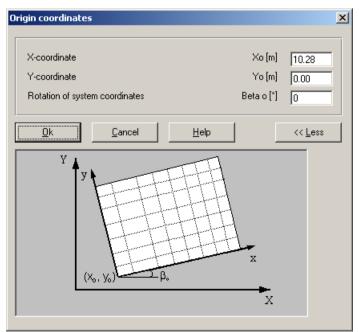


Figure 9-34 "Origin coordinate" dialog box

After modifying the foundation properties, do the following two steps:

- Choose "Save foundation properties" command from "File" menu to save the foundation properties
- Choose "Close foundation properties" command from "File" menu to close the "Foundation properties" embedded program and return to *ELPLA-Data*

#### **3.4** Neighboring foundations

To enter the neighboring foundations, do the following steps:

- Choose "Neighboring foundations" command from "Data" menu from *ELPLA*-*Data* (Figure 9-35)
- Click "Add neighboring foundations" button in "Data" menu and open the project "New Building II"
- Click "Save" button

Creating the projects of the two foundations 1 and 2 is now complete.

N	eighboring fou	Indations	×
	Neighboring foundation No.	File name of neighboring foundation	Save
	1	New Building II	<u>C</u> ancel
			Add neighboring foundation
			<u>R</u> emove neighboring foundation
			Load
			Save <u>A</u> s
			New
			<u>H</u> elp

Figure 9-35 "Neighboring foundations" menu

#### 4 Carrying out the calculations

The calculations of the New Building II should be first done to get the contact pressure under its foundation, then the calculation of the Old Building I with the influence of building II is carried out.

#### 4.1 Starting *ELPLA-Solver*

To analyze the problem open the file "New Building II" from "File" menu from *ELPLA-Data*, then leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window appears, Figure 9-36.

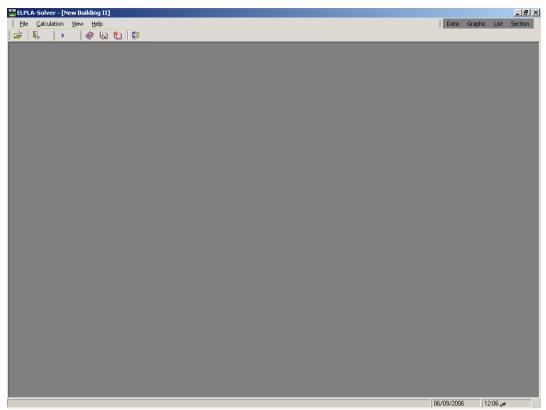


Figure 9-36 Opening screen of the sub program ELPLA-Solver

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. The "Calculation" menu contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis.

For the project of the New building II, the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Assembling the slab stiffness matrix
- Solving the system of linear equations
- Determining deformation, internal forces, contact pressures

While for the project of the Old building I, the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Influence of neighboring foundations on settlements
- Assembling the slab stiffness matrix
- Solving the system of linear equations
- Determining deformation, internal forces, contact pressures

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

#### 4.2 Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Calculation" menu in *ELPLA*-*Solver* Window

#### **Analysis progress**

Analysis progress menu in Figure 9-37 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

Determining flexibility coefficients of the soil	×
Assembling the flexibility matrix!	
Time remaining = 00:00:04 I = 81 from 189 steps	<u>C</u> ancel

Figure 9-37 Analysis progress menu

#### Check of the solution

Once the analysis is complete, a check menu of the solution appears, Figure 9-38. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

Check of the solution		
V - Load		
Total load	[kN] =	8000.0
Sum of contact pressures	[kN] =	8022.1
X - Moment		
Sum Mx from loads	field and	0.0
	[kN.m] =	
Sum Mx from contact pressures	[kN.m] =	-2.2
Y - Moment		
Sum My from loads	[kN.m] =	0.0
Sum My from contact pressures	[kN.m] =	28.5
Ok <u>H</u> elp		

Figure 9-38 Menu "Check of the solution"

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

#### 5 Viewing data and results

*ELPLA* can view the data and results for each foundation separately or for the two foundations together. Individual data or results can be viewed in a manner similar to that in the previous examples. Here, you will learn to view the results of the foundations together through the following two samples.

#### 5.1 Viewing result graphics

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

*ELPLA-Graphic* window appears, Figure 9-39. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

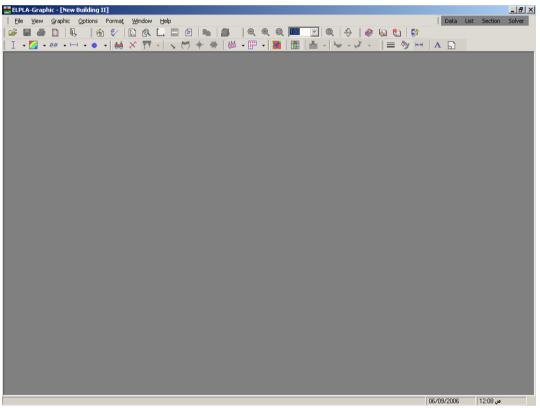


Figure 9-39 Opening screen of the sub program *ELPLA-Graphic* 

To view the results for the new foundation only as contour lines

- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 9-40 appears
- In "Results as contour lines" option box, select "Settlements *s*" as a sample for the results to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for old foundation only as shown in Figure 9-41.

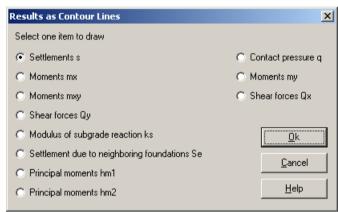


Figure 9-40 "Results as contour lines" option box

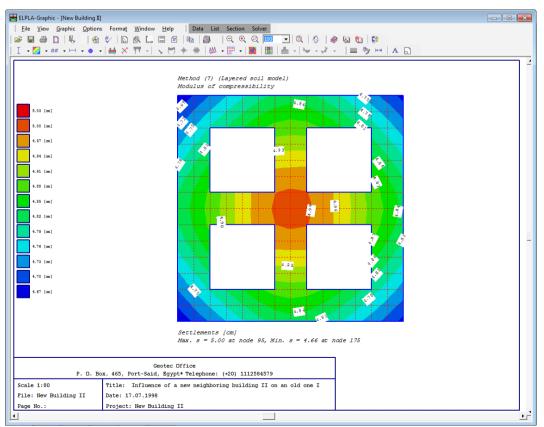


Figure 9-41 Settlements as contour lines for the old foundation only

To view the settlement results for the two foundations together as contour lines

- Choose "Old Building I" file from "File" menu from ELPLA- Graphic
- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The option box in Figure 9-40 appears
- In "Results as contour lines" option box, select "Settlements *s*" as a sample for the results to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for the old foundation as shown in Figure 9-42.

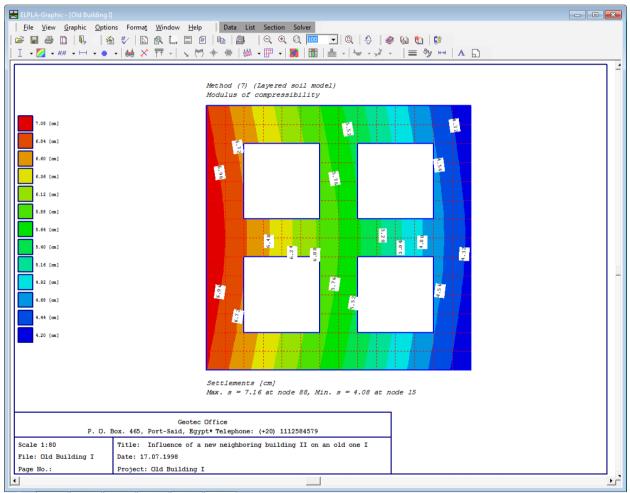


Figure 9-42 Settlements as contour lines for the old foundation

#### 5.2 Plot a diagram of results

To plot a diagram of results, leave *ELPLA-Graphic* to *ELPLA-Section*. This is done by clicking on "Section" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Graphic* window.

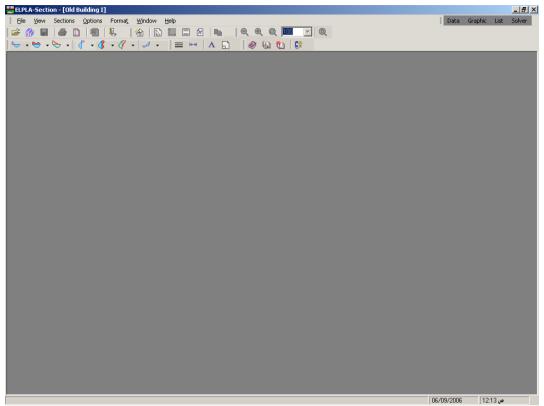


Figure 9-43 Opening screen of the sub program *ELPLA-Section* 

To consider the results for the two foundations together

- Choose "Combination from many projects" command from "File" menu of *ELPLA-Section*. The following list box in Figure 9-44 appears. *ELPLA-Section* automatically considers the project "Old Building I" in the list of projects to be combined
- Click "Add project" button in "Combination from many projects" list box. Then open the project "New Building II"
- Click "OK" button in the "Combination from many projects" list box

Co	mbin	ation from Many Project	s	×
٢L	ist of (	projects to be combined		
	No.	File name of the project	Project Identification	<u></u>
	1	G:\TEMP\Influence of a	New Building II	Cancel
	2	G:\TEMP\Influence of a	Old Building I	
				A <u>d</u> d Project
				<u>R</u> emove Project
				New
				<u>H</u> elp

Figure 9-44 "Combination from many projects" list box

To plot a section in *x*-direction

- Choose "Section in *x*-direction" command from "Sections" menu. The following option box in Figure 9-45 appears
- In "Section in *x*-direction" option box, select "Settlements *s*" as a sample for the results to be plotted in a diagram
- Click "OK" button

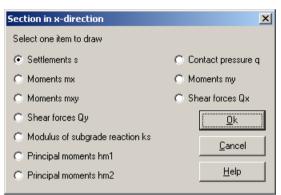


Figure 9-45 "Section in *x*-direction" option box

The following option box in Figure 9-46 appears to specify the section in x-direction that is required to be plotted.

In this dialog box

- Type 5.04 in the "Section at *y*-coordinate" edit box to plot a diagram at the middle of the two foundations
- Click "OK" button

The settlements are now plotted in a diagram at the middle of the two foundations together as shown in Figure 9-47.

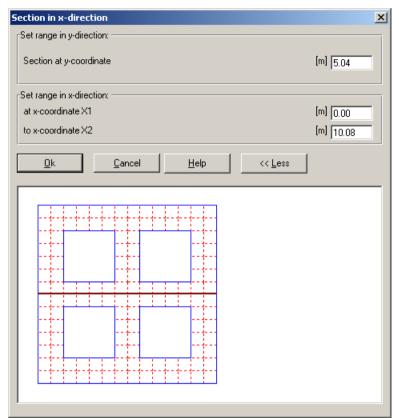


Figure 9-47 "Section in *x*-direction" dialog box

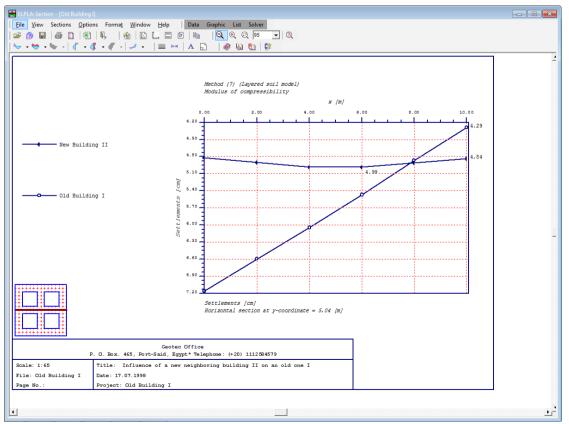


Figure 9-48 Diagram of settlements in *x*-direction at the middle of the two foundations

### 6 Index

# A

# B

# С

Calculation		۲۷
Calculation methods		
Check of the solution		
Computation of all		۲۸
Contact pressures	5, 26,	۲۷
Contour lines	٣٠,	۳١
Coordinates	۳, 0, ۲۱, ۲٤,	۲0

# D

Deformation	۲۷
Diagram	۳١

# E

ELPLA-Data $7, \Lambda, 9, 1., 15, 1\Lambda,$	۲٢,	۲۳, ۲٤, ۲٦
ELPLA-Graphic		۲۹, ۳۰, ۳۱
ELPLA-Section		۳۱٫۳۲
ELPLA-Solver	۲٦,	۲۷, ۲۸, ۲۹
E-Modulus		١٩

# F

FE-Net	•
File ٦, ٨, ١٤, ١٨, ٢٢, ٢٣, ٢٦, 27, 29, ٣١, ٢	۲۲
Flexibility coefficients	í٧
Foundation ٣, ٥, ٦, ١٠, ١١, ١٦, 1٩, ٢٠, ٢١, 23, 2	4,
۲٦, 29, 30, 33	
Foundation properties	٨

# G

Generation parameters		۱۱
German specification code ·		۱٥
Graphic	29,	۳.
Groundwater	۱٦,	۱۷

# Ι

Internal forces5,	۲۷
L	
Load vector	<b>T</b> V
Loads	۲۱

# M

Main soil data	۷
Modulus of compressibility for loading	٥
Modulus of compressibility for reloading	٥

## N

Nodes	٣	٥	١٢	١٣
10000	,	,	,	

## 0

Origin coordinate ...... 23, 25

# P

Point loads	۳, ۲۱, ۲۲
Poisson's ratio	5, 19
Project identification	9,23,75

## R

۱	١	٢
	۱	۱١

# S

Save	. v
Save project as	۲۳
Select nodes	
Slab type	
Soil properties	١٤
Soil stiffness matrix	
System symmetry <sup>V</sup> ,	۲٣

# U

Unit weight	 5, 15,	۲۰
Unsymmetrical system	 	. ۲

Example 10

Analysis of raft with straight and curved borders

## ELPLA-Tutorial

## Contents

1	Desc	cription of the problem	3
	1.1		3
	1.2	Foundation material and thickness	4
	1.3	Soil properties	5
	1.4	Mathematical model	5
2	Crea	ting the project	5
	2.1	Calculation method	5
	2.2	Project identification	9
	2.3	FE-Net data	10
	2.4	Soil properties	19
	2.5	Foundation properties	23
	2.6	Loads	26
3	Carry	ying out the calculations	28
	3.1	Starting ELPLA-Solver	28
	3.2	Carrying out all computations	29
4	View	ving data and results	31
	4.1	Viewing result graphics	31
	4.2		32
	4.3	Listing data and result in tables	35
5	Indo		20

5 Index

39

### **1** Description of the problem

An example of a raft with straight and curved borders is selected to illustrate some of the essential features of *ELPLA*.

#### 1.1 Loads and dimensions

The raft is subjected to 12 concentrated loads, each is 500 [kN] as shown in Figure 10-1 and Table 10-1.

Load No.	Load value	X-coord.	Y-coord.
I	P	x	Y
[-]	[kN]	[m]	[m]
1	500	5.42	9.5
2	500	3.25	4.96
3	500	7	0.5
4	500	8.02	9.5
5	500	11.12	9.5
6	500	11.14	0.5
7	500	15	2
8	500	15	8
9	500	0.75	0.5
10	500	16.25	4.98
11	500	3.84	0.5
12	500	8	5

Table 10-1Point loads P

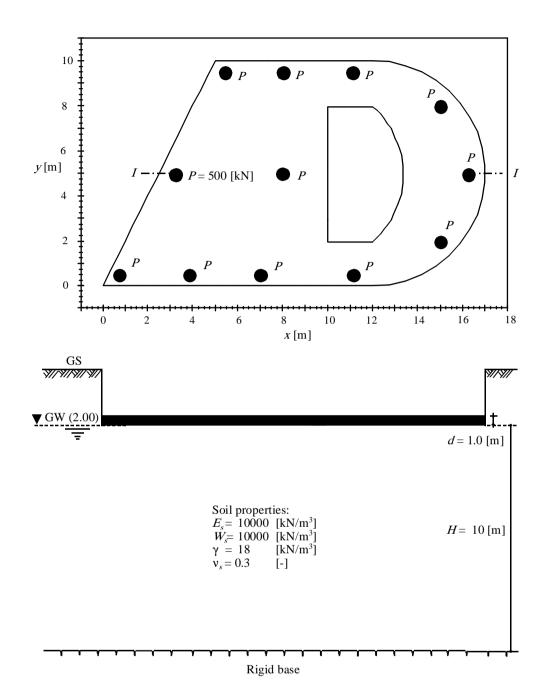


Figure 10-1 Plan and section elevation with raft geometry [m] and acting loads [kN]

#### **1.2** Foundation material and thickness

Raft material and thickness are supposed to have the following parameters:

Raft thickness	d	= 1.00	[m]
Young's modulus	$E_b$	$=2 * 10^{7}$	$[kN/m^2]$
Unit weight	$\gamma_b$	= 25	$[kN/m^3]$
Poisson's ratio	$v_b$	= 0.25	[-]

#### 1.3 Soil properties

The subsoil under the raft consists of a layer of silt, 12 [m] thick, the water table level is 2 [m] below the ground surface. Foundation level is also 2 [m] below the ground surface as shown in Figure 10-1. The soil is supposed to have the following parameters:

Modulus of elasticity for loading	$E_s$	= 10000	$[kN/m^2]$
Modulus of elasticity for reloading	$W_s$	= 10000	$[kN/m^2]$
Unit weight	$\gamma_s$	= 18	$[kN/m^3]$
Poisson's ratio	$v_s$	= 0.3	[-]

#### **1.4** Mathematical model

In this example, the Modulus of compressibility method (method 6) is chosen to analyze the raft. This Tutorial manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for the soil models and numerical calculation methods are well documented in the User's Guide of *ELPLA*.

#### 2 Creating the project

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

#### 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation method" wizard appears, Figure 10-2. This wizard will guide you through the steps required to create the project. As shown in Figure 10-2, the first form of the wizard is the "Analysis type" form. In this form, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a foundation problem, select "Analysis of slab foundation" then click "Next" button to go to the next form.

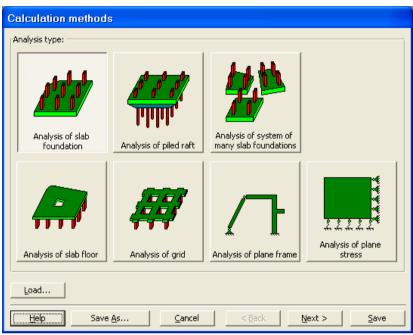


Figure 10-2 "Calculation method" wizard with "Analysis type" form

After clicking "Next" button, the "Calculation methods" form appears, Figure 10-3.

To define the calculation method

- Select the calculation method "6-Modulus of Compressibility (Iteration)"
- Click "Next" button to go to the next form

Calculation methods
Calculation methods:
C 1- Linear Contact Pressure (Conventional Method)
C 2/3- Constant/ Variable Modulus of subgrade Reaction
C 4- Modification of Modulus of subgrade Reaction by Iteration
C 5- Isotropic Elastic Half Space
6- Modulus of Compressibility (Iteration)
C 7- Modulus of Compressibility (Elimination)
C 8- Modulus of Compressibility for Rigid Raft
C 9- Flexible foundation
Subsoil model: C Half Space model C Layered soil model
Help     Save As     Cancel     < Back     Next >

Figure 10-3 "Calculation methods" form

The next form is the "System symmetry" (Figure 10-4). In this form choose "Unsymmetrical system" then click "Next" button.

Calculation methods		
System symmetry:		
Unsymmetrical system		
Symmetrical system about x-axis	Double-symmetrical system	
Symmetrical system about y-axis	Anti-symmetrical system about x-axis	
Help Save As Cancel	< Back	-
Symmetrical system about y-axis	Anti-symmetrical system about x-axis	

Figure 10-4 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 10-5. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other. There isn't any requested choices, so click the "Save" button.

Calculation methods
Options:
Slab with girders
Addtional springs
Supports/ Boundary Conditions
Determining limit depth
Nonlinear subsoil model
Determining displacements in soil
Determining stresses in soil
Determining strains in soil
Triblence of the temperature change on raft
$\Box$ Influence of additional settlements on raft
Select All
Nonlinear analysis of piled raft:
Nonlinear analysis using a hyperbolic function for load-settlement
C Nonlinear analysis using DIN 4014 for load-settlement
Help         Save As         Cancel         < Back

Figure 10-5 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 10-6.

In this dialog box

- Type a file name for the current project in the "File name" edit box. For example type "Curved Raft". *ELPLA* will use automatically this file name in all reading and writing processes
- Click "Save" button

Save As		?	
Save jn: 🗀 Tutorial e	examples	- 🖬 🍅 🖬 -	
Beam.P01 Curved Raft.P01 Example.P01 Floor.P01 Frame.P01 Grid.P01	Vew Building II.P01		
File <u>n</u> ame: Curved I	Raft	<u>S</u> ave	
Save as <u>type</u> : Isolated	slab foundation-files (*.PO1	i) 💽 Cancel	

Figure 10-6 "Save as" dialog box

*ELPLA* will activate the "Data" menu. In addition, the file name of the current project [Curved Raft] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 10-7. In the "Data" menu, the user can enter the remaining data of the project using the same sequence of commands in this menu. The first command in the menu is "Calculation methods", which has been already entered. Therefore, *ELPLA* has put the sign " $\sqrt{}$ " beside this command, Figure 10-7. *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.

### ELPLA-Tutorial

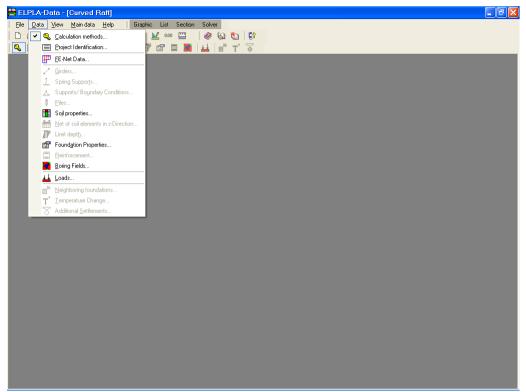


Figure 10-7 ELPLA-Data after defining the calculation method

### 2.2 **Project identification**

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 10-8 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a raft with straight and curved borders"
- Type the date of the project in the "Date" edit box
- Type "Tutorial Manual" in the "Project" edit box
- Click "Save" button

Project I	dentification	×
Project Ide	entification:	
Title	Analysis of a raft with straight and curved borders	-
Date	15.08.2004	•
Project	Tutorial Manual)	
<u>S</u> ave	Cancel Help Load Save As	

Figure 10-8 "Project identification" dialog box

#### 2.3 FE-Net data

Choose "FE-Net data" command from "Data" menu of *ELPLA-Data*. The "FE-Net generation" wizard appears as shown in Figure 10-9. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 10-9 the first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have regular shapes. For the given problem, the foundation has an irregular shape.

To generate the FE-Net

- In the "Slab type" options, choose the irregular slab option
- Click "Next" button to go to the next form

FE-Net Generation	
Slab type:	
	0
rSquare slab:	
Length/ width of square slab L/ B [m]	20.00
Width of rectangular slab B [m]	14.00
Help Cancel < Back Next >	Einish

Figure 10-9 "Slab type" form

After clicking "Next" in "Slab type" form, the following "Generation type" form appears, Figure 10-10. *ELPLA* can deal with various type of generations with rectangular/ triangular elements. Choose the triangular element which is the last option as the generation type. Then click "Next" button to go to the next form.

FE-Net Generatio	n		
Generation type:			
Help		< <u>B</u> ack Next >	Einish

Figure 10-10 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "FE-Net Generation" dialog box in Figure 10-11 appears with the default generation parameters. In this dialog box, click "Finish" button.

FE-Net Generation	
Generation parameters:	
Min. angle theta [°]	30
Element circumradius r [m]	1.00
Min. element circumradius rmin [m]	0.25
Mesh optimization: Smoothing mesh Directing border elements	
Help Cancel < Back Mext >	Einish

Figure 10-11 "Fe-Net generation" dialog box

*ELPLA* will generate an imaginary net as shown in the following embedded program, Figure 10-12.

		<u>G</u> raphic																			
		۳.											100	•	Q	1	6	Û	6		
al .	·•	*	-	P	6	78	•   #	#   •				4									
							, ÷														
1											 									 	
							· · · · ·			·	 		ļ							 	
											 		+							 	
					ļ					·	 		ļ							 	
										· · · · ·	 		ļ							 	
											 		ļ							 	
					·				÷												

Figure 10-12 Imaginary net of a rectangular area on the screen

You can either draw the raft directly on this imaginary net or input the raft corner points in tabulated form. To input the FE-Net corner points, choose "Slab corners" command from "In Table" menu. The dialog box in Figure 10-13 appears.

Slab corners				2
Slab corners: Segment No. 1 from 3	segments: ·			
Start poistion	×1	[m] 0.00		
	у1	[m] 0.00		
End position	x2	[m] 0.00		
	y2	[m] 0.00		
Use arc data				
Arc radius	R	[m] 0.00		
Min. arc radius	Rmin	[m] 0.00	Copy Segment	
Reverse rotat	ion direction		Insert Segment	
Reverse radiu	s position			
			Delete Segment	
<u>Qk</u>	Cano	el	<u>H</u> elp <u>N</u> ew <u>R</u> efresh	

Figure 10-13 "Slab corners" dialog box

To define the first segment of the raft that connects the first two points on the raft of Figure 10-1

- Do not change the default coordinates of the first point, which the program considers as (0.00, 0.00)
- Type 12 in the " $x^2$ " edit box to define the *x*-coordinate of the second point
- Type zero in the "y2" edit box to define the y-coordinate of the second point

To define the second segment of the raft

- Use the Scroll bar to define segment No. 2
- Type 12 in the "x2 edit box to define the x-coordinate of the third point
- Type 10 in the "y2" edit box to define the y-coordinate of the third point
- Select the "Use arc data" option to convert the line segment to arc segment
- Select the "Reverse rotation direction" option to reverse the direction of rotation
- Select the "Reverse radius position" option to reverse the radius position
- Type 5 in the "Arc radius" edit box to define the radius of the arc segment

In the "Slab corners" dialog box, *ELPLA* assumed that there are at least three segments with three corner points. Since the raft drawing contains four segments, you can use the command "Segment insert" to insert the last segment of the raft. Use the corner points and arc information provided in Table 10-2 to complete the definition of the raft corner points. Repeat the steps used for defining the first two segments to define any segment.

_	uble 10 2	Ruit comer	Somes			
	Segment	Start pos	sition	End pos	sition	Arc Radius
	No.	$x_1$	<i>y</i> 1	$x_2$	<i>y</i> 2	
	[-]	[m]	[m]	[m]	[m]	[m]
	1	0	0	12	0	
	2	12	0	12	10	5
	3	12	10	5	10	
	4	5	10	0	0	

Table 10-2Raft corner points

After completing the definition of the raft corner points, the slab corner dialog box should be like this in Figure 10-14 where a small sketch of the raft appears in the dialog box picture. Click the "OK" button to see the drawing of the raft outlines as shown in Figure 10-15.

Slab corners				
Slab corners: Segment No. 1 from 4	segments:			
Start poistion	×1	[m] 0.00		
	y1	[m] 0.00		
End position	x2	[m] 12.00		
	y2	[m] 0.00		
Use arc data				
Arc radius	R	[m] 0.00		
Min. arc radius	Rmin	[m] 6.00	Copy Segment	
Reverse rotati	on direction		Insert Segment	
Reverse radiu:	s position		Delete Segment	-1
Ok	⊆and	el	Help New	Refresh
<u> </u>	Cano			Kenesn

Figure 10-14 "Slab corners" dialog box after defining the raft corner points

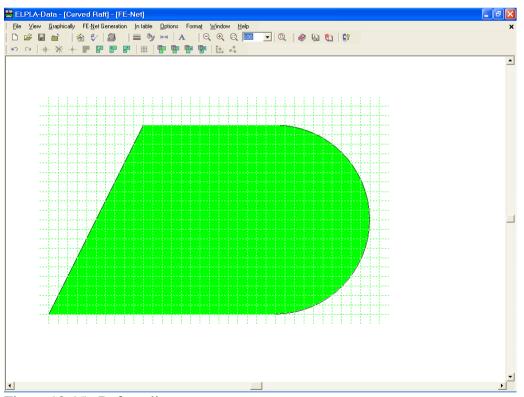


Figure 10-15 Raft outlines

### **Raft openings**

To draw the openings in the raft, choose the command "Opening corners" from "In Table" menu, the dialog box in Figure 10-16 appears.

Openings				×			
There is no opening: Segment No. 0 from 0 s	egments: -			-			
Start poistion	×1	[m] 0.00					
	у1	[m] 0.00					
End position	x2	[m] 0.00					
	y2	[m] 0.00					
Use arc data							
Arc radius	R	[m] 0.00					
Min. arc radius	Rmin	[m] <b>0.00</b>	<u>C</u> opy Segment				
🗌 🗌 Reverse rotatio	n direction		Insert Segment				
Reverse radius	position		Delete Segment				
Copy Opening Insert Opening Delete Opening							
	<u>C</u> ance		Help New Refresh				

Figure 10-16 "Opening corners" dialog box

In this dialog box, press on "Insert opening" button, then write the coordinates of the opening exactly as in slab corners.

To define the first segment of the raft opening

- Type 10 in the "x1" edit box to define the x-coordinate of the first point
- Type 2 in the "y1" edit box to define the y-coordinate of the first point
- Type 12 in the " $x^2$ " edit box to define the *x*-coordinate of the second point
- Type 2 in the "y2" edit box to define the y-coordinate of the second point

To define the second segment of the raft opening

- Use the Scroll bar to define segment No. 2
- Type 12 in the "x2" edit box to define the x-coordinate of the third point
- Type 8 in the "y2" edit box to define the y-coordinate of the third point
- Select the "Use arc data" option to convert the line segment to arc segment
- Select the "Reverse rotation direction" option to reverse the direction of rotation
- Select the "Reverse radius position" option to reverse the radius position
- Type 4 in the "Arc radius" edit box to define the radius of the arc segment

In the "Opening corners" dialog box, *ELPLA* assumed that there are at least three segments with three corner points. Since the raft opening drawing contains four segments, you can use the command "Segment insert" to insert the last segment of the raft opening. Use the corner points and arc information provided in Table 10-3 to complete the definition of the raft opening corner points. Repeat the steps used for defining the first two segments to define any segment.

	There are a second				
Segment	Start pos	sition	End pos	sition	Arc Radius
No.	$x_1$	<i>y</i> 1	$x_2$	<i>y</i> 2	
[-]	[m]	[m]	[m]	[m]	[m]
1	10	2	12	2	
2	12	2	12	8	4
3	12	8	10	8	
4	10	8	10	2	

Table 10-3Raft opening corner points

After completing the definition of the raft opening corner points, the opening corners dialog box should be like this in Figure 10-17 where a small sketch of the raft opening appears in the dialog box picture. Click the "OK" button to see the drawing of the opening outlines as shown in Figure 10-18.

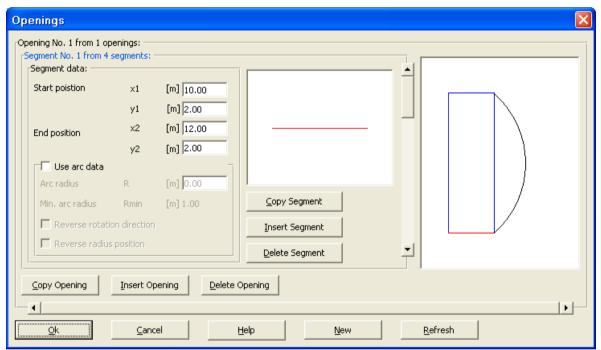


Figure 10-17 "Opening corners" dialog box after defining the opening corner points

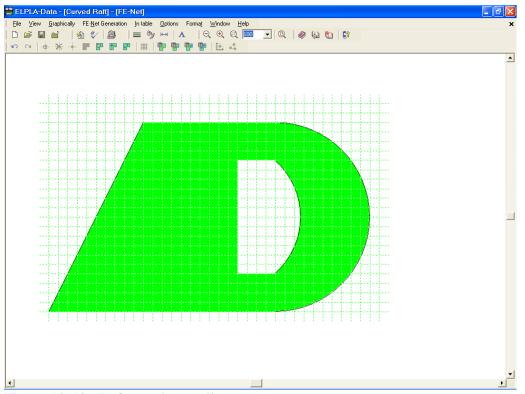


Figure 10-18 Raft opening outlines

To complete the FE-Net generation of the problem, choose "Generating FE-Net" command from "FE-Generation" menu. The following "Generation of FE-Net" dialog box appears.

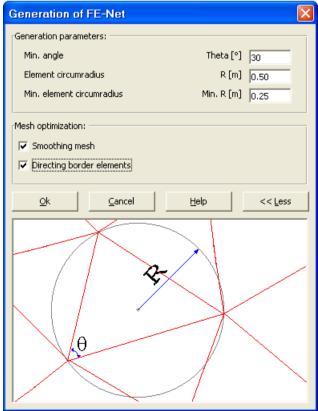


Figure 10-19 "Generation of FE-Net" dialog box

To enter the generation data

- Type 0.5 in the "Element circumradius" edit box to define the radius of circles containing elements
- Type 0.25 in the "Min. element circumradius" edit box to define the minimum radius of circles containing elements
- Check the "Smoothing mesh" option to optimize the dimension of FE-Net by making all elements having nearly the same area as possible as
- Click "OK" button

After clicking the "OK" button, FE-Net generation progress menu in Figure 10-20 appears in which various phases of the generation are progressively reported. After finishing the generation of the FE-Net and creating the raft FE-Net as shown in Figure 10-21, do the following two steps:

- Choose "Save FE-Net" command from "File" menu in Figure 10-21 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 10-21 to close the "FE-Net" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "FE-Net data" command in the "Data" menu of *ELPLA-Data*.

The generation of FE-Net is carrie	d out!	
Please wait!		
		Abort

Figure 10-20 "FE-Net generation" progress menu

### ELPLA-Tutorial

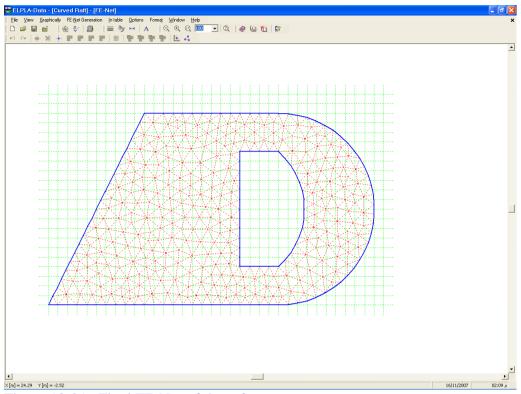


Figure 10-21 Final FE-Net of the raft

# 2.4 Soil properties

To define the soil properties, choose "Soil properties" command from "Data" menu of *ELPLA-Data*. The following sub program in Figure 10-22 appears with a default-boring log.

ELPLA-Boring - [Curved Raft]			. 7 🗙
<u>File View D</u> ata <u>G</u> raphically <u>O</u> ptions Forma <u>t W</u> indow <u>M</u> ain data <u>H</u> elp			
🗈 🖆 🖩 🤼 省 😻 🥔 🛍 💱			
📓 🗏 🚔 🦻 🗚 🔍 Q. Q. 🔟 🚽 Q. 🕲 🛠			
$a$ = 10.000 pm $ma_1/ma_1 = 30$ <sup>1</sup> $a$ = 10.000 pm $ma_1/ma_1 = 30$ <sup>1</sup> a = $a$ =			
			-
	20/10/2007	00.00	•
	30/10/2007	م 08:33	

Figure 10-22 *ELPLA-Boring* sub program with a default-boring log

To enter the soil properties for the boring log of the current example

- Choose "Soil data" command from "Data" menu in the window of Figure 10-22. The following dialog box in Figure 10-23 with default-boring log data appears

Soil data		×
Boring log No. 1 from 1 boring logs: Layer No. 1 from 1 layers: Soil and rock symbols: Main soil type 1 G, Gravel Main soil type 2 -, No symbole submain soil 1 -, No symbole submain soil 2 -, No symbole	Geotechnical data of the layer: Soil properties are defined by Modulus of Elasticity E E [kN/m2] 10000 Fhi [°] 30 W [kN/m2] 10000 c [kN/m2] 0	-
Color ge, yellow  Short text G	Gam     [kN/m3]     18     Nue     [-]     0.3       Layer depth under the ground surface     [m]     10.00	
Copy Boring log Delete Boring -	X-coordinate of boring [m] 0.00 Y-coordinate of boring [m] 0.00 Boring Log Label B1	
Cancel	New	Help

Figure 10-23 "Soil data" dialog box with default-boring log data

In the "Geotechnical data of the layer" dialog group box in Figure 10-23, define the geotechnical data of the first soil layer of the boring log as follows:

$E_s$	= 10000	$[kN/m^2]$
$W_s$	= 10000	$[kN/m^2]$
$\gamma_s$	= 18	$[kN/m^3]$
$v_s$	= 0.3	[-]

In the current example, the angle of internal friction  $\varphi$  and the cohesion *c* of the soil are not required because the selected type of the analysis is linear analysis. Therefore, the user can let the default values of the internal friction and the cohesion.

φs	= 30	[°]
С	= 0	$[kN/m^2]$

Due to the presence of the ground water, the soil above the ground water level has a deferential unit weight from the soil under that level. Therefore, the layer depth of the first layer is taken to be 2 [m], which is equal to the ground water level. Now, type this value in "Layer depth under the ground surface" edit box.

In order to draw the soil layers by different symbols according to the German Standard DIN 4023, the soil type and color for each layer must be defined.

To define the soil type and color for the first layer, select "U, Silt" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box, Figure 10-23. The color of the silt according to the German Standard DIN 4023 will be automatically created. The user can change this color. Also, a short text "U" will be automatically created for the silt.

To enter the second layer

- Click "Layer copy" button in 0. A layer that has the same properties of the first layer will be copied
- Use the vertical scrollbar to move to the second soil layer. Layer No. will be typed automatically at the upper-left corner of the main dialog box of soil layers as a head title
- Change the value of the unit weight of the soil for the second soil layer from 18  $[kN/m^3]$  to 9  $[kN/m^3]$
- Change the value of the layer depth under the ground surface from 2 [m] to [12 m]

Note that the unit weight of the soil is used to determine the overburden pressure  $q_v$  [kN/m<sup>2</sup>] due to the removed soil, which is equal to  $\gamma_s * d_f$ . This means that the unit weight of the soil under the foundation depth  $d_f$  is not required. However, the unit weight of the soil under the foundation depth for all soil layers is entered by the value 9 [kN/m<sup>3</sup>].

After editing the geotechnical data, the boring coordinates and labels which describe the boring will be entered.

To enter the boring coordinates and label

- Type 0 for *x*-coordinate in "*x*-coordinate of boring log [m]" edit box
- Type 0 for *y*-coordinate in "*y*-coordinate of boring log [m]" edit box
- Type B1 as a label name for the first boring in "Label of boring log" edit box

To enter the main soil data for all layers, choose "Main soil data" command from "Data" menu in Figure 10-22. The following dialog box in Figure 10-24 appears with default main soil data. The main soil data for the current example, which are required to be defined, are the settlement reduction factor  $\alpha$  [-] and the groundwater depth under the ground surface  $G_w$  [m]. Any other data corresponding to main soil data are not required in this example. Therefore, the user can take these data from the default soil properties.

In the dialog box of Figure 10-24, enter the settlement reduction factor  $\alpha$  [-] and the groundwater depth under the ground surface  $G_w$  [m]. Then click "OK" button.

Main Soil Data			$\mathbf{X}$
Soil properties Calculation parameters of flexibility coefficients	Bearing capacity fact	ors	
Main Soil Data:			
Settlement reduction factor Alfa <= 1	Alfa	[-]	1
Groundwater depth under the ground surface	Gw	[m]	2.00
	1		
<u></u>	Help		

Figure 10-24 "Main soil data" dialog box

Now all data and parameters for the boring log have been entered. After finishing the creation of the boring log, click "OK" button in "Soil data" dialog box in Figure 10-23 to see the defined boring log on the screen where the user can control or modify the input data and parameters, Figure 10-25.

### ELPLA-Tutorial

👖 ELPLA-Boring - [Curved Raft]	
Eile View Data Graphically Options Format Window Main data Help	
E ☞ 🖬 🔍 🐴 ∜ / ● 🍇 🐮 🕃   🖀 🔲 := ≫ ↦ 🗚 ୧୯ ୧୯ 🚾 🔍 ୧୯ ୮୬	
📓 🗏 🚔 🍬 🗚 🛛 Q Q 🔟 💽 🛠	
185	
E = 10000(MM/m2), Phi = 0[*] W = 10000(MM/m2), C = 11(MM/m2)	
u v = 10000[38/m2],c/= 13[88/m2] Gw 2.00 Gen = 10[18/m3],RUe = 0.1[-]	
U E = 10000180(m2), xPn1 = 0["] W = 10000180(m2), C = 13[00(m2) 12,000 Game = 0(50(m2), C = 0.3[-]	
12.00 Gam = B(kk/m3), New = 0.3(-)	
•	
	16/11/2007 02:26 p

Figure 10-25 Boring log on the screen

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

- Choose "Save boring log" command from "File" menu in Figure 10-25 to save the data of the boring log
- Choose "Close boring logs" command from "File" menu in Figure 10-25 to close the *ELPLA-Boring* sub program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Soil Properties" command in the "Data" menu of *ELPLA-Data*.

#### 2.5 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 10-26 appears with default foundation properties. The data of foundation properties for the current example, which are required to define, are foundation material and foundation thickness. Any other data corresponding to foundation properties in the program menus are not required. Therefore, the user can take these data from the default foundation properties.

### ELPLA-Tutorial

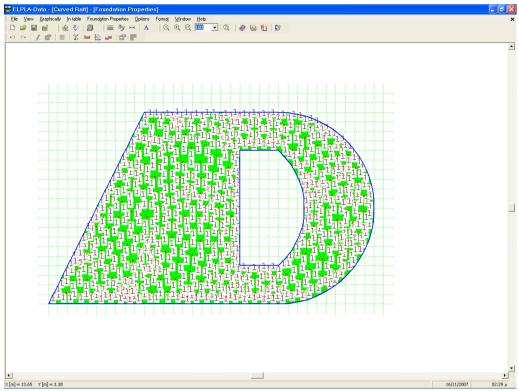


Figure 10-26 "Foundation properties" embedded program

To enter the foundation material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 10-26. The following list box in Figure 10-27 with default data appears. To enter or modify a value in this list box, type that value in the corresponding cell then press "Enter" key. In the list box of Figure 10-27, enter E-Modulus of the foundation, *Poisson's* ratio of the foundation and foundation thickness
- Click "OK" button

I	Defining element groups (with the same thickness and 🔀					
	Group No.	E-Modulus of slab [kN/m2]	Poisson's ratio of slab [-]	Slab thickness d [m]		<u>Ok</u> Cancel
	1	2E+07	0.25	1		
						Insert
						⊆ору
						Delete
						New
						Help
						<u>E</u> xcel

Figure 10-27 "Defining element groups" list box

To enter the unit weight of the foundation

- Choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 10-26. The following dialog box in Figure 10-28 with a default unit weight of 25 [kN/m<sup>3</sup>] appears. Let the default value as written in the edit box "Unit weight of the foundation"
- Click "OK" button

Unit weight of the found	ation	×
Unit weight of the foundation	Gb [kN/m3] 25	
<u>Ok</u> ew	Cancel <u>H</u> elp	

Figure 10-28 "Unit weight of the foundation" dialog box

To enter the foundation level

- Choose "Foundation depth" command from "Foundation properties" menu in the window of Figure 10-26. The following dialog box in Figure 10-29 appears
- In this dialog box type 2 in the "Foundation depth under ground surface (a)/ (b)" edit box
- Click "OK" button

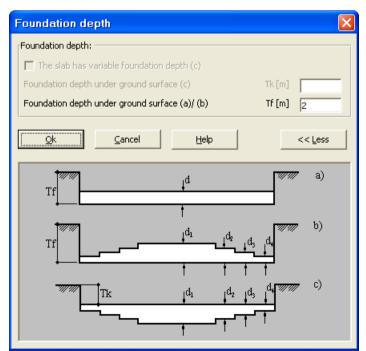


Figure 10-29 "Foundation depth" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 10-26 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 10-26 to close the "Foundation properties" embedded program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Foundation Properties" command in the "Data" menu of *ELPLA-Data*.

#### 2.6 Loads

To define the loads, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 10-30 appears.

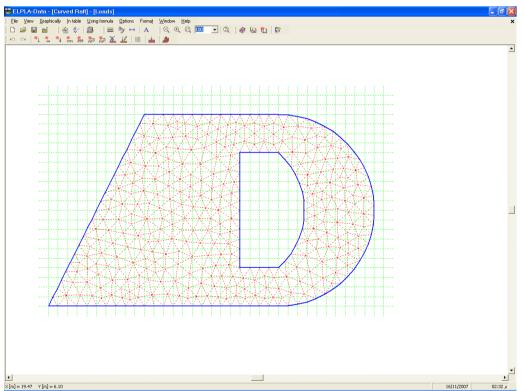


Figure 10-30 "Loads" embedded program

To enter loads

- Choose "Point loads" command from "In Table" menu in the window of Figure 10-30. The following list box in Figure 10-31 appears
- Enter the external point loads P [kN] and their corresponding coordinates (x, y) in the list box of Figure 10-31. This is done by typing the value in the corresponding cell and then press "Enter" key. The coordinates of the point load are related to the lower-left corner of the foundation (local coordinates)
- Click "OK" button

Poin	t loads				X
No. I [-]	Column types I [-]	Load P [kN]	X-position × [m]	Y-position y [m]	
1	1	500	5.42	9.50	<u>C</u> ancel
2	1	500	3.25	4.96	Insert
3	1	500	7.00	0.50	
4	1	500	8.02	9.50	⊆opy
5	1	500	11.12	9.50	
6	1	500	11.14	0.50	Delete
7	1	500	15.00	2.00	
8	1	500	15.00	8.00	New
9	1	500	0.75	0.50	<u></u>
10	1	500	16.25	4.98	Help
11	1	500	3.84	0.50	
12	1	500	8.00	5.00	Excel

Figure 10-31 "Point loads *P*" list box

After you have completed the definition of all load data, the screen should look like the following Figure 10-32.

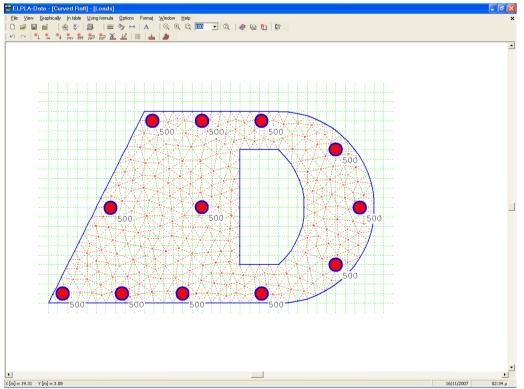


Figure 10-32 Loads on the screen

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

- Choose "Save loads" command from "File" menu in Figure 10-32 to save the load data
- Choose "Close loads" command from "File" menu in Figure 10-32 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Loads" command in the "Data" menu of *ELPLA-Data*.

Creating the project of the raft is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

### **3** Carrying out the calculations

### 3.1 Starting *ELPLA-Solver*

To analyze the problem leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window appears, Figure 10-33.

🚆 ELPLA-Solver - [Curved Raft]		- 7 🛛
Ele Calculation View Help Data Graphic List Section		
	_	_
	16/11/2007	02:35 p
Figure 10.22 Organize sources of the sub-success FLDIA Schum		

Figure 10-33 Opening screen of the sub program ELPLA-Solver

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. The "Calculation" menu contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Iteration process
- Determining deformation, internal forces, contact pressures

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

#### **3.2** Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Calculation" menu in *ELPLA*-*Solver* Window, the "Iteration parameters" option box will appear (Figure 10-34)
- Choose the suitable iteration option
- Press the "OK" button

Iteration parameters		
Which option is ending the iteration process?		
Accuracy [m]	0.0001	
C Iteration No.	10	
[		
Ok <u>C</u> ancel	<u>H</u> elp	

Figure 10-34 "Iteration parameters" option box

#### **Analysis progress**

Analysis progress menu in Figure 10-35 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

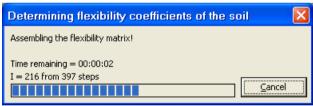


Figure 10-35 Analysis progress menu

#### **Iteration process**

Information about the convergence progress of the computations is displayed in the "Iteration process" list box in Figure 10-36 during the iteration process.

Iteration process				
Iteration No.	Accuracy [m]	Τ	Stop	
1	0.02049388000			
2	0.00100515400		<u>P</u> ause	
			Help	
Iteration cycles is ended at accuracy [m]<= 0.0001				
Computation time = 00:00:02				

Figure 10-36 "Iteration process" list box

#### Check of the solution

Once the analysis is complete, a check menu of the solution appears, Figure 10-37. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

Check of the solution	
V - Load	
Total load	[kN] = 8915
Sum of contact pressures	[kN] = 8864
X - Moment	
Sum M× from loads	[kN.m] = -137
Sum Mx from contact pressures	[kN.m] = -234
Y - Moment	
Sum My from loads	[kN.m] = -506
Sum My from contact pressures	[kN.m] = -667
( <u>O</u> k <u>H</u> elp	

Figure 10-37 Menu "Check of the solution"

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

## 4 Viewing data and results

*ELPLA* can view and print a wide variety of results in graphics, diagrams or tables through the three sub programs *ELPLA-Graphic*, *ELPLA-Section* and *ELPLA-List*. Data can also be viewed again and printed by the sub programs *ELPLA-Graphic* and *ELPLA-List*. Note that *ELPLA-Data* is used only to define and view the data of the problem. *ELPLA-Graphic* is used to print data graphically while *ELPLA-List* is used to print data numerically.

## 4.1 Viewing result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, leave *ELPLA-Solver* to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window. *ELPLA-Graphic* window appears, Figure 10-38. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

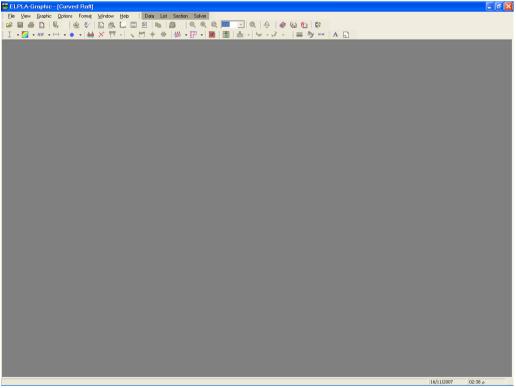
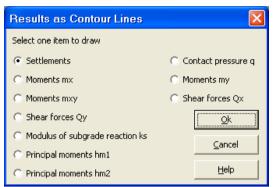


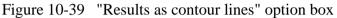
Figure 10-38 Opening screen of the sub program *ELPLA-Graphic* 

To view the results for the raft as contour lines

- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 10-39 appears
- In "Results as contour lines" option box, select "Settlements *s*" as a sample for the results to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for the raft as shown in Figure 10-40.





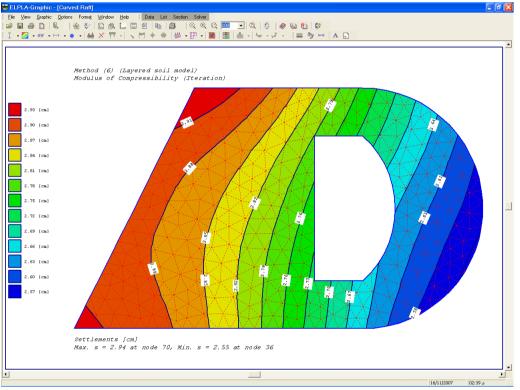


Figure 10-40 Settlements as contour lines for the raft

#### 4.2 Plot a diagram of the results

To plot a diagram of the results, leave *ELPLA-Graphic* to *ELPLA-Section*. This is done by clicking on "Section" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Graphic* window. *ELPLA-Section* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Section* window.

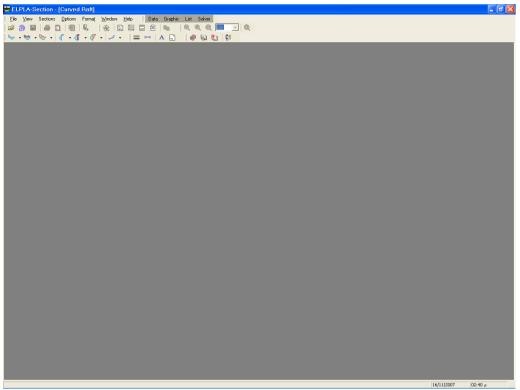


Figure 10-41 Opening screen of the sub program ELPLA-Section

Only the first command of the "Sections" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Options", "Format" and "Window" menus, which are used to define the preferences of the drawing such as plot parameters, scale, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

To plot a section in *x*-direction

- Choose "Section in *x*-direction" command from "Sections" menu. The following option box in Figure 10-42 appears
- In the "Section in *x*-direction" option box, select "Settlements *s*" as an example for the results to be plotted in a diagram
- Click "OK" button

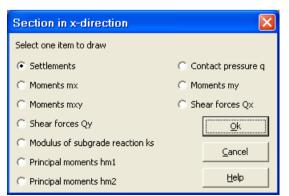


Figure 10-42 "Section in *x*-direction" option box

The following option box in Figure 10-43 appears to specify the section in x-direction that is required to be plotted.

In this dialog box

- Type 5.00 in the "Section at y-coordinate" edit box to plot a diagram at the middle of the raft
- Click "OK" button

The settlements are now plotted in a diagram at the middle of the raft as shown in Figure 10-44.

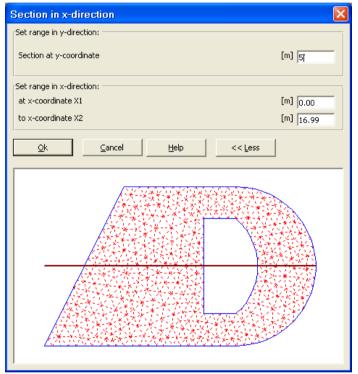


Figure 10-43 "Section in *x*-direction" dialog box

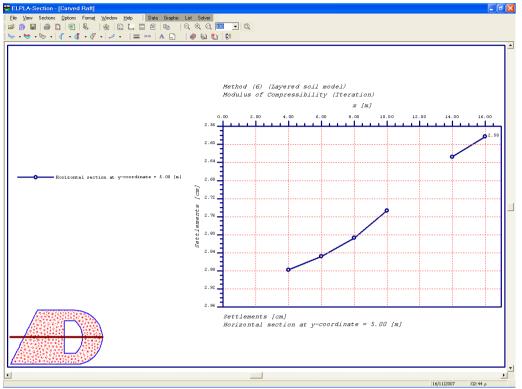


Figure 10-44 Diagram of settlements in x-direction at the middle of the raft

#### 4.3 Listing data and result in tables

To list tables of data and results, switch to *ELPLA-List*. This is done by clicking on "List" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Section* window. *ELPLA-List* window appears, Figure 10-45.

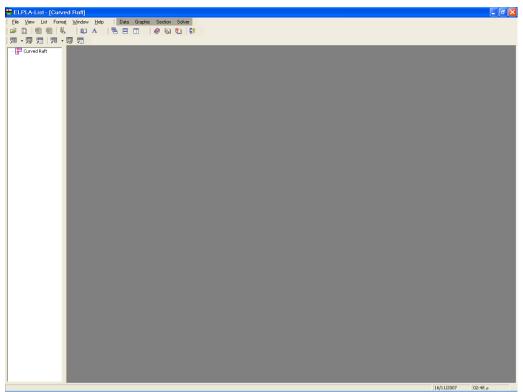


Figure 10-45 Opening screen of the sub program ELPLA-List

The function of *ELPLA-List* is listing and printing data and results in tables. The data and results can be exported to other Windows applications to prepare reports or add further information. *ELPLA-List* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-List* window.

Only one command of the "List" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Format" and "Window" menus, which are used to define the preferences of the tables such as page format, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

To list results in a table

- Choose "Display tables of results" command from "List" menu. The following option box in Figure 10-46 appears
- Select "Settlements" as an example for the results to be listed in a table
- Click "OK" button. The settlement results are now listed on the screen (Figure 10-47)
- Choose "Send to Excel" from "File" menu if you wish to export the table to the MS Excel application, Figure 10-48

Display Tables of Results	
Select one item to list	
C Modulus of subgrade reaction	Settlements
C Contact pressure	Slab deformation
C Moments	
C Shear forces	Qk
C Flexibility matrix [Ce]	Cancel
C Flexibility matrix [Cw]	
○ Soil stiffness matrix [ks]	Help



LPLA-List - [Curved Raft] - [S					
View List Format Window					
	· 1880 (*	Ø (1)   Pa			
•男月 <b>月</b> •月 日					
Curved Raft Node Total No.	Reloading Loading 🔺				
I s [-] [cm]	su se —				
1 2.92					
2 2.92	2 0.60 2.32				
3 2.91					
5 2.90	0 0.88 2.02				
6 2.90 7 2.89					
8 2.89	9 1.03 1.86				
9 2.88 10 2.87					
10 2.87					
12 2.85					
13 2.84					
15 2.82					
16 2.80 17 2.78					
18 2.77	7 1.11 1.66				
19 2.75					
21 2.71	1.05 1.66				
22 2.70 23 2.68					
24 2.66	5 0.99 1.67				
25 2.65 26 2.63					
27 2.61	l 0.92 1.69				
28 2.60 29 2.58					
30 2.57	7 0.90 1.67				
31 2.57 32 2.56					
33 2.55	5 0.90 1.66				
34 2.55 35 2.55					
36 2.55					
37 2.55					
38 2.55					
40 2.56					
41 2.56 42 2.57					
43 2.57	7 0.90 1.67				
44 2.58 45 2.59					
46 2.60	0 0.89 1.70				
47 2.61 48 2.62	0.90 1.72				
49 2 64				16/11/2007	م 02:49

Figure 10-47 List of settlement results

				<u>I</u> cols <u>D</u> a														a question for		
				V   X 🗈 I	B. • 🛷	17 - (1	- 😫 Σ	· 21 XI I	🏨 🦓 🛞	ari 🗧	el 🛛	• 10	• B /	Ū≣	= = =	3 %	• • • • • • • • • • • • • • • • • • •	律律[8]	<u>- 🗞 -</u>	4
	G17		fx:																	_
	A	В	C	D	E	F	G	н		J	K	L	M	N	0	P	Q	R	S	
	ode	Total	Reloading	Loading																
	lo.																			
!				se																
[-				[cm]																
	1																			
	2		0.6																-	
	3																			
	4			2.1															-	
	5			2.02															-	
	7			1.96						-				-			-	-		
	8			1.91			-	-		-				-						
	9			1.86														+	+	
	9		1.06																+	
	11							-	-								-	-	-	
	12						-	-	-	-							-	-	+	
	13																	-	-	
	14			1.72															-	
	15								-								-	-	-	
	16																	-	-	
	17			1.66				4											-	
	18								-								-	-	-	
	19			1.65			-													
	20			1.66																
	21			1.66			-													
	22			1.67																
	23			1.67																
	24			1.67																
	25			1.68																
	26			1.69																
	27		0.92	1.69															-	
	28																	-	-	
	29																		-	
	30	2.57	0.9	1.67			-	-	1	1				1			1	1	1	
	31							-	-	-								-	+	
	32			1.66					1								-		-	
	33																	-		
	34																-	-	-	
	35			1.65																
	36																	-		
	37																			
	38																			
	39			1.65																
	40		0.9	1.66																
	41																			
	42	2.57																		
	43	2.57	0.9	1.67																
	4.4	2.68	N 9 et2 / Shee	1.68																

Figure 10-48 Exported results in MS Excel

# 5 Index

# A

Analysis	5, 9, 21
Automatically8, 18, 21, 27,	31, 32, 36

# B

Borders	3, 9
Boring log19,	20, 23

# C

Calculation	
Calculation methods	5
Check of the solution	
Computation of all	
Contact pressures	
Contour lines	
Coordinates	
Curved	

# D

Deformation		.28
Diagram	31,	32

# E

ELPLA-Boring	
ELPLA-Data	5, 8, 10, 18, 23, 26, 27, 28, 31
ELPLA-Graphic	
ELPLA-List	
ELPLA-Section	
ELPLA-Solver	
E-Modulus	

# F

FE-Net	10, 11, 12, 17, 18, 19
File	5, 8, 18, 23, 26, 28, 36
Flexibility coefficients	
Foundation	
Foundation properties	

# G

aphic 31
----------

# Ι

Internal forces	28
Iteration process	8, 29

# K

# L

List	9, 33, 35, 36
Load vector	
Loads	

# M

Main soil data	22
Modulus of compressibility for loading	5
Modulus of compressibility for reloading	5

# N

Numerical calculation ...... 5

# 0

Opening	14,	15,	16
Options		7,	33

# P

Point loads	
Poisson's ratio	4, 5, 24
Project	
Project identification	9

# R

Raft 3, 4, 5, 9, 12, 13, 14, 15, 16, 17, 18, 19, 28, 32, 34, 35

# S

Save7, 8	, 9, 18, 23, 26, 27
Save as	8
Segment	
Settlement	
Slab type	
Soil	5, 19, 21
Soil properties	
Soil stiffness matrix	
Straight	
System symmetry	6

# T

# U

Unit weight	4, 5, 21, 25
Unsymmetrical system	б
User's Guide	

# Y

# Ζ

# Example 11

# Analysis of pile groups

# Contents

# Page

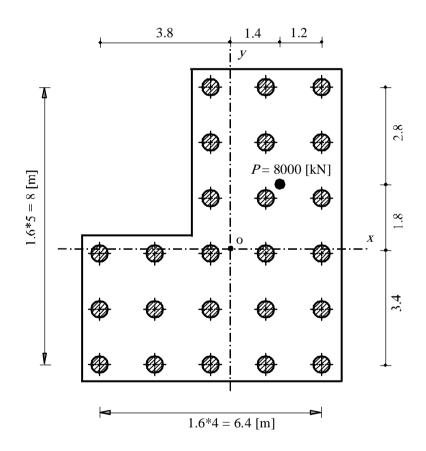
1	Desc	cription of the problem	3
	1.1	Load and dimensions	3
	1.2	Soil properties	3
	1.3	Mathematical model	4
2	Crea	ting the project	4
	2.1	Calculation method	4
	2.2	Project identification	7
	2.3	FE-Net data	8
	2.4	Element length of pile	12
	2.5	Piles	12
	2.6	Soil properties	16
	2.7	Foundation properties	19
	2.8	Loads	21
3	Carry	ying out the calculations	23
	3.1	Starting <i>ELPLA-Solver</i>	23
	3.2	Carrying out all computations	24
4	View	ving data and results	25
	4.1	Viewing result graphics	25
	4.2	Listing data and result in tables	26
5	Index	X	30

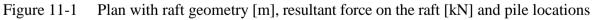
#### **1** Description of the problem

An example of pile groups is selected to illustrate some of the essential features of *ELPLA* for analyzing pile groups.

#### 1.1 Load and dimensions

A pile cap on 24 vertical piles is considered as shown in Figure 11-1. All piles are equal in diameter and length where pile diameter is 0.5 [m] while pile length is 10 [m]. Spacing between piles is 1.6 [m]. It is required to analyze the pile cap as rigid free-standing raft due to a vertical load of P = 8000 [kN] acting on the pile cap with eccentricities  $e_x = 1.4$  [m] and  $e_y = 1.8$  [m] in both *x*- and *y*-directions.





#### **1.2** Soil properties

The subsoil under the raft consists of a layer of silt, 30 [m] thick. The raft lies directly on the ground surface. There is no effect of water table on the raft. The soil is supposed to have the following parameters:

Modulus of elasticity for loading	$E_s$	= 10000	$[kN/m^2]$
Modulus of elasticity for reloading	$W_s$	= 10000	$[kN/m^2]$
Poisson's ratio	$v_s$	= 0.3	[-]

#### **1.3** Mathematical model

In this example, rigid free-standing raft (method 9) is chosen to analyze the raft. This Tutorial manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for the soil models and numerical calculation methods are well documented in the User's Guide of *ELPLA*.

#### 2 Creating the project

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

#### 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation method" wizard appears, Figure 11-2. This wizard will guide you through the steps required to create the project. As shown in Figure 11-2, the first form the wizard is the "Analysis type" form. In this form, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is pile groups, select "Analysis of piled raft" then click "Next" button to go to the next page.

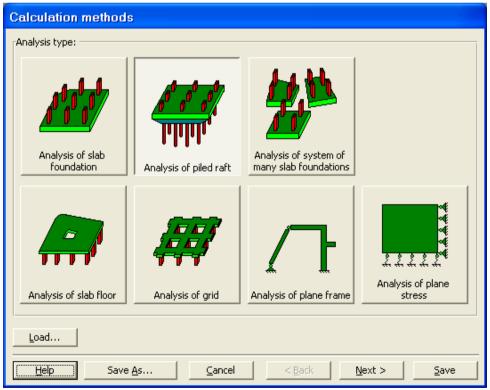


Figure 11-2 "Calculation method" wizard with "Analysis type" form

After clicking "Next" button, the "Calculation methods" form appears, Figure 11-3.

To define the calculation method

- Select the calculation method "9-Rigid free-standing raft"
- Select the subsoil model "Layered soil model"
- Click "Next" button to go to the next form

Calculation methods		
Calculation methods:		
C 1- Linear Contact Pressure (Conventional Method)		
C 2/3- Constant/ Variable Modulus of subgrade Reaction		
C 4- Modification of Modulus of subgrade Reaction by Iteration		
C 5- Isotropic Elastic Half Space		
C 6- Modulus of Compressibility (Iteration)		
C 7- Modulus of Compressibility (Elimination)		
C 8- Rigid piled raft foundation		
9- Rigid free-standing raft		
Subsoil model: C Half Space model C Layered soil model		
Help     Save As     Cancel     < Back     Next >		

Figure 11-3 "Calculation methods" form

The next form is the "System symmetry" (Figure 11-4).

Calculation methods	
System symmetry:	
Unsymmetrical system	
Symmetrical system about x-axis	Double-symmetrical system
Symmetrical system about y-axis	Anti-symmetrical system about x-axis
·	
Help Save As Cancel	< <u>Back</u>

Figure 11-4 "System symmetry" form

In this form choose "Unsymmetrical system" then click "Next" button.

The last form of the wizard assistant contains the "Option" list, Figure 11-5. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other. There isn't any requested choices, so click the "Save" button.

Calculation methods
Options:
□ ✓ Slab with girders
Addtional springs
Supports/ Boundary Conditions
Determining limit depth
Concrete design
<ul> <li>Nonlinear subsoil model</li> <li>Determining displacements in soil</li> <li>Determining stresses in soil</li> <li>Determining strains in soil</li> <li>Influence of neighboring foundations on the slab</li> <li>Influence of the temperature change on raft</li> <li>Influence of additional settlements on raft</li> </ul>
Select All
Nonlinear analysis of piled raft:
R Nonlinear analysis using a hyperbolic function for load-settlement
C Nonlinear analysis using DIN 4014 for load-settlement
Help     Save As     Cancel     < Back

Figure 11-5 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 11-6.

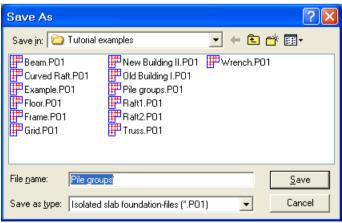


Figure 11-6 "Save as" dialog box

In this dialog box

- Type a file name for the current project in the "File name" edit box. For example type "Pile groups". *ELPLA* will use automatically this file name in all reading and writing processes
- Click "Save" button

*ELPLA* will activate the "Data" menu. In addition, the file name of the current project [Pile groups] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 11-7. In the "Data" menu, the user can enter the remaining data of the project using the same sequence of commands in this menu. The first command in the menu is "Calculation methods", which has been already entered. Therefore, *ELPLA* has put the sign " $\sqrt{}$ " beside this command, Figure 11-7. *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.

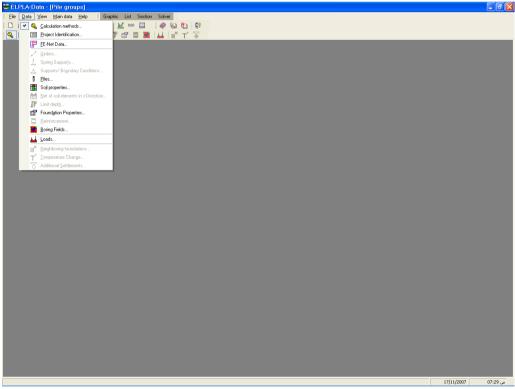


Figure 11-7 *ELPLA-Data* after defining the calculation method

#### 2.2 **Project identification**

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 11-8 appears. In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of pile groups"
- Type the date of the project in the "Date" edit box
- Type "Tutorial Manual" in the "Project" edit box
- Click "Save" button

Project I	dentification				
Project Ide	entification:				
Title	Analysis of pile groups				
Date	17.11.2007				
Project	Tutorial Manual				
<u>S</u> ave					

Figure 11-8 "Project identification" dialog box

## 2.3 FE-Net data

Choose "FE-Net data" command from "Data" menu of *ELPLA-Data*. The "FE-Net generation" wizard appears as shown in Figure 11-9. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 11-9 the first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have regular shapes. For the given problem, the foundation has an irregular shape.

In pile group problems, it is not required to generate a FE-Net for the entire dimension of the raft. Only a net containing pile location is enough. *ELPLA* has different procedures for defining the FE-Net. The easy procedure to define the FE-Net of these pile groups is generating a mesh for the entire area firstly, then removing the unnecessary nodes to get the foundation shape.

FE-Net Generation
Slab type:
Rectangular slab:
Length of rectangular slab L [m] 6,4
Width of rectangular slab B [m] 8
Help     Cancel     < Back     Next >     Einish

Figure 11-9 "FE-Net generation" wizard with "Slab type" form

To generate the FE-Net

- In the "Slab type" options, choose the rectangular slab option
- In the "Rectangular slab" frame, enter the total length and width of the raft in the corresponding edit boxes
- Click "Next" button to go to the next form

After clicking "Next" in "FE-Net generation" wizard, the following "Generation type" form appears, Figure 11-10. *ELPLA* can deal with various type of generations with triangle and/ or rectangular elements. Choose the first type of rectangular elements. Then click "Next" button to go to the next form.

FE-Net Generation	)		
Generation type:			
		× · × · × · × × · × · × · × · ×	· · · · · · · · · · · · · · · · · · ·
Help	Cancel	< <u>B</u> ack <u>N</u> ext >	• <u>Einish</u>

Figure 11-10 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "Grid definition" dialog box in Figure 11-11 appears with default values of constant element size.

In this dialog box

- In "Grid in x-direction" frame, type 4 in the "No. of grid spaces" edit box
- In "Grid in y-direction" frame, type 5 in the "No. of grid spaces" edit box
- Type 1.6 in the "Grid interval" edit box
- Click "OK" button

Cartesian Grid		×
Grids in x-direction:		Qk
No. of grid intervals Grid interval Dx [m]	4 ·	Cancel
Grids in y-direction:		Help
🔽 Constant grid interval		
No. of grid intervals	5	
Grid interval Dy [m]	1.60	

Figure 11-11 "Grid definition" form

*ELPLA* will generate a FE-Net for a rectangular area of 6.4 [m] length and 8 [m] width with square elements of 1.6 [m] each side. The following embedded program in Figure 11-12 appears with the generated net.

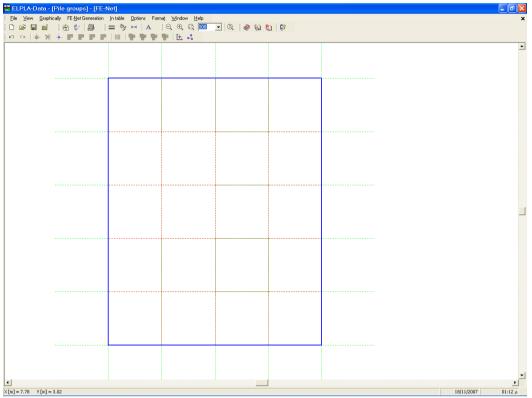


Figure 11-12 FE-Net of a rectangular slab on the screen

#### **Deleting nodes from the FE-Net**

To select the unnecessary nodes that are required to be removed from the net, first choose "Select nodes" command from "Graphically" menu. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair. The command "Remove nodes" in the menu "Graphically" will be enabled, indicating the mode in which is being operated. Next, select the required nodes by clicking on each node individually or selecting a group of nodes as shown in Figure 11-13. To remove the selected nodes, choose "Remove nodes" command from the "Graphically" menu. The action of this command is indicated in Figure 11-14.

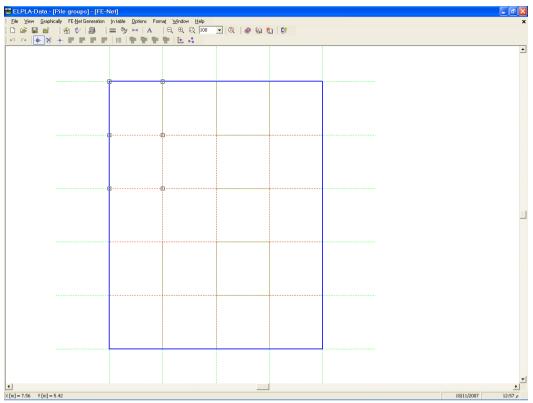


Figure 11-13 Generated FE-Net after selecting the unnecessary nodes

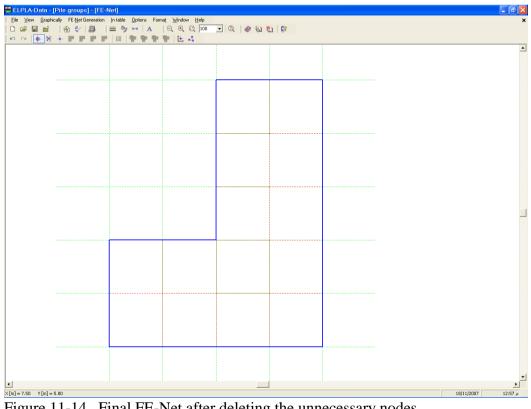


Figure 11-14 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save FE-Net" command from "File" menu in Figure 11-14 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 11-14 to close the "FE-Net" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "FE-Net data" command in the "Data" menu of *ELPLA-Data*.

#### 2.4 Element length of pile

To define the element length of pile, choose "Preferences" command from "Main data" menu of *ELPLA-Data*. The "Preferences" dialog box in Figure 11-15 appears.

In this dialog box

- Type 0.5 in the "Element length of pile" edit box
- Click "Save" button

Preferences			X
FE-Net preferences:			
Check element overlaps			
🔽 Check element size			
Minimum distance between nodes	Sm	[m]	0.05
Element length of pile	Dz	[m]	0.5
Calculation preferences: The Internal forces are determined at: the element centers and then distribute the element nodes	d to the	element n	odes
<u>S</u> ave <u>C</u> ancel			Help

Figure 11-15 "Preferences" dialog box

#### 2.5 Piles

To define piles, choose "Piles" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 11-16 appears.

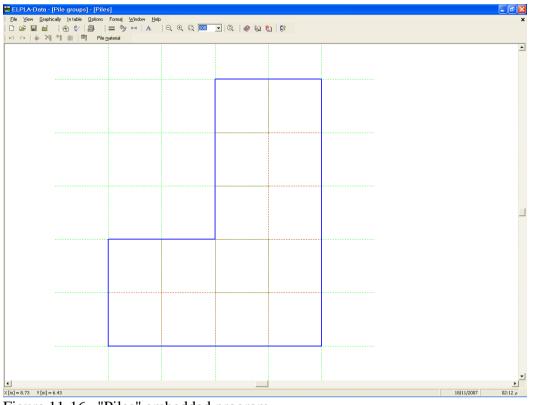


Figure 11-16 "Piles" embedded program

#### **Pile groups**

To define pile groups those have the same diameter and length, choose "Pile groups" command from "In Table" menu in Figure 11-16. When the "Pile groups" command is chosen, the following table in Figure 11-17 appears allowing you to define the pile diameter and length.

1	Definir	ig pile grou	ps	X
	Group No.	Pile diameter D	Pile length L	<u>O</u> k
		[m]	[m]	⊆ancel
	1	0.5	10	Insert
				inserc
				⊆ору
				Delete
				New
				Help
				<u>E</u> xcel

Figure 11-17 Defining pile groups

#### **Pile material**

To define pile material, choose "Pile material" command from "In Table" menu in Figure 11-16. When the "Pile material" command is chosen, the following dialog box in Figure 11-18 appears allowing you to define the pile material. Type 0 in the "Unit weight of the pile concrete" edit box to neglect the pile weight.

Pile material			X
Pile material:			
Unit weight of pile concrete	Gp	[kN/m3]	0
Modulus of elasticity of pile	Ep	[kN/m2]	3E+07
<u>Q</u> k	<u>C</u> ancel		Help

Figure 11-18 "Defining element groups" dialog box

## **Pile location**

To define pile locations on the net

- Choose "Select nodes" command from "Graphically" menu in Figure 11-16
- Select nodes that have piles as shown in Figure 11-19
- Choose "Add piles" command from "Graphically" menu in Figure 11-19. The "Defining pile groups" dialog box in Figure 11-20 appears. In this dialog box click "OK" button

😤 ELPLA-Data - [Pile groups] - [Pile					
Ele ⊻iew Graphically In table Options	Format <u>W</u> indow <u>H</u> elp				×
□ 🛩 🔜 🖆 🔺 🚳 📗	= 🦻 🛏 🛛 A	ର୍ ବ୍ ପ୍ 🧰	🗉 🔍 🥔 🗞 🕻	D 67	
မက 🖶 🖓 ᡟ 🏢 📑 Pileg	naterial				
					<b>_</b>
		6	<del>} (</del>		p
			1	8	
					r
			J		
	1	3	3(	3	9
	36	)6	3(	81	p
	bd	<del>. с</del>	s (		
					<u>-</u>
					10111/0007
×[m]=9.01 ¥[m]=0.20		<u> </u>		• •	18/11/2007 02:19 p

Figure 11-19 Selection of nodes that have piles

Defining pile groups	
Pile group No.	[-] 1 🗸
Cancel	Help

Figure 11-20 "Defining pile groups" dialog box

After you have completed the definition of the piles, the screen should look like the following Figure 11-21.

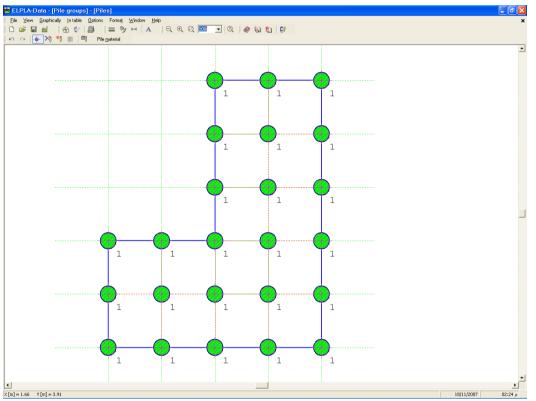


Figure 11-21 Piles on the screen

After entering all data of piles, do the following two steps:

- Choose "Save piles" command from "File" menu in Figure 11-21 to save the data of piles
- Choose "Close piles" command from "File" menu in Figure 11-21 to close the *ELPLA-Piles* sub program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Piles" command in the "Data" menu of *ELPLA-Data*.

#### 2.6 Soil properties

To define the soil properties, choose "Soil properties" command from "Data" menu of *ELPLA-Data*. The following sub program in Figure 11-22 appears with a default-boring log.

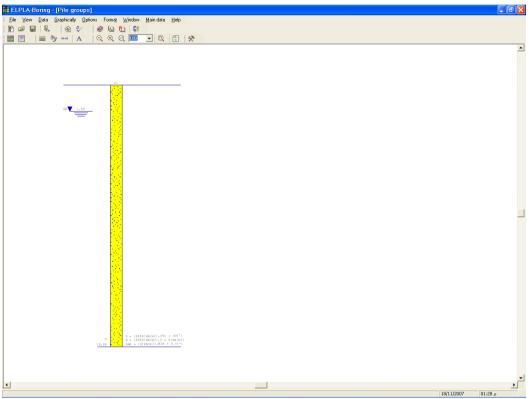


Figure 11-22 *ELPLA-Boring* sub program with a default-boring log

# Modifying data of boring log graphically

In *ELPLA*, the boring log can be defined or modified graphically, which makes the definition of the boring log very easy. By double-clicking the left mouse button on a specified screen position, the user can define or modify the soil data and input parameters graphically.

To enter the geotechnical data of the soil layer

- Double-click on the geotechnical data of the soil layer. The corresponding dialog box appears allowing you to modify the geotechnical data of the soil layer, Figure 11-23
- In the dialog group box "Geotechnical data of the layer" in Figure 11-23, define the geotechnical data of the soil layer as follows:

$E_s$	$= 10\ 000$	$[kN/m^2]$
$W_s$	$= 10\ 000$	$[kN/m^2]$
$v_s$	= 0.3	[-]

The values *Es* and *Ws* are the same because the effect of reloading on the soil is not required. The unit weight of the soil is used to determine the overburden pressure  $q_v$  [kN/m<sup>2</sup>] due to the removed soil, which is equal to  $\gamma_s * d_f$ . In the current example  $d_f = 0.0$ , which means the unit weight of the soil is not required. However, the unit weight of the soil is entered by the default value. Also, the angle of internal friction  $\varphi$  and the cohesion *c* of the soil are not required because the selected type of the analysis is linear analysis. Therefore, the user can let the default values of the internal friction and the cohesion. Next, click "OK" button.

Soil d	ata					×
Layer	log No. 1 from 1 r No. 1 from 1 la otechnical data o	yers:				
So	oil properties are	e defined by Moo	lulus of Elastic	ity E		•
E	[kN/m2	] 10000	Fhi	[°]	30	-
W	[kN/m2	10000	c	[kN/m2]	0	
Ga	m [kN/m3	] [18]	Nue	[-]	0.3	
	<u>o</u> k	<u>C</u> ancel				

Figure 11-23 "Geotechnical data of the soil layer" dialog box

To define the soil type and color for the layer

- Double-click on the soil symbol of the soil layer. The corresponding dialog box appears allowing you to modify the soil symbols of that layer, Figure 11-24
- Select "U, Silt" as the soil type in the "Main soil type 1" combo box in the dialog group box "Soil and rock symbols" in Figure 11-24. The color of the silt and a short text "U" according to the German specification code DIN 4023 will be automatically created
- Click "OK" button

Soil data		×				
Boring log No. 1 from Layer No. 1 from 1 k Soil and rock symbol	ayers:	1				
Main soil type 1	U, Silt 🔽					
Main soil type 2	-, No symbole 💌					
submain soil 1	-, No symbole 📃					
submain soil 2	-, No symbole 💌					
Color	ol, olive 🔽					
Short text	h					
	1					
<u>k</u>	⊆ancel					

Figure 11-24 "Soil and rock symbols" dialog box

To modify a layer depth

- Double-click on the layer depth. The corresponding edit box appears allowing you to modify the layer depth under the ground surface, Figure 11-25
- Type 30 in the "Layer depth under the ground surface" edit box
- Click "OK" button

Soil data	×
Boring log No. 1 from 1 boring logs: Layer No. 1 from 1 layers: Layer depth under the ground surface [m]	] [30
<u>Ok</u> <u>C</u> ancel	

Figure 11-25 "Layer depth under the ground surface" edit box

To modify the groundwater depth under the ground surface

- Double-click on the groundwater level. The corresponding edit box appears allowing you to modify the groundwater depth under the ground surface, Figure 11-26. To neglect the uplift pressure on the raft, groundwater level is chosen at anywhere under the raft basement
- Type 30 in the "Groundwater depth under the ground surface" edit box
- Click "OK" button

Groundwater		×
Groundwater:	th under the ground surface [m] $30$	
Qk	Cancel	

Figure 11-26 "Groundwater depth under the ground surface" edit box

To modify the label of a boring log

- Double-click on the label of the boring log. The corresponding edit box appears allowing you to modify the label of the boring log, Figure 11-27
- Type B1 in the edit box of Figure 11-27
- Press "Enter" key to consider the text

B1

Figure 11-27 "Label of the boring log" edit box

After the user has completed the definition of all soil properties and parameters, the screen should look like the following Figure 11-28.

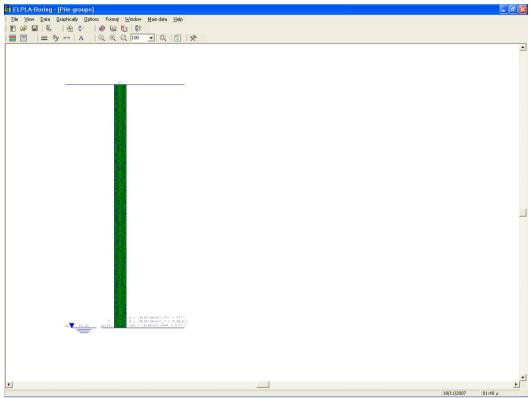


Figure 11-28 Boring log on the screen

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

- Choose "Save boring logs" command from "File" menu in Figure 11-28 to save the data of boring log
- Choose "Close boring logs" command from "File" menu in Figure 11-28 to close the *ELPLA-Boring* sub program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Soil Properties" command in the "Data" menu of *ELPLA-Data*.

#### 2.7 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 11-29 appears with default foundation properties. In the current example, it is required to define only the unit weight of the foundation. Any other data corresponding to foundation properties in the program menus are not required. Therefore, the user can take these data from the default foundation properties.

View Graphica	ally In table Foundatio	ndation Properties on Properties Options	Forma <u>t W</u> indow <u>H</u> elp			
ê 🖬 🖬	2  ♥   2          X □	= ∛y ⊷   A	ର୍ ବ୍ ଢ୍ 🎹	• Q 🔌 😡 (	b (?	
*   ./ 🕮	III   Jc III 🗱		1			
				1	1	
				1	1	
				1	1	
		1	1	1	1	
		1	1	1	1	
			<u> </u>	±	-	
					Í	11/2007 01

Figure 11-29 "Foundation properties" embedded program

To enter the unit weight of the foundation

- Choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 11-29. The following dialog box in Figure 11-30 with a default unit weight of 25 [kN/m<sup>3</sup>] appears. Type 0 to neglect the foundation weight in the edit box "Unit weight of the foundation"
- Click "OK" button

Unit weight of the foundation	
Unit weight of the foundation Gb [kN/m3]	
<u>Qk</u> Newancel	Help

Figure 11-30 "Unit weight of the foundation" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 11-29 to save the foundation properties
- Choose "Close foundation properties" command to close the "Foundation properties" embedded program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Foundation Properties" command in the "Data" menu of *ELPLA-Data*.

#### 2.8 Loads

To define the loads, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 11-31 appears.

🗄 ELPLA-Date	a - [P	ile gro	ups]·	- [Loac	s]										
Elle View Gr	aphica	ly _in ta	ble <u>L</u>	sing form	ula Option	ns Format	₩indow	Help							×
🗅 😂 🗏 🕯	ŕ.	<b>a</b> (	1		= 🦻 '	A	Q	Q Q 🛙	• 00	] 🔍 🤌	60 🖞	1   💱			
ର ର 🔺	* 1	ŧ m	ter i	a 74	X T	🎟   📥									
															<u> </u>
									····						
											+				
															1
							<del> </del>								
															-
X[m] = 9.16 Y[m]	1-6	13												18/11/2007	01:01 /
v [n] = 2.10 T [0	-j = 0.1	10												10/11/2007	01.01 0

Figure 11-31 "Loads" embedded program

To enter the concentrated load choose "Point loads" command from "Graphically" menu in the window of Figure 11-31. When "Point loads" command is chosen, the cursor is changed from an arrow to a cross hair. Then, the load can be defined by clicking on the screen (grid). When the screen is clicked, the following dialog box in Figure 11-32 appears with the load value and coordinates.

In this dialog box

- Type 8000 in the "Load" edit box
- Type 5.2 in the "*x*-position" edit box
- Type 5.2 in the "y-position" edit box
- Click "OK" button

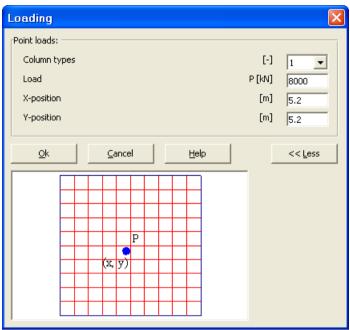


Figure 11-32 "Point loads *P*" dialog box

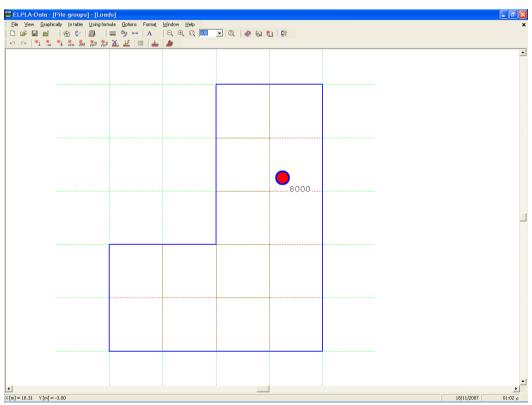


Figure 11-33 Loads on the screen

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

- Choose "Save loads" command from "File" menu in Figure 11-33 to save the load data
- Choose "Close loads" command from "File" menu in Figure 11-33 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Loads" command in the "Data" menu of *ELPLA-Data*.

Creating the project of the pile groups is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

## 3 Carrying out the calculations

#### 3.1 Starting *ELPLA-Solver*

To analyze the problem leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window appears, Figure 11-34.

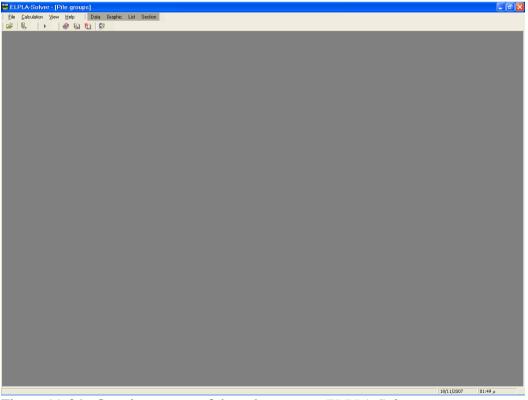


Figure 11-34 Opening screen of the sub program ELPLA-Solver

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. The "Calculation" menu contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Analysis of rigid pile groups
- Determining deformation, internal forces, contact pressures

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

#### **3.2** Carrying out all computations

To carry out all computations in one time, choose "Computation of all" command from "Calculation" menu in *ELPLA-Solver* window. The analysis progress menu in Figure 11-35 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

Determining flexibility coefficients of the soil
Assembling the flexibility matrix!
Time remaining = 00:00:07 I = 102 from 504 steps

Figure 11-35 Analysis progress menu

Once the analysis is complete, a check menu of the solution appears, Figure 11-36. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

[kN] = 8000
[kN] = 8000
[kN.m] = 15086
[kN.m] = 15086
[kN.m] = 10514
[kN.m] = 10514

Figure 11-36 Menu "Check of the solution"

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

## 4 Viewing data and results

*ELPLA* can view and print a wide variety of results in graphics, diagrams or tables through the three sub programs *ELPLA-Graphic*, *ELPLA-Section* and *ELPLA-List*. Data can also be viewed again and printed by the sub programs *ELPLA-Graphic* and *ELPLA-List*. Note that *ELPLA-Data* is used only to define and view the data of the problem. *ELPLA-Graphic* is used to print data graphically while *ELPLA-List* is used to print data numerically.

## 4.1 Viewing result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, leave *ELPLA-Solver* to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window. *ELPLA-Graphic* window appears, Figure 11-37. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

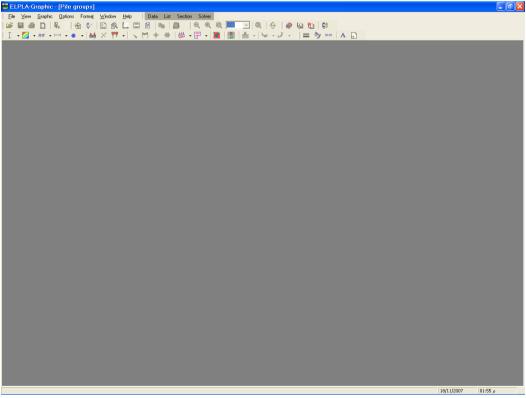


Figure 11-37 Opening screen of the sub program ELPLA-Graphic

To view the results for pile groups as circular diagrams

- Choose "Results as Circular Diagrams" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 11-38 appears
- In "Results as circular diagrams" option box, select "Settlements *s*" as a sample for the results to be displayed
- Click "OK" button

Settlements are now displayed as circular diagrams for piles as shown in Figure 11-39.

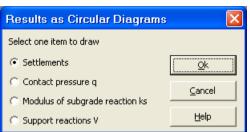


Figure 11-38 "Results as circular diagrams" option box

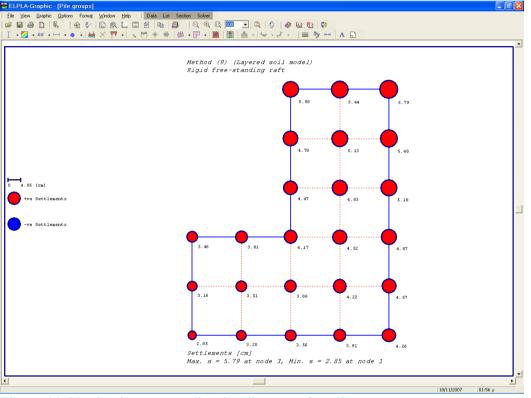


Figure 11-39 Settlements as circular diagrams for piles

# 4.2 Listing data and result in tables

To list tables of data and results, switch to *ELPLA-List*. This is done by clicking on "List" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Section* window. *ELPLA-List* window appears, Figure 11-40.

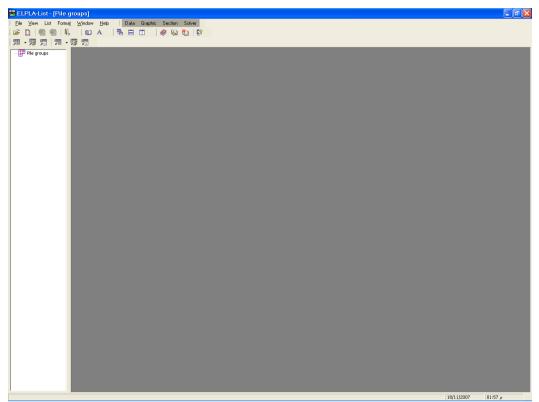


Figure 11-40 Opening screen of the sub program ELPLA-List

The function of *ELPLA-List* is listing and printing data and results in tables. The data and results can be exported to other Windows applications to prepare reports or add further information. *ELPLA-List* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-List* window.

Only one command of the "List" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Format" and "Window" menus, which are used to define the preferences of the tables such as page format, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

To list results in a table

- Choose "Display tables of results" command from "List" menu. The following option box in Figure 11-41 appears
- In this option box, select "Pile results" as an example for the results to be listed in a table
- Click "OK" button. The pile results are now listed on the screen (Figure 11-42)
- Choose "Send to Word" from "File" menu if you wish to export the text to the MS Word application, Figure 11-43

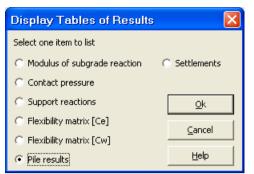


Figure 11-41 "Display tables of results" option box

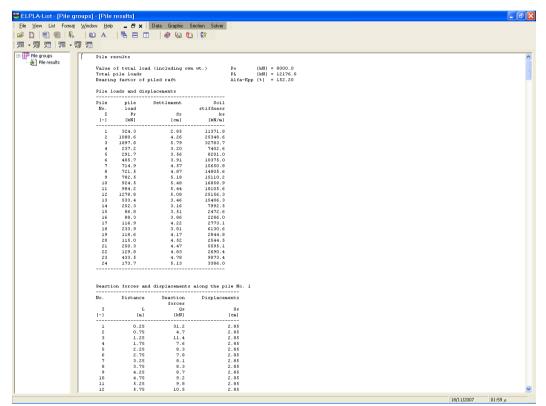
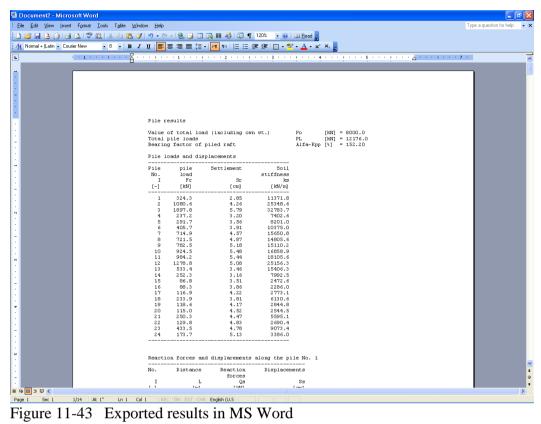


Figure 11-42 List of settlement results



# 5 Index

# A

# С

Calculation	23, 24
Calculation methods	4, 5, 7
Check of the solution	
Computation of all	
Contact pressures	

# D

# E

ELPLA-Data 4, 7, 8, 12, 15, 16	5, 19, 20, 25
ELPLA-Graphic	
ELPLA-List	25, 26, 27
ELPLA-Solver	23, 24, 25

# F

FE-Net	3
File 4, 7, 15, 19, 24, 25, 27	7
Flexibility coefficients	ł
Foundation	)
Foundation properties 19, 20	)

# G

Generation type	)
Graphic	

# Ι

Internal forces 1	7,	24
-------------------	----	----

# L

List		26,	27
Load vector			24
Loads	21,	22,	23

# M

Modulus of elasticity for loading	3
Modulus of elasticity for reloading	3

# N

Numerical calculation	4	4
-----------------------	---	---

# 0

# P

Point loads	
Poisson's ratio	
Project	4, 7, 23, 24
Project identification	7

# R

Raft	3,	4,	5,	8,	18
Remove nodes	••••	••••	••••		10

# S

Save6, 7, 12, 15, 20, 2 Save as	
Save as	
Select nodes 10, 1	4
Settlement	26
Soil	
Soil properties	19
Soil stiffness matrix	
System symmetry	5

# T

## $\overline{U}$

Unit weight	14, 17, 19, 20
Unsymmetrical system	6
User's Guide	

Example 12

Analysis of piled raft

# Contents

# Page

1	Desc	cription of the problem	3
	1.1		3
	1.2	Raft material and thickness	3
	1.3		3
	1.4	Soil properties	4
	1.5	Mathematical model	4
2	Crea	ting the project	4
	2.1	Calculation method	4
	2.2	Project identification	8
	2.3	FE-Net data	9
	2.4	Element length of pile	11
	2.5	Piles	12
	2.6	Soil properties	15
	2.7	Foundation properties	19
	2.8	Loads	22
3	Carr	ying out the calculations	24
	3.1	Starting <i>ELPLA-Solver</i>	24
	3.2	Carrying out all computations	25
4	View	ving data and results	26
	4.1	Viewing result graphics	26
	4.2	Plot a diagram of the results	28
	4.3	Listing data and result in tables	31
5	Inde	X	35

## **1** Description of the problem

An example of piled raft is selected to illustrate some of the essential features of *ELPLA* for analyzing piled raft.

## 1.1 Load and dimensions

A raft on 42 bored piles is considered as shown in Figure 12-1. All piles are equal in diameter and length where pile diameter is 0.9 [m] while pile length is 20 [m]. Spacing between piles is 3.5 [m]. It is required to analyze the raft with piles as piled raft foundation due to a uniform load of p = 150 [kN/m] acting on the raft.

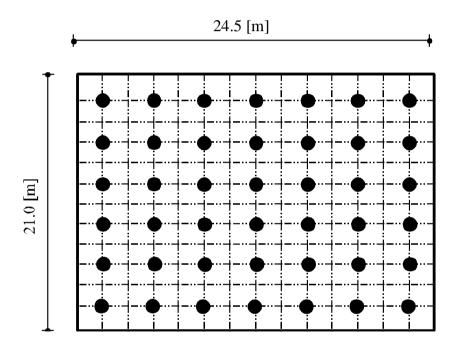


Figure 12-1 Plan with piled raft geometry

## 1.2 Raft material and thickness

Raft material and thickness are supposed to have the following parameters:

Raft thickness	d	= 1.5	[m]
Young's modulus	$E_b$	$= 3.4 * 10^7$	$[kN/m^2]$
Unit weight	$\gamma_b$	= 25	$[kN/m^3]$
Poisson's ratio	$\mathbf{v}_b$	= 0.2	[-]

## **1.3** Pile material

The pile material is supposed to have the following parameters:

Young's modulus	$E_b$	$= 2.35 * 10^7$	$[kN/m^2]$
Unit weight	$\gamma_b$	= 25	$[kN/m^3]$

## 1.4 Soil properties

The subsoil under the raft consists of a layer of silt with 50 [m] thickness, the water table level is 2 [m] below the ground surface. Foundation level is also 2 [m] below the ground surface. The soil is supposed to have the following parameters:

Modulus of elasticity for loading	$E_s$	= 10000	$[kN/m^2]$
Modulus of elasticity for reloading	$W_s$	= 10000	$[kN/m^2]$
Unit weight	$\gamma_s$	= 18	[kN/m <sup>3</sup> ]
Poisson's ratio	$v_s$	= 0.3	[-]

#### 1.5 Mathematical model

In this example, the Modulus of compressibility method (method 7) is chosen to analyze the piled raft. This Tutorial manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for the soil models and numerical calculation methods are well documented in the User's Guide of *ELPLA*.

## 2 Creating the project

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

#### 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation method" wizard appears, Figure 12-2. This wizard will guide you through the steps required to create the project. As shown in Figure 12-2, the first form of the wizard is the "Analysis type" form. In this form, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is piled raft, select "Analysis of piled raft" then click "Next" button to go to the next page.

Calculation methods Analysis type:			
Analysis of slab foundation	Analysis of piled raft	Analysis of system of many slab foundations	
Analysis of slab floor	Analysis of grid	Analysis of plane frame	Analysis of plane stress
Load	As <u>C</u> ancel		ext > <u>S</u> ave

Figure 12-2 "Calculation method" wizard with "Analysis type" form

After clicking "Next" button, the "Calculation methods" form appears, Figure 12-3. To define the calculation method

- Select the calculation method "7-Modulus of compressibility (Elimination)"
- Click "Next" button to go to the next form

Calculation methods
Calculation methods:
🔿 1- Linear Contact Pressure (Conventional Method)
C 2/3- Constant/ Variable Modulus of subgrade Reaction
C 4- Modification of Modulus of subgrade Reaction by Iteration
C 5- Isotropic Elastic Half Space
C 6- Modulus of Compressibility (Iteration)
7- Modulus of Compressibility (Elimination)
C 8- Rigid piled raft foundation
C 9- Rigid free-standing raft
Determining modulus of subgrade reaction: Modulus is calculated from Half Space Modulus is calculated from soil layers Modulus is defined by the user
Help     Save As     Cancel     < Back     Next >     Save

Figure 12-3 "Calculation methods" form

The next form is the "System symmetry" (Figure 12-4). In this form choose "Unsymmetrical system" then click "Next" button.

Calculation methods	
System symmetry:	
Unsymmetrical system	
	A/A 1
Symmetrical system about x-axis	Double-symmetrical system
Symmetrical system about y-axis	Anti-symmetrical system about x-axis
Help Save As Cancel	< Back <u>Next &gt; S</u> ave

Figure 12-4 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 12-5. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other. There isn't any requested choices, so click the "Save" button.

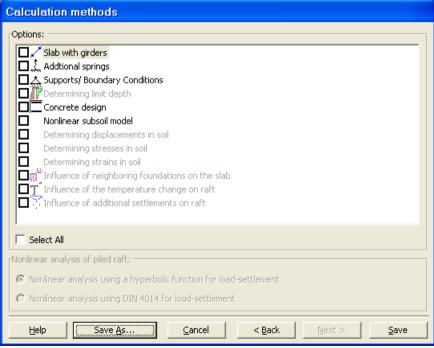


Figure 12-5 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 12-6.

In this dialog box

- Type a file name for the current project in the "File name" edit box. For example type "Piled raft". *ELPLA* will use automatically this file name in all reading and writing processes
- Click "Save" button

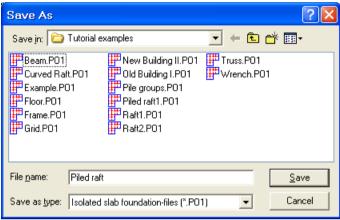


Figure 12-6 "Save as" dialog box

*ELPLA* will activate the "Data" menu. In addition, the file name of the current project [Piled raft] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 12-7. In the "Data" menu, the user can enter the remaining data of the project using the same sequence of commands in this menu. The first command in the menu is "Calculation methods", which has been already entered. Therefore, *ELPLA* has put the sign " $\sqrt{}$ " beside this command, Figure 12-7. *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.

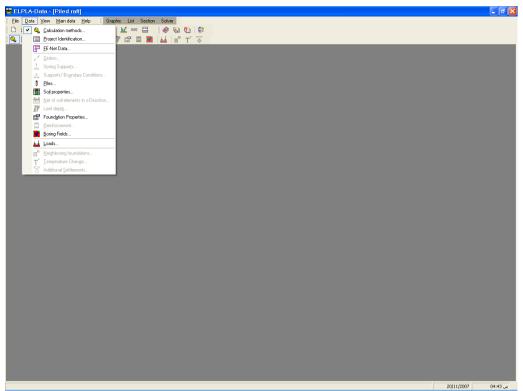


Figure 12-7 *ELPLA-Data* after defining the calculation method

# 2.2 Project identification

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 12-8 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of piled raft"
- Type the date of the project in the "Date" edit box
- Type "Tutorial Manual" in the "Project" edit box
- Click "Save" button

Project I	dentification 🛛 🛛 🗙
Project Ide	ntification:
Title	Analysis of piled raft
Date	20.11.2007
Project	Tutorial Manual
<u>S</u> ave	Cancel Help Load Save As

Figure 12-8 "Project identification" dialog box

#### 2.3 FE-Net data

Choose "FE-Net data" command from "Data" menu of *ELPLA-Data*. The "FE-Net generation" wizard appears as shown in Figure 12-9. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 12-9 the first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have regular shapes. For the given problem, the foundation has a rectangular shape.

FE-Net Generation	
Slab type:	
	0
Rectangular slab:	
Length of rectangular slab	24.5
Width of rectangular slab B [m]	21
Help Cancel < Back Next >	Einish

Figure 12-9 "FE-Net generation" wizard with "Slab type" form

To generate the FE-Net

- In the "Slab type" options, choose the rectangular slab option
- In the "Rectangular slab" frame, enter the total length and width of the raft in the corresponding edit boxes
- Click "Next" button to go to the next form

After clicking "Next" in "FE-Net generation" wizard, the following "Generation type" form appears, Figure 12-10. *ELPLA* can deal with various type of generations with triangle and/ or rectangular elements. Choose the first type of rectangular elements. Then click "Next" button to go to the next form.

FE-Net Generation			
Generation type:	<pre>/ / / / / / / / / / / / / / / / / / /</pre>	× × × × × × × × × × × × × × × × × × ×	+ + + + + + + + + + + + + + + + + + +
Help	Cancel	< <u>B</u> ack <u>N</u> ext >	- <u>F</u> inish

Figure 12-10 "Generation type" form

After clicking "Next" button in "Generation type" form, the following "Grid definition" dialog box in Figure 12-11 appears with default values of constant element size.

In this dialog box

- In "Grid in x-direction" frame, type 14 in the "No. of grid spaces" edit box
- In "Grid in y-direction" frame, type 12 in the "No. of grid spaces" edit box
- Type 1.75 in the "Grid interval" edit box
- Click "OK" button

FE-Net Generation	
Grid definition:	
Grids in x-direction:	
🔽 Constant grid interval	
No. of grid intervals	14 •
Grid interval Dx [m]	1.75
Grids in y-direction: Constant grid interval No. of grid intervals Grid interval Dy [m]	12 • 1.75
<u>H</u> elp <u>C</u> anc	tel < Back Mext > Finish

Figure 12-11 "Grid definition" form

*ELPLA* will generate a FE-Net for a rectangular area of 24.5 [m] length and 21 [m] width with square elements of 1.75 [m] each side. The following embedded program in Figure 12-12 appears with the generated net.

w <u>G</u>	iraphically	FE Net	Generation	įn table	☐ptions	Format	₩indow	Help											
	ei	∕∌ ∜	2	= 🎭	юн   Д	A [6	ર્ ૨ ૯	100	• 0	1 🧶 🤅	ا 🕒 🛯	<b>C</b> ?							
÷	* +	P P	12.2	##	P P	P P	1 🗄 🛃												
																İ			
									+										
															<u> </u>				
																·····			
			+				+		+						+				
									ļ					ļ		ļ			
									1	-					1				
																1			
			+	+			+	·	+	÷	·	·	+	+	+	+			
																1			
																· · · · · ·			
																1			
																1			
							-												
																1			
		<u> </u>	-	-		-	-		-	-		-	-	-	-	<b>.</b>			

Figure 12-12 FE-Net of a rectangular slab on the screen

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

- Choose "Save FE-Net" command from "File" menu in Figure 12-12 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 12-12 to close the "FE-Net" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "FE-Net data" command in the "Data" menu of *ELPLA-Data*.

#### 2.4 Element length of pile

To define the element length of pile, choose "Preferences" command from "Main data" menu of *ELPLA-Data*. The "Preferences" dialog box in Figure 12-13 appears.

In this dialog box

- Type 2 in the "Element length of pile" edit box
- Click "Save" button

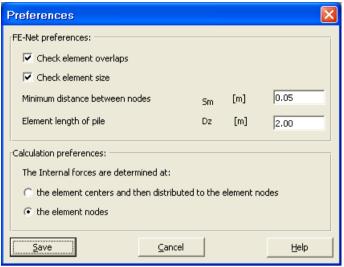


Figure 12-13 "Preferences" dialog box

## 2.5 Piles

To define piles, choose "Piles" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 12-14 appears.

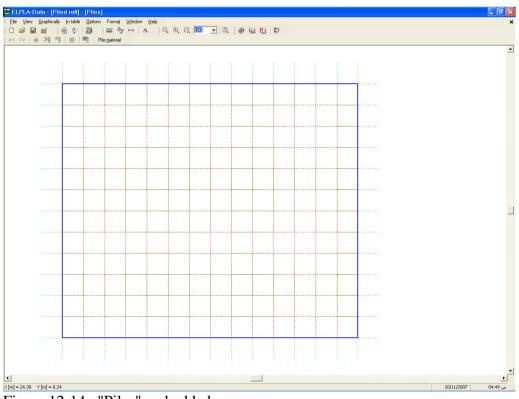


Figure 12-14 "Piles" embedded program

#### **Pile groups**

To define pile groups that have the same diameter and length, choose "Pile groups" command from "In Table" menu in Figure 12-14. When the "Pile" groups" command is chosen, the following table in Figure 12-15 appears allowing you to define the pile diameter and length.

I	Definir	ig pile group	ps	X	]
	Group No.	Pile diameter D	Pile length	Qk	
		[m]	[m]	Cancel	
	1	0.9	201	Insert	
				<u><u>C</u>opy</u>	
				Delete	
				New	
				Help	
				<u>E</u> xcel	

Figure 12-15 Defining pile groups

#### Pile material

To define pile material, choose "Pile material" command from "In Table" menu in Figure 12-14. When the "Pile material" command is chosen, the following dialog box in Figure 12-16 appears.

In this dialog box

- Type 25 in the "Unit weight of the pile concrete" edit box
- Type 2.35E+07 in the "Modulus of elasticity of pile" edit box
- Click "OK" button

Pile material				×
Pile material:				
Unit weight of pile concrete	Gp	[kN/m3]	25	
Modulus of elasticity of pile	Ep	[kN/m2]	2.35E+07	
	<u>C</u> ancel		Help	

Figure 12-16 "Defining element groups" dialog box

#### **Pile location**

To define pile locations on the net

- Choose "Select nodes" command from "Graphically" menu in Figure 12-14
- Select nodes those have piles as shown in Figure 12-17
- Choose "Add piles" command from "Graphically" menu in Figure 12-17. The "Defining pile groups" dialog box in Figure 12-18 appears. In this dialog box click "OK" button

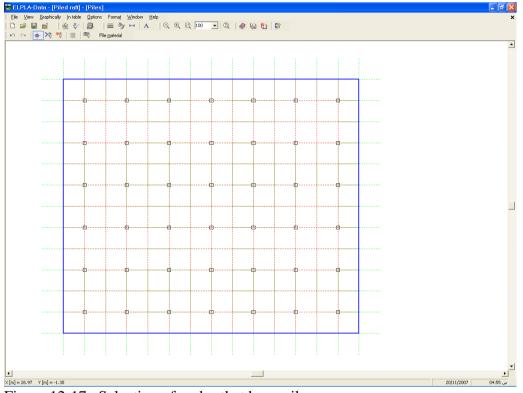


Figure 12-17 Selection of nodes that have piles

Defining pile groups		×
Pile group No.	[-]	1
		Help

Figure 12-18 "Defining pile groups" dialog box

After you have completed the definition of the piles, the screen should look like the following Figure 12-19.

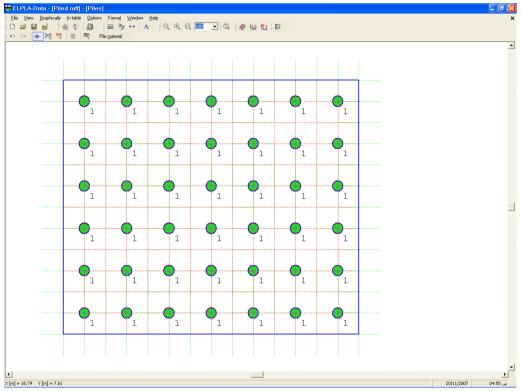


Figure 12-19 Piles on the screen

After entering all data of piles, do the following two steps:

- Choose "Save piles" command from "File" menu in Figure 12-19 to save the data of piles
- Choose "Close piles" command from "File" menu in Figure 12-19 to close the *ELPLA-Piles* sub program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Piles" command in the "Data" menu of *ELPLA-Data*.

## 2.6 Soil properties

To define the soil properties, choose "Soil properties" command from "Data" menu of *ELPLA-Data*. The following sub program in Figure 12-20 appears with a default-boring log.

ELPLA-Boring - [Curved Raft]		∎₽⊠
<u>File View Data Graphically Options Format Window Maindata H</u> elp		
📓 🗏 🚔 🦘 🗛 🔍 🍳 🚥 🚽 🔍 🔁 🛠		
		<b>A</b>
91		
R = 10 000 000 (val. This = 301 <sup>11</sup> )		
$ \begin{array}{c} a \\ a \\ b \\ b \\ c \\ b \\ c \\ c \\ b \\ c \\ c \\ c$		
		<b>_</b>
	30/10/2007	08:33 p
	30/10/2007	00:00 0

Figure 12-20 *ELPLA-Boring* sub program with a default-boring log

To enter the soil properties for the boring log of the current example

- Choose "Soil data" command from "Data" menu in the window of Figure 12-20. The following dialog box in Figure 12-21 appears with default-boring log data.

Soil data		×
Boring log No. 1 from 1 boring logs: Layer No. 1 from 1 layers: Soil and rock symbols: Main soil type 1 G, Gravel Main soil type 2 -, No symbole submain soil 1 -, No symbole submain soil 2 -, No symbole	Geotechnical data of the layer: Soil properties are defined by Modulus of Elasticity E E [kN/m2] 10000 Fhi [°] 30 W [kN/m2] 10000 c [kN/m2] 0	-
Color ge, yellow  Short text G	Gam     [kN/m3]     18     Nue     [-]     0.3       Layer depth under the ground surface     [m]     10.00	
Copy Boring log Delete Boring	X-coordinate of boring [m] 0.00 Y-coordinate of boring [m] 0.00 Boring Log Label B1	
Cancel	New	Help

Figure 12-21 "Soil data" dialog box with default-boring log data

In the "Geotechnical data of the layer" dialog group box in Figure 12-21, define the geotechnical data of the first soil layer of the boring log as follows:

$E_s$	= 10000	$[kN/m^2]$
$W_s$	= 10000	$[kN/m^2]$
$\gamma_s$	= 18	$[kN/m^3]$
$v_s$	= 0.3	[-]

In the current example, the angle of internal friction  $\varphi$  and the cohesion *c* of the soil are not required because the selected type of the analysis is linear analysis. Therefore, the user can let the default values of the internal friction and the cohesion.

φs	= 30	[°]
С	= 0	$[kN/m^2]$

Due to the presence of the ground water, the soil above the ground water level has a deferential unit weight from the soil under that level. Therefore, the layer depth of the first layer is taken to be 2 [m], which is equal to the ground water level. Now, type this value in "Layer depth under the ground surface" edit box.

In order to draw the soil layers by different symbols according to the German Standard DIN 4023, the soil type and color for each layer must be defined.

To define the soil type and color for the first layer, select "U, Silt" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box, Figure 12-21. The color of the silt according to the German Standard DIN 4023 will be automatically created. The user can change this color. Also, a short text "U" will be automatically created for the silt.

To enter the second layer

- Click "Layer copy" button in Figure 12-21. A layer that has the same properties of the first layer will be copied
- Use the vertical scrollbar to move to the second soil layer.
   Layer No. will be typed automatically at the upper-left corner of the main dialog box of soil layers as a head title
- Change the value of the unit weight of the soil for the second soil layer from 18  $[kN/m^3]$  to 9  $[kN/m^3]$
- Change the value of the layer depth under the ground surface from 2 [m] to 50 [m]

Note that the unit weight of the soil is used to determine the overburden pressure  $q_v$  [kN/m<sup>2</sup>] due to the removed soil, which is equal to  $\gamma_s * d_f$ . This means that the unit weight of the soil under the foundation depth  $d_f$  is not required. However, the unit weight of the soil under the foundation depth for all soil layers is entered by the value 9 [kN/m<sup>3</sup>].

After editing the geotechnical data, the boring coordinates and labels which describe the boring will be entered.

To enter the boring coordinates and label

- Type 0 for *x*-coordinate in "*x*-coordinate of boring log [m]" edit box
- Type 0 for *y*-coordinate in "*y*-coordinate of boring log [m]" edit box
- Type B1 as a label name for the first boring in "Label of boring log" edit box

To enter the main soil data for all layers, choose "Main soil data" command from "Data" menu in Figure 12-20. The following dialog box in Figure 12-22 appears with default main soil data. The main soil data for the current example, which are required to be defined, are the settlement reduction factor  $\alpha$  [-] and the groundwater depth under the ground surface  $G_w$  [m]. Any other data corresponding to main soil data are not required in this example. Therefore, the user can take these data from the default soil properties.

In the dialog box of Figure 12-22, enter the settlement reduction factor  $\alpha$  [-] and the groundwater depth under the ground surface  $G_w$  [m]. Then click "OK" button.

Main Soil Data			$\mathbf{X}$
Soil properties Calculation parameters of flexibility coefficients	Bearing capacity factor	s	
Main Soil Data:			
Settlement reduction factor Alfa $\leq 1$	Alfa	[-]	1
Groundwater depth under the ground surface	Gw	[m]	2.00
	1		
<u>Q</u> k <u>C</u> ancel	Help		

Figure 12-22 "Main soil data" dialog box

Now all data and parameters for the boring log have been entered. After finishing the creation of the boring log, click "OK" button in "Soil data" dialog box in Figure 12-21 to see the defined boring log on the screen where the user can control or modify the input data and parameters, Figure 12-23.

ELPLA-Boring - [Piled raft]		
<u>Eile ⊻iew D</u> ata <u>G</u> raphically <u>O</u> ptions Forma <u>t</u> <u>W</u>		
b 🚅 🖬 🔍 🔺 🏠 🖗 🕼 🛍	67 C	
🔳 📄 🎽 🦻 🖬 🔺 २.२.२	🗕 🕘 🕲 🛠	
		·
2003	10000[k8/m2]_Fh1 = 0[*]	
CNV 2.08 U 2.00 G	180881884021,2h1 = 81 <sup>47</sup> 180881840421,2c = 15184421 211842401,3h24 = 8.31 <sup>-1</sup>	
<u>i i</u>		
見る		
55 E		
10 E		
10 C		
18 B		
u <b>200</b> K	$\begin{array}{l} 100001[080/m2], \mathcal{F}N1 = 0 \left[ \begin{array}{c} 0 \\ \end{array} \right] \\ 100001[080/m2], \mathcal{C} = 1.5 \left[ 080/m2 \\ \end{array} \\ \mathcal{F}100001 = 0.31 - 1 \end{array}$	
50,00 <b>14.1</b> Ga	= 8(k8/m3), Nue = 0.3(-)	
		-
		•
		 ص 55:15 20/11/2007

Figure 12-23 Boring log on the screen

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

- Choose "Save boring logs" command from "File" menu in Figure 12-23 to save the data of boring log
- Choose "Close boring logs" command from "File" menu in Figure 12-23 to close the *ELPLA-Boring* sub program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Soil properties" command in the "Data" menu of *ELPLA-Data*.

## 2.7 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 12-24 appears with default foundation properties. The data of foundation properties for the current example, which are required to define, are foundation material and foundation thickness. Any other data corresponding to foundation properties in the program menus are not required. Therefore, the user can take these data from the default foundation properties.

	- 134	<b>-</b>													
 1	1	1	1	1	1	1	1	1	1	1	1	1	1		
		1		1	1	1	1	1		1	1	1	1		
 1	1		1						1						
 1	1	1	1	1	1	1	1	1	1	1	1	1	1		
1	1	1	1	1	1	1	1	1	1	1	1	1	1		
 1	1	1	1	1	1	1	1	1	1	1	1	1	1		
 1	1	1	1	1	1	1	1	1	1	1	1	1	1		
 1	1	1	1	1	1	1	1	1	1	1	1	1	1		
 1	1	1	1	1	1	1	1	1	1	1	1	1	1		
 1	1	1	1	1	1	1	1	1	1	1	1	1	1		
								ļ							
 1	1	1	1	1	1	1	1	1	1	1	1	1	1		
1	1	1	1	1	1	1	1	1	1	1	1	1	1		
1	1	1	1	1	1	1	1	1	1	1	1	1	1		

Figure 12-24 "Foundation properties" embedded program

To enter the foundation material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 12-24. The following list box in Figure 12-25 with default data appears. To enter or modify a value in this list box, type that value in the corresponding cell then press "Enter" key. In the list box of Figure 12-25, enter E-Modulus of the foundation, *Poisson's* ratio of the foundation and foundation thickness
- Click "OK" button

Definir	ng elemen	t groups (with	the same thi	ckn	ess and 🔀
Group No.	E-Modulus of slab [kN/m2]	Poisson's ratio of slab [-]	Slab thickness d [m]		<u>O</u> k
1	3.4E+07	0.2			<u>C</u> ancel
	1				Insert
					⊆ору
					Delete
					New
					Help
					<u>E</u> xcel

Figure 12-25 "Defining element groups" list box

To enter the unit weight of the foundation

- Choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 12-24. The following dialog box in Figure 12-26 with a default unit weight of 25 [kN/m<sup>3</sup>] appears. Let the default value as written in the edit box "Unit weight of the foundation"
- Click "OK" button

Unit weight of the founda	ation	×
Unit weight of the foundation	Gb [kW/m3] 25	
<u>O</u> kNew	Cancel Help	

Figure 12-26 "Unit weight of the foundation" dialog box

To enter the foundation level

- Choose "Foundation depth" command from "Foundation properties" menu in the window of Figure 12-24. The following dialog box in Figure 12-27 appears
- In this dialog box type 2 in the "Foundation depth under ground surface (a)/ (b)" edit box
- Click "OK" button

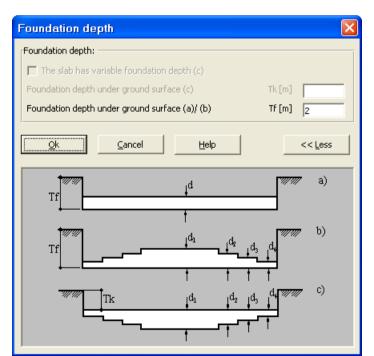


Figure 12-27 "Foundation depth" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 12-24 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 12-24 to close the "Foundation properties" embedded program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Foundation properties" command in the "Data" menu of *ELPLA-Data*.

#### 2.8 Loads

To define the loads, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 12-28 appears.

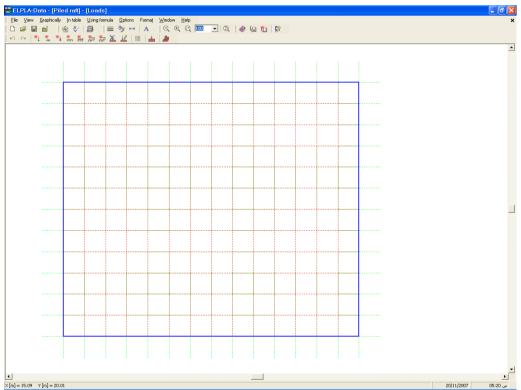


Figure 12-28 "Loads" embedded program

## To enter loads

- Choose "Distributed load (rectangular)" command from "Graphically" menu in the window of Figure 12-28. When "Distributed loads (rectangular)" command is chosen, the cursor is changed from an arrow to a cross hair. Then, the load can be defined by holding the left mouse button down at the starting point of the distributed load. As the mouse is dragged, a box appears, indicating a distributed load is being defined. When the left mouse button is released, the following dialog box in Figure 12-29 appears with the load value and coordinates In this dialog box

- Type 150 in the "Load value" edit box
- Modify the load coordinate
- Click "OK" button

Loading				×
Distributed loads: —				
Load value			p [kN/m2]	150
Load start			×1 [m]	0.00
Load start			y1 [m]	21.0
Load end			×2 [m]	24.5
Load end			y2 [m]	0.00
<u>k</u>	<u>C</u> ancel	Help		<< <u>L</u> ess
	p	(*2, 72)		

Figure 12-29 "Distributed loads *p*" list box

After you have completed the definition of all load data, the screen should look like the following Figure 12-30.

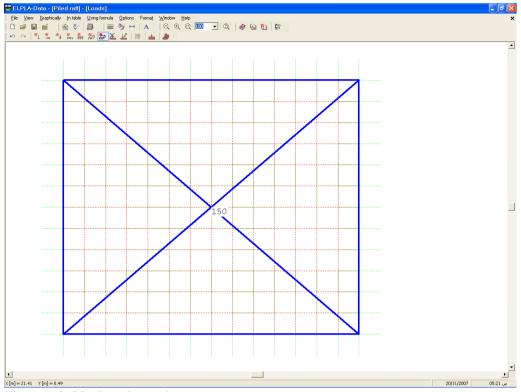


Figure 12-30 Loads on the screen

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

- Choose "Save loads" command from "File" menu in Figure 12-30 to save the load data
- Choose "Close loads" command from "File" menu in Figure 12-30 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*.

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Loads" command in the "Data" menu of *ELPLA-Data*.

Creating the project of the raft is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

## **3** Carrying out the calculations

## 3.1 Starting ELPLA-Solver

To analyze the problem, leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window appears, Figure 12-31.

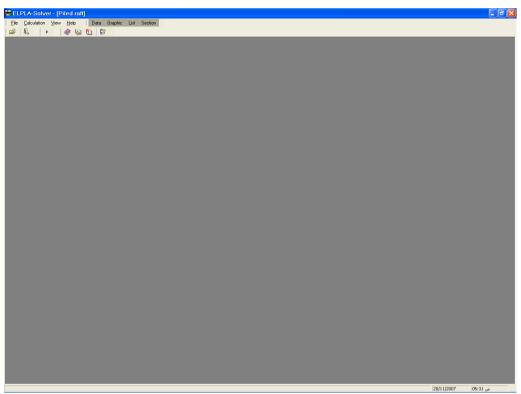


Figure 12-31 Opening screen of the sub program ELPLA-Solver

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. The "Calculation" menu contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Assembling the slab stiffness matrix
- Solving the system of linear equations (unsymmetrical matrix)
- Determining deformation, internal forces, contact pressures

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

#### **3.2** Carrying out all computations

To carry out all computations in one time, choose "Computation of all" command from "Calculation" menu in *ELPLA-Solver* window. The analysis progress menu in Figure 12-32 appears in which various phases of calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

Determining flexibility coefficients of the soil
Assembling the flexibility matrix!
Time remaining = 00:00:03
I = 138 from 363 steps

Figure 12-32 Analysis progress menu

Once the analysis is complete, a check menu of the solution appears, Figure 12-33. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

Check of the solution	
V - Load	
Total load	[kN] = 109828
Sum of contact pressures	[kN] = 108588
X - Moment	
Sum M× from loads	[kN.m] = 0
Sum Mx from contact pressures	[kN.m] = 114
Y - Moment	
Sum My from loads	[kN.m] = 0
Sum My from contact pressures	[kN.m] = 575
Ok Help	

Figure 12-33 Menu "Check of the solution"

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

# 4 Viewing data and results

*ELPLA* can view and print a wide variety of results in graphics, diagrams or tables through the three sub programs *ELPLA-Graphic*, *ELPLA-Section* and *ELPLA-List*. Data can also be viewed again and printed by the sub programs *ELPLA-Graphic* and *ELPLA-List*. Note that *ELPLA-Data* is used only to define and view the data of the problem. *ELPLA-Graphic* is used to print data graphically while *ELPLA-List* is used to print data numerically.

## 4.1 Viewing result graphics

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

*ELPLA-Graphic* window appears, Figure 12-34. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

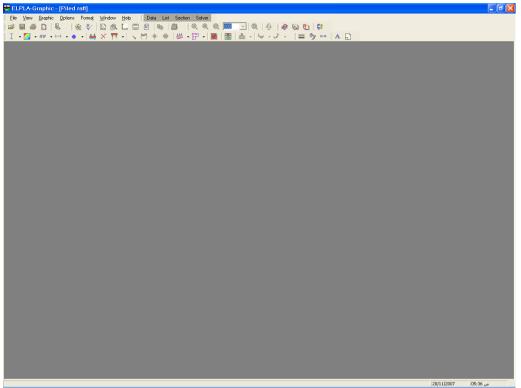


Figure 12-34 Opening screen of the sub program *ELPLA-Graphic* 

To view the results for the raft as contour lines

- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 12-35 appears
- In "Results as contour lines" option box, select "Settlements *s*" as a sample for the results to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for the raft as shown in Figure 12-36.

Results as Contour Lines		
Select one item to draw		
Settlements	C Contact pressure q	
C Moments mx	C Moments my	
C Moments mxy	C Shear forces Qx	
C Shear forces Qy		
C Support reactions V	Qk	
C Modulus of subgrade reaction ks	Cancel	
C Principal moments hm1		
C Principal moments hm2	<u>H</u> elp	

Figure 12-35 "Results as contour lines" option box

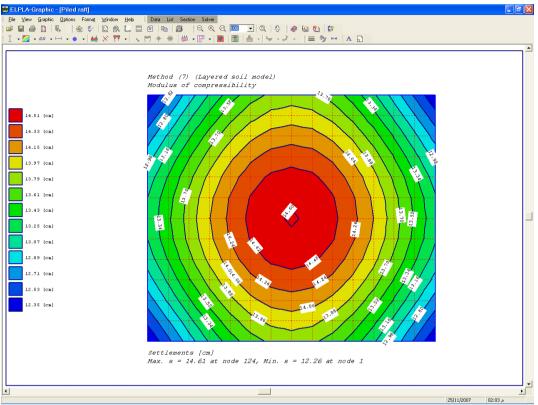


Figure 12-36 Settlements as contour lines for the raft

# 4.2 Plot a diagram of the results

To plot a diagram of the results, leave *ELPLA-Graphic* to *ELPLA-Section*. This is done by clicking on "Section" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Graphic* window. *ELPLA-Section* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Section* window.

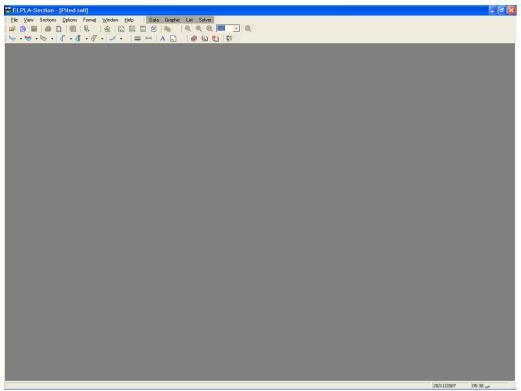


Figure 12-37 Opening screen of the sub program ELPLA-Section

Only the first command of the "Sections" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Options", "Format" and "Window" menus, which are used to define the preferences of the drawing such as plot parameters, scale, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

To plot a section in *x*-direction

- Choose "Section in *x*-direction" command from "Sections" menu. The following option box in Figure 12-38 appears
- In the "Section in *x*-direction" option box, select "Settlements *s*" as an example for the results to be plotted in a diagram
- Click "OK" button

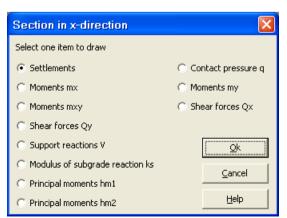


Figure 12-38 "Section in *x*-direction" option box

The following option box in Figure 12-39 appears to specify the section in x-direction that is required to be plotted.

In this dialog box

- Type 10.5 in the "Section at *y*-coordinate" edit box to plot a diagram at the middle of the raft
- Click "OK" button

The settlements are now plotted in a diagram at the middle of the raft as shown in Figure 12-40.

Section in x-direction	X
Set range in y-direction:	
Section at y-coordinate	[m] 10.5
Set range in x-direction:	
at x-coordinate X1	[m] 0.00
to x-coordinate X2	[m] 24.50
Qk <u>C</u> ancel <u>H</u> elp << Less	
]	

Figure 12-39 "Section in *x*-direction" dialog box

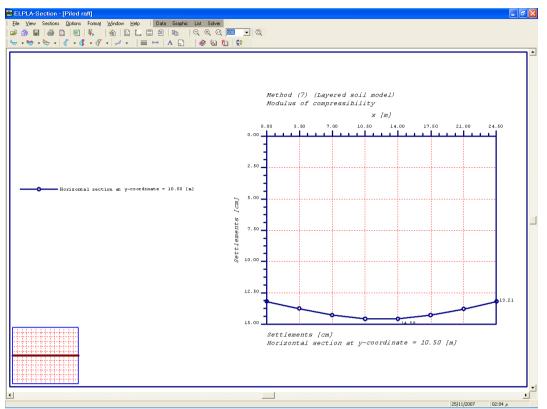


Figure 12-40 Diagram of settlements in x-direction at the middle of the raft

## 4.3 Listing data and result in tables

To list tables of data and results, switch to *ELPLA-List*. This is done by clicking on "List" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Section* window. *ELPLA-List* window appears, Figure 12-41.

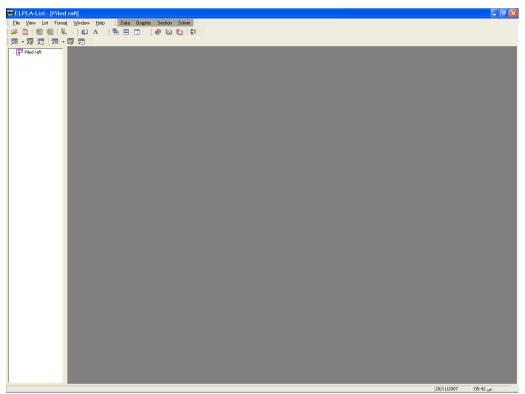


Figure 12-41 Opening screen of the sub program ELPLA-List

The function of *ELPLA-List* is listing and printing data and results in tables. The data and results can be exported to other Windows applications to prepare reports or add further information. *ELPLA-List* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-List* window.

Only one command of the "List" menu is explained here. In the same way, the user can carry out the remaining commands of the previous list. The commands of "Format" and "Window" menus, which are used to define the preferences of the tables such as page format, font, etc., are discussed in detail in the User's Guide of *ELPLA*.

To list results in tables

- Choose "Display tables of results" command from "List" menu.
- The following option box in Figure 12-42 appears
- In this option box, select "Pile results" as an example for the results to be listed in a table
- Click "OK" button. The pile results are now listed on the screen (Figure 12-43)
- Choose "Send to Word" from "File" menu if you wish to export the text to the MS Word application, Figure 12-44

Display Tables of Results	
Select one item to list	
C Modulus of subgrade reaction	C Settlements
C Contact pressure	C Slab deformation
C Moments	C Shear forces
C Support reactions	
C Flexibility matrix [Ce]	Qk
C Flexibility matrix [Cw]	Cancel
C Soil stiffness matrix [ks]	
Pile results	Help



draft					
Pile results	l.	results			
	Total	pile loads	oad (including o f piled raft	m ut.)	Po [kH] = 109928.4 PL [kH] = 83842.2 Alfa-Rpp [6] = 76.34
			isplacements		
	Pile	pile	Settlement	Soil	
	No.	load	Decoremento	stiffness	
	I	Fr	Sr	ks	
	[-]	[kN]	[cn]	[hN/m]	
	1	4009.8	12.80	31337.8	
	2	2674.5	13.39	19979.2	
	3	2582.0	13.82	18689.4	
	4	2594.4	13.97	18568.8	
	5	2583.4 2677.7	13.82 13.40	18693.3 19988.6	
	2	4018.1	13.40	31368.7	
		2577.8	13.22	19503.6	
	9	1183.5	13.79	8584.4	
	10	1096.5	14.20	7722.0	
	11	1106.4	14.35	7711.7	
	12 13	1095.8 1182.0	14.19 13.78	7719.7 8579.1	
	14	2571.9	13.20	19479.6	
	15	2433.7	13.45	18089.9	
	16	1068.6	14.02	7622.5	
	17	983.4	14.43	6814.4	
	18	993.9	14.58	6816.3	
	19 20	984.0 1069.6	14.44 14.03	6816.3 7625.1	
	21	2438.8	13.47	18110.4	
	22	2438.6	13.47	18110.2	
	23	1069.4	14.03	7624.3	
	24	983.7	14.43	6814.8	
	25 26	993.8 983.6	14.58 14.43	6815.7 6816.1	
	27	1069.0	14.02	7625.4	
	28	2434.1	13.45	18092.5	
	29	2572.6	13.20	19484.5	
	30	1182.7	13.78	8584.6	
	31 32	1096.2 1106.1	14.19 14.35	7723.2 7710.6	
	32	105.1	14.20	7718.7	
	34	1183.0	13.78	8582.2	
	35	2577.2	13.21	19502.7	
	36	4015.9	12.81	31360.9	
	37	2675.6	13.39	19979.3	
	38	2581.2 2592.7	13.82 13.97	18683.1 18560.8	
	40	2581.2	13.81	18687.0	
	40	2674.3	13.39	19979.1	
	42	4009.4	12.79	31337.0	

Figure 12-43 List of pile results

📴 Document2 - Microsoft Word					
File Edit View Insert Format Tools Table Window Help					Type a guestion for help 🗸 🗙
		n 📪 🎫 🦓 🖾 🦷	100% - 🕜	Read	
👍 Normal + (Latin - Courier New - 8 - B I 🕎 📰					
	-			4	~
	i i	; 7	: 1		
·					
1					
7					
1					
-	Pile results				
	Value of tot Total pile 1	al load (including o oads	own wt.)	Po [kN] = 109828.4 PL [kN] = 83842.2	
	Bearing fact	or of piled raft		Alfa-Kpp [4] = 76.34	
		nd displacements			
7	Pile pi No. lo		Soil stiffness		
	I	Fr Sr N] [cn]	lts [ltN/n]		
			31337.8		
	2 2674 3 2582	.5 13.39	19979.2 18689.4		
N -	4 2594 5 2583		18568.8 18693.3		
	6 2677 7 4018	.7 13.40	19988.6 31368.7		
·	8 2577 9 1183	.8 13.22	19503.6 8584.4		
	10 1096 11 1106	.5 14.20	7722.0 7711.7		
	12 1095 13 1182	.8 14.19	7719.7 8579.1		
	14 2571 15 2433	.9 13.20	19479.6 18089.9		
	16 1068 17 983	.6 14.02	7622.5		
	18 993 19 984	.9 14.58	6816.3 6816.3		
	20 1069 21 2438	.6 14.03	7625.1		
	22 2438 23 1069	.6 13.47	18110.2 7624.3		
	24 983 25 993	.7 14.43	6814.8		
	26 983 27 1069	.6 14.43	6816.1 7625.4		
-	28 2434 29 2572	.1 13.45	18092.5		
	30 1182 31 1096	.7 13.78	8584.6		
	32 1106 33 1095	.1 14.35	7710.6		
	34 1183	.0 13.78	8582.2		
۵۵ ۲	35 2577 36 4015 37 2675	.9 12.81	19502.7 31360.9 19979.3		
	37 2675 38 2581 39 2592	.2 13.82	18683.1		
	40 2581	. 2 13.81	18560.8 18687.0		1
	41 2674 42 4009		19979.1 31337.0		•
					*
Page 1 Sec 1 1/17 At 1" Ln 1 Col 1 R	EC TRK EXT C	WR English (U.S			
Figure 12-44 Exported	result	s in MS	Word		

Figure 12-44 Exported results in MS Word

# 5 Index

# A

# B

Boring log 15, 17	7, 19
-------------------	-------

# С

Calculation	24
Calculation methods	4, 5
Check of the solution	26
Computation of all	25
Contact pressures	25
Contour lines	27

# D

Deformation			25
Diagram	26,	28,	31

# E

ELPLA-Boring	
<i>ELPLA-Data</i> 4, 8, 9, 11, 12,	15, 19, 22, 24, 26
ELPLA-Graphic	
ELPLA-List	
ELPLA-Section	
ELPLA-Solver	
E-Modulus	

# F

9
4, 7, 11, 15, 22, 24, 26, 32

# G

Generation type	9,	10
Graphic		26

# Ι

Internal forces
-----------------

## K

-	
Kev	

# L

List	31,	32
Load vector		25
Loads	22,	24

## M

Main soil data	17, 13	8
Modulus of elasticity for loading	····· ·	4
Modulus of elasticity for reloading		4

# N

Numerical calculation	1
-----------------------	---

## 0

# P

Poisson's ratio	
Project	
Project identification	

# R

Raft ...... 3, 4, 7, 9, 24, 27, 28, 30, 31

# S

Save	. 7, 8, 11, 15, 22, 24
Save as	
Soil	4, 17 25
Soil properties	
Soil stiffness matrix	
System symmetry	6

# T

# Tutorial manual ...... 4

# U

Unit weight	3, 4, 17, 21
Unsymmetrical system	6
User's Guide	4, 29, 32

## 

Y

# Example 13

# Interaction of lowering the foundation of a building with an underground tunnel

## Contents

1	Desci	ription of the problem	3
	1.1	Loads and dimensions	3
	1.2	Foundation material and thickness	5
	1.3	External wall properties and dimensions	5
	1.4	Soil properties	3 3 5 5 5 5
	1.5	Mathematical model	5
	1.6	Analysis	6
2	Creat	ing the project	6
	2.1	Calculation method	6
	2.2	Project identification	10
	2.3	FE-Net data	11
	2.4	External wall properties	15
	2.5	Soil properties	19
	2.6	Foundation properties	25
	2.7	Boring fields	28
	2.8	Loads	29
3	Creat	ing the project of the second case (including the underground tunnel)	32
	3.1	Modifying the calculation methods options	32
	3.2	Modifying the project identification	33
	3.3	Additional Settlements	33
4	Carry	ing out the calculations	35
	4.1	Starting ELPLA-Solver	35
	4.2	Carrying out all computations	37
5	View	ing data and results	39
	5.1	Viewing result graphics	39
	5.2	Plot a diagram of results	42
6	Index		47

#### **1** Description of the problem

An example of lowering the foundation of a building with an underground tunnel is selected to illustrate some of the essential features of *ELPLA* for studying the influence of an underground tunnel on a building foundation.

#### 1.1 Loads and dimensions

The building has the outlines shown in Figure 13-1. It consists of two combined rectangles, their corners are not connected. The raft foundation is 50 [cm] thick and lies at a depth 2.50 [m] below the ground surface. The planned tunnel runs diagonally to the building axis. The tunnel is represented as a settling mould of 10 [m] width with maximum settlement of 3 [cm] caused by the tunnel. The settling mould which is shown in Figure 13-1 is marked by lines; each line represents a settlement value running symmetrically to the tunnel axis. This settlement values are taken in the calculation as secondary effects.

The foundation loads consist of two concentrated loads of P = 18 [MN] and line load of p = 300 [kN/m] on the external walls. The soil was explored through three borings from BPN1 to BPN3 to 14 [m] depth under the raft.

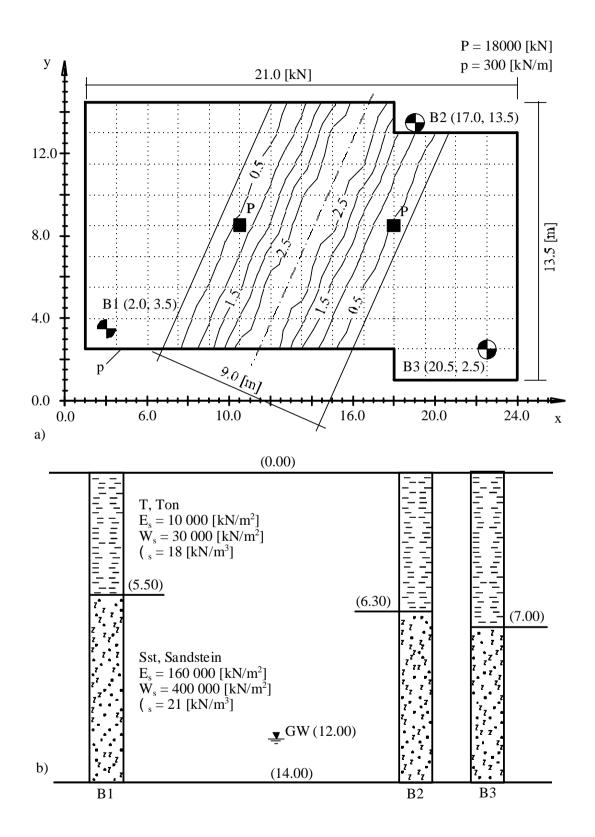


Figure 13-1 a) Plan of the raft with settlement lines through the underground tunnel and boring points b) Boring logs BPN1 to BPN3

#### **1.2** Foundation material and thickness

Foundation material and thickness are supposed to have the following parameters:

Foundation thickness	d	= 0.5	[m]
Young's modulus	$E_c$	$=3 * 10^{7}$	$[kN/m^2]$
Unit weight	$\gamma_c$	= 25	$[kN/m^3]$
Poisson's ratio	$v_c$	= 0.2	[-]

#### **1.3** External wall properties and dimensions

The wall properties and dimensions are supposed to have the following parameters:

Breadth	b	= 0.3	[m]
Depth	d	= 3	[m]
Moment of Inertia	Ι	= 0.675	[m <sup>4</sup> ]
Shear modulus of the wall	G	$= 1.25 * 10^7$	$[kN/m^2]$
Torsional moment of inertia	J	= 0.0252	[m <sup>4</sup> ]
Walls own weight	pb	= 22.50	[kN/m]

#### 1.4 Soil properties

The subsoil under the building consists of a layer of clay, overlying a layer of stony sand. The clay layer is supposed to have the following parameters:

Modulus of compressibility for loading	$E_s$	= 10000	$[kN/m^2]$
Modulus of compressibility for reloading	$W_s$	= 30000	$[kN/m^2]$
Unit weight	$\gamma_s$	= 18	$[kN/m^3]$

The stony sand layer is supposed to have the following parameters:

Modulus of compressibility for loading	$E_s$	= 160000	$[kN/m^2]$
Modulus of compressibility for reloading	$W_s$	= 400000	$[kN/m^2]$
Unit weight	$\gamma_s$	= 21	$[kN/m^3]$

#### **1.5** Mathematical model

The influence of surrounding structures and external loads can be taken into consideration only for the Continuum model (methods 4 to 9). The Continuum model based on, the settlement at any node is affected by the forces at all the other nodes. In this example, the Modulus of compressibility method (method 7) is chosen to analyze our problem.

This Tutorial manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis, a complete reference for the soil models and numerical calculation methods are well documented in the User's Guide of *ELPLA*.

#### 1.6 Analysis

To analyze the foundation, it's subdivided into 112 quadratic elements with side length equals 1.5 [m] as shown in Figure 13-1. Two independent names define the data of the two models which are chosen. The data are quite similar for the two models except the underground tunnel effect in the second model.

The analysis of the first model is carried out first to obtain the internal forces in the first case (without consideration of the underground tunnel).Then, the second model is carried out to obtain the internal forces in the second case (with considering the underground tunnel effect). After that, the user should compare the results of the two cases to find out the influence of constructing the tunnel under a raft foundation.

#### 2 Creating the project

In this section, the user will learn how to create a project for analyzing a raft foundation lowered with an underground tunnel. Thus is done by first entering the data of the foundation in the same manner of the previous foundation examples.

#### 2.1 Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation method" wizard appears, Figure 13-2. This wizard will guide you through the steps required to create the project. As shown in Figure 13-2, the first form of the wizard is the "Analysis type" form. In this form, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a foundation problem, select "Analysis of slab foundation" then click "Next" button to go to the next page.

Calculation methods	3		
Analysis of slab foundation	Analysis of piled raft	Analysis of system of many slab foundations	
Analysis of slab floor	Analysis of grid	Analysis of plane frame	Analysis of plane stress
Load Help Save	<u>A</u> s <u>C</u> ancel	<u> </u>	ext > Save

Figure 13-2 "Calculation method" wizard with "Analysis type" form

After clicking "Next" button, the "Calculation methods" form appears, Figure 13-3.

To define the calculation method

- Select the calculation method "6-Modulus of Compressibility (Iteration)"
- For the subsoil model select the option "Layered soil model"
- Click "Next" button to go to the next form

Calculation methods												
Calculation methods:												
🔿 1- Linear Contact Pressure (Conventional Method)												
C 2/3- Constant/ Variable Modulus of subgrade Reaction												
C 4- Modification of Modulus of subgrade Reaction by Iteration												
C 5- Isotropic Elastic Half Space												
6- Modulus of Compressibility (Iteration)												
C 7- Modulus of Compressibility (Elimination)												
C 8- Modulus of Compressibility for Rigid Raft												
C 9- Flexible foundation												
Subsoil model:												
C Half Space model												
C Layered soil model												
Help     Save As     Cancel     < Back     Next >												

Figure 13-3 "Calculation methods" form

The next form is the "System symmetry" (Figure 13-4). In this form choose "Unsymmetrical system" then click "Next" button.

Calculation methods	
System symmetry:	
Unsymmetrical system	
Symmetrical system about x-axis	Double-symmetrical system
Symmetrical system about y-axis	Anti-symmetrical system about
	X-0XI5
Help Save As Cancel	<back save<="" td=""></back>

Figure 13-4 "System symmetry" form

The last form of the wizard assistant contains the "Option" list, Figure 13-5. In this list, *ELPLA* displays some of the available options corresponding to the used numerical model, which differ from model to other. Check the option "Slab with girders", then click the "Save" button.

Calculation methods										
Options:										
Slab with girders										
Addtional springs										
Supports/ Boundary Conditions										
Nonlinear subsoil model										
Determining displacements in soil     Determining stresses in soil										
Determining stresses in soil										
$\overline{\Pi} \underline{T}^*$ Influence of the temperature change on raft										
□ Tinfluence of additional settlements on raft										
Select All										
Nonlinear analysis of piled raft:										
Nonlinear analysis using a hyperbolic function for load-settlement										
C Nonlinear analysis using DIN 4014 for load-settlement										
Help     Save As     Cancel     < Back										

Figure 13-5 "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 13-6.

In this dialog box

- Type a file name for the current project in the "File name" edit box. For example type "TU1". *ELPLA* will use automatically this file name in all reading and writing processes
- Click "Save" button

Save As				? 🗙
Save jn: 隘	TU	•	*ے 🖻	<b>.</b>
TU1				
File <u>n</u> ame:	TU1			<u>S</u> ave
Save as <u>t</u> ype:	Isolated slab foundation-files (*.PO1)		•	Cancel

Figure 13-6 "Save as" dialog box

*ELPLA* will activate the "Data" menu. Also the file name of the current project [TU1] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar, Figure 13-7.

In the "Data" menu, the user can enter the remaining data of the project using the same sequence of commands in this menu. The first command in the menu is "Calculation methods", which has been already entered. Therefore, *ELPLA* has put the sign " $\sqrt{}$ " beside this command, Figure 13-7. *ELPLA* puts this sign beside the commands those the user has entered so that the user can know which data were defined.

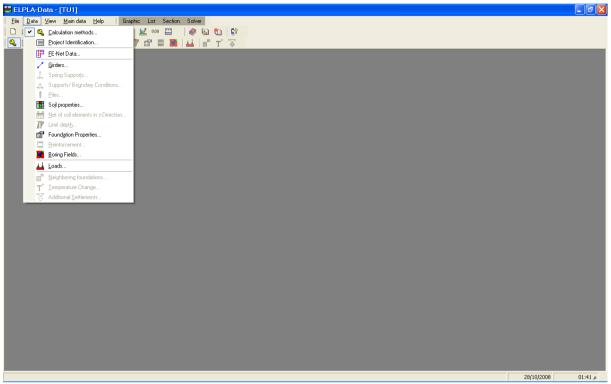


Figure 13-7 ELPLA-Data after defining the calculation method

#### 2.2 **Project identification**

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 13-8 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Influence of ground lowering due to a tunnel on a building"
- Type the date of the project in the "Date" edit box
- Type "Without tunnel" in the "Project" edit box
- Click "Save" button

Project I	Identification	X										
Project Id	entification:											
Title	Influence of ground lowering due to a tunnel on a building											
Date	28 October 2008											
Project	Without tunnel											
<u>S</u> ave												

Figure 13-8 "Project identification" dialog box

#### 2.3 FE-Net data

Choose "FE-Net data" command from "Data" menu of *ELPLA-Data*. The "FE-Net generation" wizard appears as shown in Figure 13-9. This wizard will guide you through the steps required to generate the FE-Net. As shown in Figure 13-9 the first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have regular shapes. For the given problem, the foundation has nearly a rectangular shape.

To generate the FE-Net

- In the "Slab type" options, choose the rectangle slab option
- Type 21 in the "Length of rectangular slab" edit box
- Type 13.5 in the "Width of rectangular slab" edit box
- Click "Next" button to go to the next form

FE-Net Generation
Slab type:
Rectangular slab:
Length of rectangular slab
Width of rectangular slab B [m] 13.5
Help Cancel < Back Next > Einish

Figure 13-9 "Slab type" form

After clicking "Next" in the "Slab type" form, the following "Generation type" form appears, Figure 13-10. *ELPLA* can deal with various types of generations with rectangular elements. Choose the rectangular elements option as the generation type. Then click "Next" button to go to the next form.

FE-Net Generation	1		
Generation type:		× + × + × + × × + × + × + × + × × + × + × + × + × × + × + × + × + × × + × + × + ×	·····································
Help	Cancel	< Back Next >	• <u> </u>

Figure 13-10 "Generation type" form

The next form of the "FE-Net generation" wizard is the "Grid definition" dialogue box. In this dialogue box, the default values of constant element size appear, Figure 13-11.

In this "Grid definition" dialog box

- Type 14 in the "No. of grid intervals" edit box for grids in *x*-direction
- Type 1.5 in the "Grid interval  $D_x$  [m]" edit box for grids in x-direction
- Type 9 in the "No. of grid intervals" edit box for grids in y-direction
- Type 1.5 in the "Grid interval  $D_y$  [m]" edit box for grids in y-direction
- Click "Finish"

FE-Net Generation	
Grid definition:	
Grids in x-direction:	
🔽 Constant grid interval	
No. of grid intervals	14 •
Grid interval Dx [m]	1.5
Grids in y-direction: Constant grid interval No. of grid intervals Grid interval Dy [m]	9 :
<u>H</u> elp <u>C</u> ance	el < Back Next > Finish

Figure 13-11 "Fe-Net generation" form

Click "Finish" button in the "Generation parameters" form. *ELPLA* will generate a suitable FE-Net for a rectangular foundation of 21 [m] length and 13.5 [m] width with rectangular elements, which have equal areas. The following embedded program in Figure 13-12 appears with the generated net.

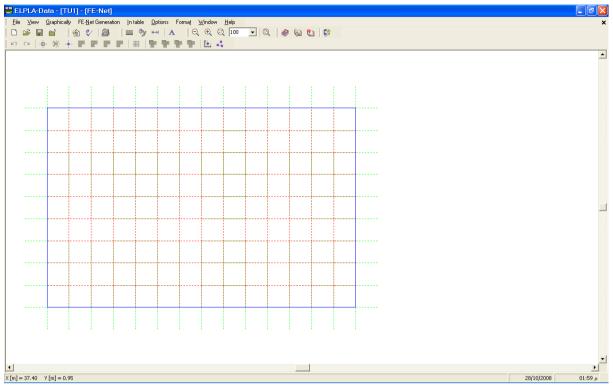


Figure 13-12 FE-Net on the screen

#### **Deleting nodes from the FE-Net**

To select the unnecessary nodes, that are required to be removed from the net, first choose "Select nodes" command from the "Graphically" menu in Figure 13-12. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair. The command "Remove nodes" in the menu "Graphically" will be enabled, indicating the mode in which is being operated. Next, select the required nodes by clicking on each node individually or selecting a group of nodes as shown in Figure 13-13. A group of nodes can be selected by holding the left mouse button down at the corner of the region. Then, dragging the mouse until a rectangle encompasses the required group of nodes. When the left mouse button is released, all nodes in the rectangle are selected.

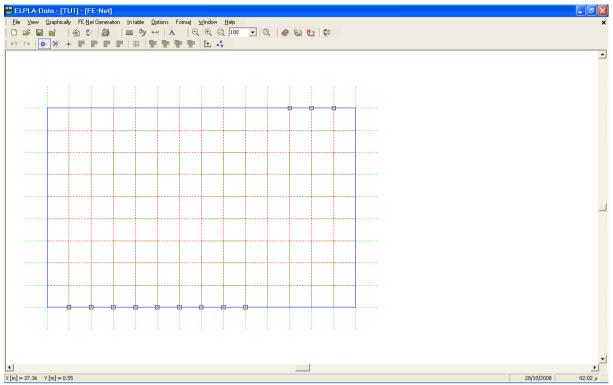


Figure 13-13 Generated FE-Net after selecting the unnecessary nodes

To remove the selected nodes, choose "Remove nodes" command from the "Graphically" menu. The action of this command is indicated in Figure 13-14. To leave the graphic mode, press "Esc" key.

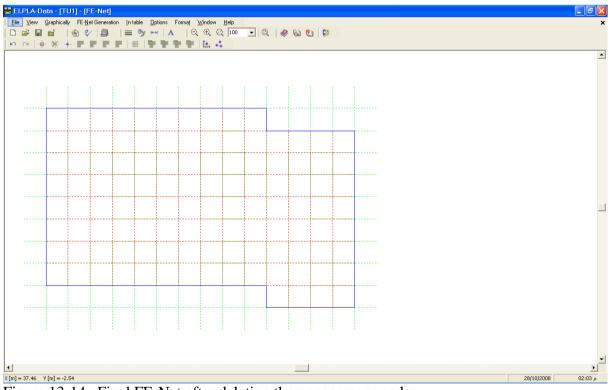


Figure 13-14 Final FE-Net after deleting the unnecessary nodes

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

- Choose "Save FE-Net" command from "File" menu in Figure 13-14 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 13-14 to close the "FE-Net" embedded program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "FE-Net" command in the "Data" menu of *ELPLA-Data*.

#### 2.4 External wall properties

To define the external walls, choose "Girders" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 13-15 appears.

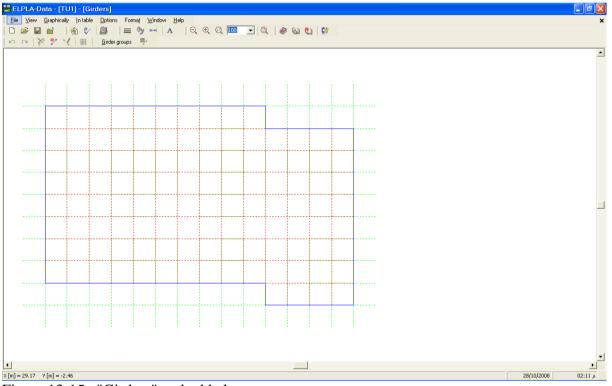


Figure 13-15 "Girders" embedded program

To enter the cross section of the girders

- Choose "Girder groups" command from "In Table" menu in Figure 13-15. The following option box in Figure 13-16 appears
- In this option box, select "Rectangular cross section"
- Click "OK" button

Cross section definition	X								
Cross section definition:									
<u>Rectangular cross section</u> <u>General cross section</u>	<u>k</u>								
C General cross section C Create a new element group as T/L-girder									
	Help								

Figure 13-16 "Cross section definition" option box

After clicking "OK" button in the "Cross section definition" option box, the following list box in Figure 13-17 appears.

In this list box

- Enter the material properties of the walls, cross section dimensions and the wall weight as indicated in Figure 13-17. This is done by entering the value in the corresponding cell and press "Enter" button
- Click "OK" button

àirder	groups					X
Group No.	E-Modulus of girder	G-Modul of girder G	Height of girder h	Width of girder b	Girder weight pb	
	E [kN/m2]	Cancel				
1	3E+07	1250000	3.00	0.30	22.50000	Insert
						⊆ору
						Delete
						New
						Help

Figure 13-17 "Defining girder groups" list box

#### Defining the girder locations on the net

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

To define the walls location on the net graphically

- Choose "Add girders" command from the "Graphically" menu in Figure 13-15. When "Add girders" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the start node of the first girder and drag the mouse until the end node of that girder (Figure 13-18). Then, click on the end node. The "Girder elements" dialog box in Figure 13-19 appears

		ita - 🎵																		Į
ile ⊻i	iew <u>(</u>	Graphical	y <u>I</u> n	table	<u>O</u> ptio	ns For	ma <u>t</u>	<u>//</u> indov	w <u>H</u> e	lp	-	-	_		~					
ĭ₽,			1	₹⁄	<b>a</b>	=	~ <u>&gt;</u> /	H I	Α	Q	. ⊕.	Q	100	-	Q	0	6	1	<b>6</b> 8	
Ci.		*		₩	<u>G</u> irde	er groups	- Y													_
												1			1					
					_		-	-	_		-	-			-i					 
															-					
							ļ								_					 
							ļ													 
							÷				+								+	 
							İ		~		· †									 
								-												
					_															[
															-			_	_	ļ
																				1
[m] = 8.5	5 Y	[m] = 4.3	1																	

Figure 13-18 Add girder by mouse

In this dialog box

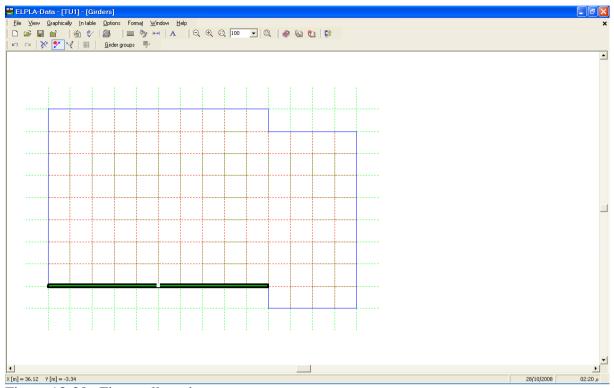
- Select the group No.
- Click "OK" button

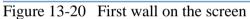
Girder elements	
Group No. Start from node No.	[-] 1 • [-] 32
End at node No.	[-] 42
QkCancel	Help

Figure 13-19 "Girder elements" dialog box

Now, the first wall is defined as shown in Figure 13-20. Note that *ELPLA* has typed automatically the girder type on it indicating the No. of girder group.

Repeat the previous steps to add the remaining walls on the net. After you have completed the definition of all walls, the screen should look like the following Figure 13-20.





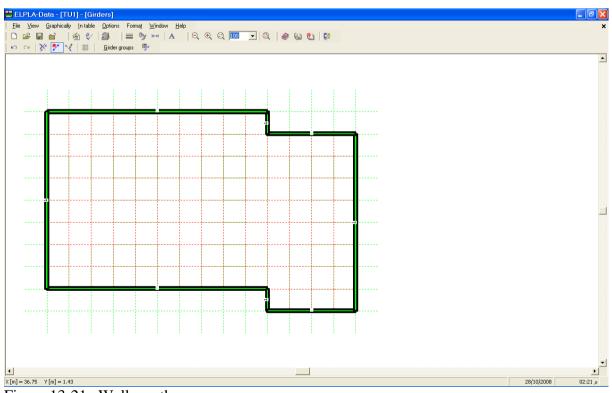


Figure 13-21 Walls on the screen

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

- Choose "Save girders" command from "File" menu in Figure 13-21 to save the data of girders
- Choose "Close girders" command from "File" menu in Figure 13-21 to close the "Girders" embedded program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Girders" command in the "Data" menu of *ELPLA-Data*.

#### 2.5 Soil properties

In *ELPLA*, there are three different soil models with several calculation methods. Therefore, the soil properties for each method are required to be defined according to the used soil model. In the current example, the soil model, which is used in the analysis, is Layered Soil Model. This model requires that the subsoil has to be defined by boring logs. In the example, three boring logs at different locations define the soil under the raft. Each boring log has two layers with different soil materials. The geotechnical data for each layer are unit weight of the soil  $\gamma_s$  and modulus of Elasticity for loading  $E_s$  and reloading  $W_s$ .

To define the soil properties, choose "Soil properties" command from "Data" menu of *ELPLA-Data*. The following sub program in Figure 13-22 appears with a default-boring log.

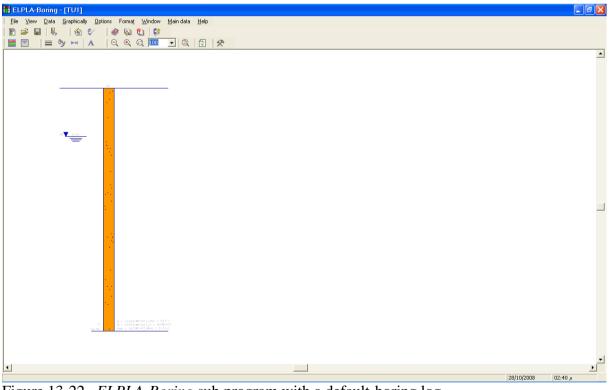


Figure 13-22 *ELPLA-Boring* sub program with a default-boring log

In Figure 13-22, soil properties are defined through the "Data" menu which contains the following two commands:

- "Soil data" command. This command is used to define the individual boring logs
- "Main soil data" command. This command is used to define the general data for all soil layers

To define the soil properties for the three boring logs of the current example

- Choose "Soil data" command from "Data" menu in the window of Figure 13-22. The following dialog box in Figure 13-23 with default-boring log data appears

Soil data							
Boring log No. 1 from 1 boring logs: Layer No. 1 from 1 layers:	Geotechnical data of the layer:						
Main soil type 1 T, Clay Main soil type 2 -, No symbole	Soil properties are defined by Modulus of Elasticity E						
submain soil 1 -, No symbole	E [kN/m2] 10000 Fhi [°] 0 W [kN/m2] 30000 c [kN/m2] 0						
Color vi, violet	Gam [kN/m3] 18 Nue [-] 0.3						
Short text	Layer depth under the ground surface [m] 5,50						
Copy Layer Insert Layer Delete Layer							
Copy Boring log	X-coordinate of boring [m] 2.00						
Delete Boring     From a file     Insert Boring     Y-coordinate of boring     [m]     3.50       Delete Boring     Boring Log Label     BPN1							
- <u>-</u>							
Qk <u>C</u> ancel	<u>N</u> ew <u>H</u> elp						

Figure 13-23 "Soil data" dialog box with default-boring log data

In the "Geotechnical data of the layer" dialog group box in Figure 13-23 define the geotechnical data of the first soil layer of the three boring logs as follows:

$E_s$	= 10000	$[kN/m^2]$
$W_s$	= 30000	$[kN/m^2]$
$\gamma_s$	= 18	$[kN/m^3]$
$v_s$	= 0.3	[-]

In the current example, the angle of internal friction  $\varphi$  and the cohesion *c* of the soil are not required. Therefore, the user can enter zero values of the internal friction and the cohesion.

$$\begin{array}{l} \varphi_s &= 0 & [°]\\ c &= 0 & [kN/m^2] \end{array}$$

The layer depth under the ground surface of the first layer is taken to be 5.5 [m]. Now, type this value in "Layer depth under the ground surface" edit box.

In order to draw the soil layers by different symbols according to the German Standard DIN 4023, the soil type and color for each layer must be defined.

To define the soil type and color for the first layer, select "T, Clay" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box, Figure 13-23. The color of the clay according to the German Standard DIN 4023 will be automatically created as violet. The user can change this color. Also, a short text "T" will be automatically created for the clay.

To define the second layer

- Click "Layer copy" button in Figure 13-23. A layer that has the same properties of the first layer will be copied
- Use the vertical scrollbar to move to the second soil layer. Layer No. will be typed automatically at the upper-left corner of the main dialog box of soil layers as a head title
- Select "Sst, Sandstone" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box, Figure 13-23. The color of the sandstone according to the German Standard DIN 4023 will be automatically created as orange. The user can change this color. Also, a short text "Sst" will be automatically created for the sandstone
- Change the geotechnical data of the second soil layer of the three boring logs from the values of the first layer to the values as follows:

$E_s$	= 160000	$[kN/m^2]$
$W_s$	=400000	$[kN/m^2]$
$\gamma_s$	= 21	$[kN/m^3]$
$v_s$	= 0.3	[-]
$\varphi_s$	= 0	[°]
С	= 0	$[kN/m^2]$

- Change the value of the layer depth under the ground surface from 5.5 [m] to 14 [m]

After editing the geotechnical data for the first boring log, the boring coordinates and labels which describe the boring will be entered.

To enter the boring coordinates and label

- Type 2 for *x*-coordinate in "*x*-coordinate of boring log [m]" edit box
- Type 3.5 for *y*-coordinate in "*y*-coordinate of boring log [m]" edit box
- Type BPN1 as a label name for the first boring in "Label of boring log" edit box

Now all data and parameters for the first boring log have been entered. The next step is to enter the data of the other two boring logs. As the three boring logs contain the same soil layers, data of the other two boring logs are created by first copying the data of the first boring log and then modifying boring logs individually. Only boring coordinates and labels are required to be modified. To create the other two boring logs, click twice "Boring log copy" button in the Figure 13-23. Two boring logs with the same data and parameters of the first boring log will be copied.

#### Modifying data of boring logs

Modifying boring coordinates is carried out only numerically while modifying the other data of boring logs may be carried out either numerically or graphically. In this example all data will be modified numerically.

To modify the boring coordinates and labels

- Use the horizontal scrollbar to switch to the second boring log. Boring log No. will be typed automatically at the upper-left corner of the main dialog box of boring logs as a head title
- Type 20.5 as *x*-coordinate in "*x*-coordinate of boring log [m]" edit box in Figure 13-23
- Type 2.5 as y-coordinate in "y-coordinate of boring log [m]" edit box in Figure 13-23
- Type BPN2 as a label name for the second boring in "Label of boring log" dialog box in Figure 13-23
- In "Layer depth under the ground surface" edit box in Figure 13-23, type the value 6.3 [m] for the first layer depth under the ground surface

Repeat the previous steps to modify the boring data for the third boring log. The data, which are required to be modified for the third boring log, are:

<i>x</i> -coordinate of boring log [m]	= 17
y-coordinate of boring log [m]	= 13.5
Label of boring log	= BPN3
Layer depth under the ground surface (1 <sup>st</sup> layer) [m]	= 7

Now, after finishing the creation of boring logs, click "OK" button in "Soil data" dialog box in Figure 13-23 to see the defined boring logs on the screen where the user can control or modify the input data and parameters. As a default plot parameter, *ELPLA* displays only the first boring log on the screen, Figure 13-24.

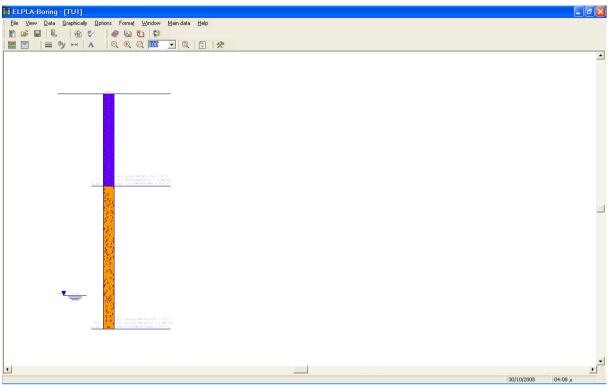


Figure 13-24 First boring log on the screen

To display all boring logs or specified boring logs on the screen, choose "Boring logs" command from "Graphically" menu in Figure 13-24, the following list box in Figure 13-25 appears.

List c	of boring l	logs		×
List of	f selected bo	ring logs to draw: —		Ok
No.	Boring log	No. Boring Log La	ibel	<u> </u>
1		1 BPN1		Cancel
				Help
List of	f the available	e boring logs:		New
Bo	ring log No.	Boring Log Label		
	1	BPN1		
	2	BPN2		Insert Boring
	3	BPN3		
				Delete Bori <u>ng</u>

Figure 13-25 "List of boring logs" list box

To select the boring logs you want to display

- Select the boring log that is required to be displayed from the list of the available boring logs in Figure 13-25
- Click "Boring insert" button. Double clicking on the required boring log in the list of the available boring logs gives the same action. Removing a boring log from the drawing list is carried out by double clicking on that boring log in the list of the selected boring logs to draw or by clicking the "Boring insert" button
- Click "OK" button in Figure 13-25. The selected boring logs appear on the screen to control or modify the boring data graphically, Figure 13-26

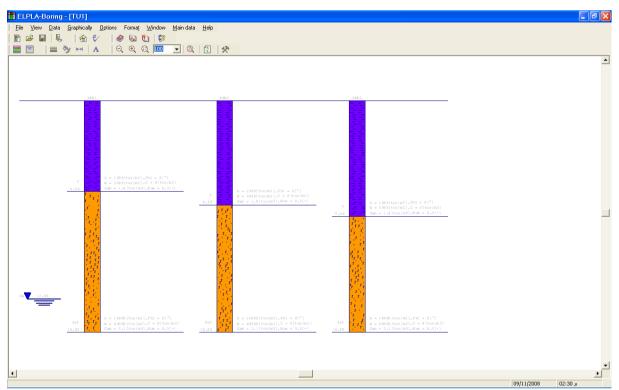


Figure 13-26 Boring logs on the screen

To enter the main soil data for all layers, choose "Main soil data" command from "Data" menu in Figure 13-26. The following dialog box in Figure 13-27 appears with default main soil data. The main soil data for the current example, which are required to be defined, are the settlement reduction factor  $\alpha$  [-] and the groundwater depth under the ground surface  $G_w$  [m]. Any other data corresponding to main soil data are not required in this example. Therefore, the user can take these data from the default soil properties.

In the dialog box of Figure 13-27, enter the settlement reduction factor  $\alpha = 1$  [-] and the groundwater depth under the ground surface  $G_w = 12$  [m]. Then click "OK" button.

Main Soil Data			$\mathbf{X}$
Soil properties Calculation parameters of flexibility coefficients	Bearing capacity fact	ors	
Main Soil Data:			
Settlement reduction factor Alfa <= 1	Alfa	[-]	1
Groundwater depth under the ground surface	Gw	[m]	2.00
<u>Ok</u> <u>C</u> ancel	Help		

Figure 13-27 "Main soil data" dialog box

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

- Choose "Save boring logs" command from "File" menu in Figure 13-26 to save the data of boring logs
- Choose "Close boring logs" command from "File" menu in Figure 13-26 to close the *ELPLA-Boring* sub program and to return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside the "Soil properties" command in the "Data" menu of *ELPLA-Data*.

## 2.6 Foundation properties

To define the foundation properties, choose "Foundation properties" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 13-28 appears with default foundation properties. The data of foundation properties for the current example, which are required to define, are foundation material, foundation thickness and foundation level. Any other data corresponding to foundation properties in the program menus are not required. Therefore, the user can take these data from the default foundation properties.

•			B												
•	•	•		•			•	•	•						
•							•								
•															
•															
•															
•							•								
•															
	-						-	•							

Figure 13-28 "Foundation properties" embedded program

To enter the foundation material and thickness

- Choose "Element groups" command from "In Table" menu in the window of Figure 13-28. The following list box in Figure 13-29 with default data appears. To enter or modify a value in this list box, type that value in the corresponding cell then press "Enter" key. In the list box of Figure 13-29, enter E-Modulus of the foundation  $E_c = 3E + 07$  [kN/m<sup>2</sup>], *Poisson's* ratio of the foundation  $v_c = 0.2$  [-] and foundation thickness  $d_c = 0.5$  [m]
- Click "OK" button

Definir	ig elemen	t groups (with	the same thi	ckn	ess and 🔀
Group No.	E-Modulus of slab [kN/m2]	Poisson's ratio of slab [-]	Slab thickness d [m]		<u>Ok</u> Cancel
1	3E+07	0.2	0.5		
					Insert
					⊆ору
					Delete
					New
					Help

Figure 13-29 "Defining element groups" list box

To enter the unit weight of the foundation

- Choose "Unit weight of the foundation" command from "Foundation properties" menu in the window of Figure 13-28. The following dialog box in Figure 13-30 with a default unit weight of 25 [kN/m<sup>3</sup>] appears; let this value as given to take the own weight of the foundation in consideration in the analysis
- Click "OK" button

Unit weight of the foundati	ion 🛛 🔀
Unit weight of the foundation	Gb [kN/m3] 25
<u>Ok</u> New	<u>Cancel</u> <u>H</u> elp

Figure 13-30 "Unit weight of the foundation" dialog box

To enter the foundation level

- Choose "Foundation depth command from "Foundation properties" menu in the window of Figure 13-28. The following dialog box in Figure 13-31 appears
- In this dialog box type 2.5 in the "Foundation depth under ground surface (a)/ (b)" edit box
- Click "OK" button

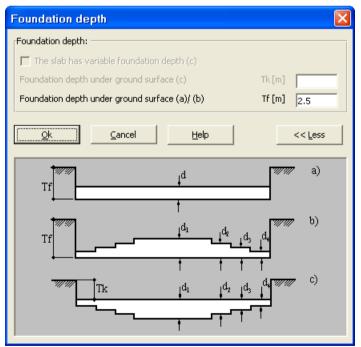


Figure 13-31 "Foundation depth" dialog box

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

- Choose "Save foundation properties" command from "File" menu in Figure 13-28 to save the foundation properties
- Choose "Close foundation properties" command from "File" menu in Figure 13-28 to close the "Foundation properties" embedded program and to return to the main window of *ELPLA-Data*

#### 2.7 Boring fields

If the subsoil under the raft is defined by two boring logs or more such as in the current example, the irregularity of the subsoil must be taken into account. "Boring fields" command let the user define which method is to be used to consider the irregularity of the subsoil. In the current example, the Interpolation Method will be used.

To consider the Interpolation Method in the analysis, choose "Boring fields" command from "Data" menu. The following embedded program in Figure 13-32 appears with a default method. *ELPLA* considers that the Interpolation Method is the default one, which takes into account the irregularity of the subsoil. In most cases *ELPLA* defines the interpolation zone types I, II, III automatically such as in this example. But in the case of extreme boring arrangements, the user must define these zones.

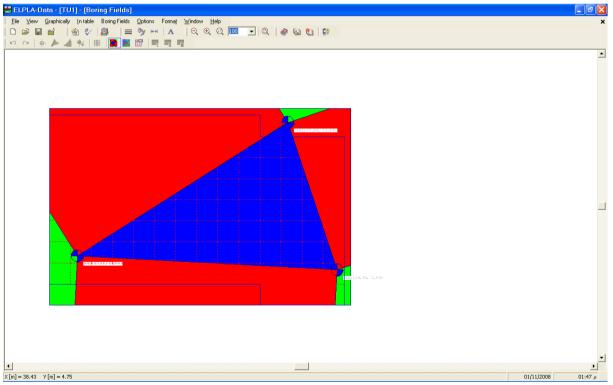


Figure 13-32 "Boring fields" embedded program

You do not need to change anything. Now, do the following two steps:

- Choose "Save boring fields" command from "File" menu in Figure 13-32 to save the data of boring fields
- Choose "Close boring fields" command from "File" menu in Figure 13-32 to close the "Boring fields" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside "Boring fields" command in "Data" menu of *ELPLA-Data*.

#### 2.8 Loads

In *ELPLA*, loads on the raft such as point loads, line loads, uniform loads or moments may be applied to the net of the finite elements at any position independently on the node position. The coordinates of the loads are related to the lower-left corner of the raft (local coordinates).

To define the loads, choose "Loads" command from "Data" menu. The following embedded program in Figure 13-33 appears.

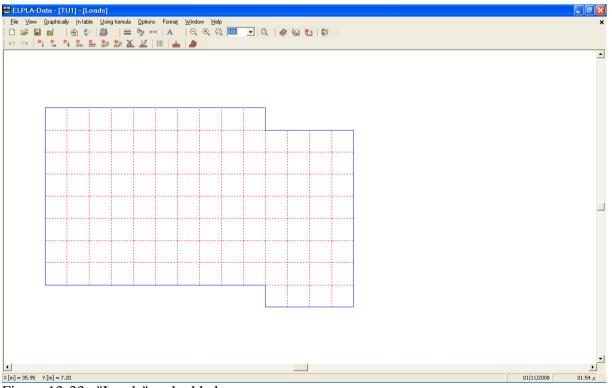


Figure 13-33 "Loads" embedded program

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Graphically" menu in Figure 13-33. In this example the user will learn the definition of loads numerically.

The foundation loads consist of two concentrated loads of P = 18 [MN] and line load of p = 300 [kN/m] on the external walls.

To neglect the dimensions of columns at foundation level

- Choose "Column types" command from "In Table" menu in the window of Figure 13-33. The following list box in Figure 13-35 appears with default dimensions a = 0.5 [m] and b = 0.5 [m]
- To neglect the dimensions of the columns in the analysis, type 0 in the column side *a* and column side *b* cells
- Click "OK" button

(	Colum	n types		X
	Group No.	Column side a [m]	Column side b [m]	Qk
	1	0.00	0.00	<u>C</u> ancel
				Insert
				⊆ору
				Delete
				New
				Help

Figure 13-34 "Column types" list box

To enter concentrated loads

- Choose "Point loads" command from "In Table" menu in the window of Figure 13-33. The following list box in Figure 13-35 appears
- Enter the external point loads *P* [KN] and their corresponding coordinates (*x*, *y*) in the list box of Figure 13-35. This is done by typing the value in the corresponding cell and then press "Enter" key. The coordinates of the point load are related to the lower-left corner of the foundation (local coordinates)
- Click "OK" button Point loads Column types Load X-position Y-position No. <u>O</u>k [m] [-] [-] [kN] [m] Cancel 18000 1 7.50 7.50 1 2 18000 15.00 7.50 Insert ⊆ору Delete New <u>H</u>elp

Figure 13-35 "Point loads P" list box

Repeat the previous steps for line loads using "Line loads" command from "In Table" menu. After you have completed the definition of all load data, the screen should look like the following Figure 13-36.

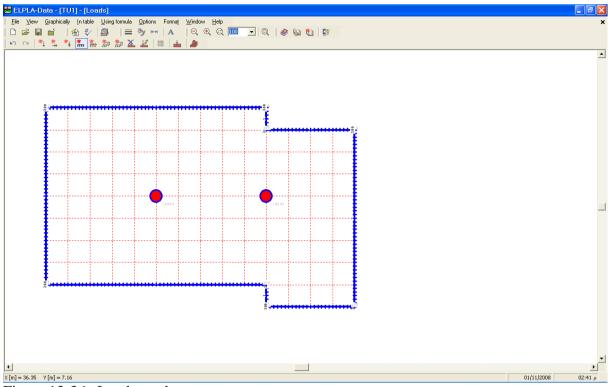


Figure 13-36 Loads on the screen

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

- Choose "Save loads" command from "File" menu in Figure 13-36 to save the load data
- Choose "Close loads" command from "File" menu in Figure 13-36 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside "Loads" command in "Data" menu of *ELPLA-Data*.

The project creation of the first case without the underground tunnel is now completed.

#### 3 Creating the project of the second case (including the underground tunnel)

The data of the two projects are quite similar except the effect of the tunnel. Project identification is entered so that you can distinguish between the two projects. The data of the second case are created by first saving the data of the first case under a new file name and then modifying the project identification.

To save the data under a new file name

- Choose "Save project as" command from "File" menu of *ELPLA-Data*. The following "Save as" dialog box appears, Figure 13-37

In this dialog box

- Type a file name for the project of the second case in the file name edit box. For example type "TU2"
- Click "Save" button

Save As	? 🛛
Save in: ն	TU 🔹 🗲 🖆 🎟 -
TU1 TU2	
File name:	TU2 Save
Save as type:	Isolated slab foundation-files (*.P01)

Figure 13-37 "Save as" dialog box

#### **3.1** Modifying the calculation methods options

To add the option of influence of additional settlements on the raft do the following steps:

- Choose "Calculation methods" command from "Data" menu
- The first form of the wizard "Calculation method" is the "Analysis type" form, Figure 13-2. In this form, click "Next" button to go to the next page
- After clicking "Next" button, the "Calculation methods" form appears, Figure 13-3. Click "Next" button to go to the next form
- The next form is the "System symmetry" (Figure 13-4). In this form click "Next" button
- The last form of the wizard assistant contains the "Option" list, Figure 13-38. In this list, check the option "Influence of additional settlements on raft", then click the "Save" button

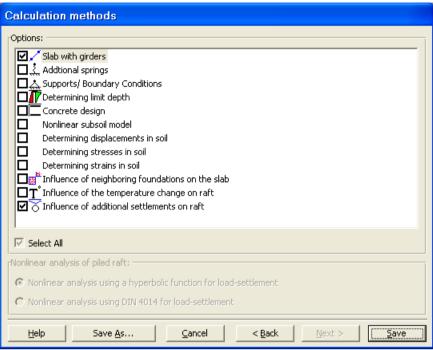


Figure 13-38 "Options" list

### **3.2** Modifying the project identification

To modify the project identification, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 13-39 appears. In this dialog box, type "With tunnel" in the "Project" edit box then click "Save" button.

Project	Identification	K					
Project Id	entification:						
Title	Influence of ground lowering due to a tunnel on a building						
Date	28 October 2008						
Project	With tunnel						
<u>S</u> ave	Gancel Help Load Save As						

Figure 13-39 "Project identification" dialog box

#### **3.3** Additional settlements

To define the settlements caused by the tunnel, choose "Additional settlements" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 13-40 appears.

● 奏:	<b>名 ∜</b> 둘   Ⅲ	5													
			 	 	 				1						
									-						
							_								

Figure 13-40 "Additional settlements" embedded program

To define the settlement values

- Choose "Additional settlements" command from "In Table" menu in the window of Figure 13-28. The following list box in Figure 13-41 appears
- In this list box type the node number and the corresponding settlement value
- Click "OK" button

Additional Settlements 🛛 🛛 🔀							
	lo. I	Node No.	Influence of additional settlements on raft	-	<u>Ok</u>		
	-]		Ss [cm]		<u>C</u> ancel		
	1	8	0.01		Incort		
	2	9	0.44		Insert		
	3	10	1.34		⊆opy		
Ŀ	4	11	2.76				
Ľ	5	12	2.86		Delete		
	6	13	1.46				
Ľ	7	14	0.56		New		
	8	15	0.04		<u>14644</u>		
	9	24	0.16		Help		
1	10	25	0.96				
	1	26	2.08	Ţ			
	-	27	2.00	<u> </u>			

Figure 13-41 "Additional settlements" list box

After you have completed the definition of the additional settlements, the screen should look like the following Figure 13-42

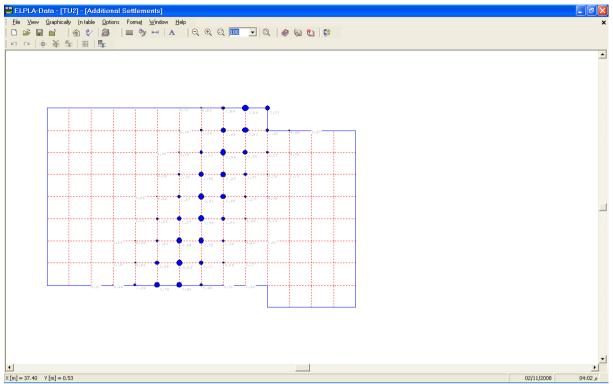


Figure 13-42 Additional settlements on the screen

After defining the additional settlements, do the following two steps:

- Choose "Save additional settlements" command from "File" menu in Figure 13-42 to save the additional settlements
- Choose "Close additional settlements" command from "File" menu in Figure 13-42 to close the "Additional settlements" embedded program and return to *ELPLA-Data*

Note that the sign " $\sqrt{}$ " is typed automatically beside "Additional settlements" command in "Data" menu of *ELPLA-Data*.

#### 4 Carrying out the calculations

The calculations of the first case should be first done, and then the calculations of the second case are carried out.

#### 4.1 Starting *ELPLA-Solver*

To analyze the problem, open the file "TU1" from "File" menu from *ELPLA-Data*, Then, leave *ELPLA-Data* to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window appears, Figure 13-43.

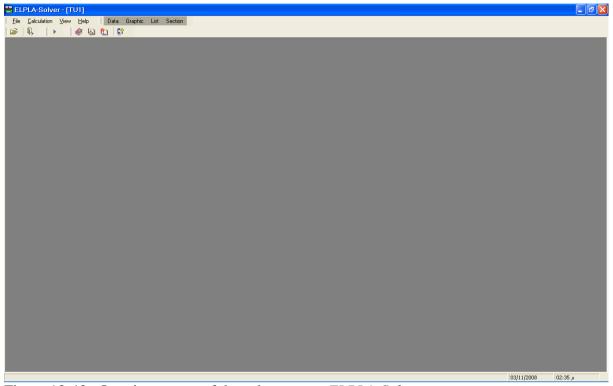


Figure 13-43 Opening screen of the sub program ELPLA-Solver

*ELPLA-Solver* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. The "Calculation" menu contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis.

For the project of the first case, the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Assembling girder stiffness matrix
- Assembling the slab stiffness matrix
- Solving the system of linear equations (Unsymmetrical matrix)
- Determining deformation, internal forces, contact pressures

While that for the project of the second case, the items, which are required to be calculated, are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Influence of additional settlements on raft
- Assembling girder stiffness matrix
- Assembling the slab stiffness matrix
- Solving the system of linear equations (Unsymmetrical matrix)
- Determining deformation, internal forces, contact pressures

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

#### 4.2 Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Calculation" menu in *ELPLA-Solver* window. The following "Iteration parameters" option box in Figure 13-44 appears
- In "Iteration parameters" option box, select which option is ending the iteration process. For this example, choose an accuracy of 0.0001 [m] to end the iteration process
- Click "OK" button

Iteration parameters							
Which option is ending the iteration process?							
• Accuracy [m]	0.0001						
C Iteration No.	10						
Qk <u>C</u> ancel	Help						

Figure 13-44 "Iteration parameters" option box

The progress of all computations according to the defined method will be carried out automatically with displaying information through menus and messages.

#### Analysis progress

Analysis progress menu in Figure 13-45 appears in which various phases of the calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.

Determining flexibility coefficients of the soil	×
Assembling the flexibility matrix!	
Time remaining = 00:00:01           I = 74 from 136 steps	;;

Figure 13-45 Analysis progress menu

#### **Iteration process**

Information about the convergence progress of the computations is displayed in the "Iteration process" list box in Figure 13-46 during the iteration process.

Iteration pro	ICess					
Iteration No.	Accuracy [m]	Stop				
1	0.02288176000	]  ,				
2	0.00044275620	Pause				
		Help				
Iteration cycles is ended at accuracy [m]<= 0.0001 Computation time = 00:00:00 Performing iteration!						

Figure 13-46 "Iteration process" list box

#### Check of the solution

Once the analysis is complete, a check menu of the solution appears, Figure 13-47. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

Check of the solution	
V - Load	
Total load	[kN] = 61403
Sum of contact pressures	[kN] = 61389
X - Moment	
Sum M× from loads	[kN.m] = 12630
Sum Mx from contact pressures	[kN.m] = 12642
Y - Moment	
Sum My from loads	[kN.m] = 31354
Sum My from contact pressures	[kN.m] = 31360
Ok Help	

Figure 13-47 Menu "Check of the solution" in the first case

To finish analyzing the first case of the problem, click "OK" button.

Now the analysis of the first case without the tunnel has ended, after that we should analysis the second case.

To analyze the second case of the example, open the file "TU2" from "File" menu in *ELPLA-Solver* and choose "Computation of all" command from "Calculation" menu in *ELPLA-Solver* window. After the analysis is complete, the "Check of the solution" menu appears (Figure 13-48). Click the "OK" button to finish the analysis of the second case of the problem.

Check of the solution	
V - Load	
Total load	[kN] = 61437
Sum of contact pressures	[kN] = 61414
X - Moment	
Sum M× from loads	[kN.m] = 12625
Sum Mx from contact pressures	[kN.m] = 12612
Y - Moment	
Sum My from loads	[kN.m] = 31361
Sum My from contact pressures	[kN.m] = 31382
Ok <u>H</u> elp	

Figure 13-48 Menu "Check of the solution" in the second case

#### 5 Viewing data and results

*ELPLA* can view and print a wide variety of results in graphics, diagrams or tables through the three sub programs *ELPLA-Graphic*, *ELPLA-Section* and *ELPLA-List*. Data can also be viewed again and printed by the sub programs *ELPLA-Graphic* and *ELPLA-List*. Note that *ELPLA-Data* is used only to define and view the data of the problem. *ELPLA-Graphic* is used to print data graphically while *ELPLA-List* is used to print data numerically.

#### 5.1 Viewing result graphics

To view the data and results of the first case of the problem that has already been defined and analyzed graphically, open the file "TU1" from "File" menu from *ELPLA-Solver*. Then, leave *ELPLA-Solver* to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window.

*ELPLA-Graphic* window appears, Figure 13-49. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.

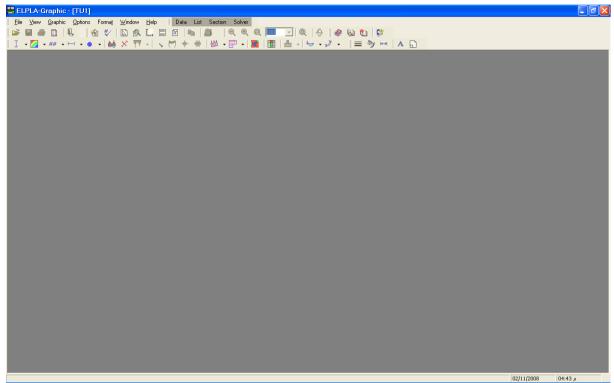


Figure 13-49 Opening screen of the sub program *ELPLA-Graphic* 

To view the results for the raft without the effect of the underground tunnel as contour lines

- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 13-50 appears
- In "Results as contour lines" option box, select "Settlements" as a sample for the results to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for the raft only before constructing the tunnel as shown in Figure 13-51.

Results as Contour Lines	
Select one item to draw	
Settlements	C Contact pressure q
C Moments mx	C Moments my
C Moments mxy	C Shear forces Qx
C Shear forces Qy	Qk
C Modulus of subgrade reaction ks	Cancel
C Principal moments hm1	
C Principal moments hm2	Help

Figure 13-50 "Results as contour lines" option box

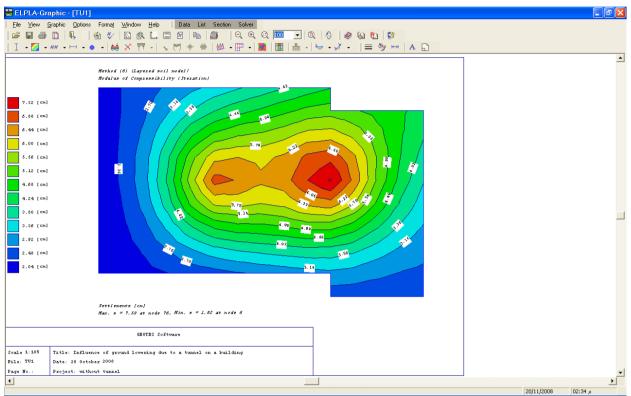


Figure 13-51 Settlements as contour lines (first case)

To view the settlement results for the raft with the effect of the underground tunnel as contour lines

- Choose "TU2" file from "File" menu from *ELPLA- Graphic*
- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The option box in Figure 13-50 appears
- In "Results as contour lines" option box, select "Settlements" as a sample for the results to be displayed
- Click "OK" button

The settlements are now displayed as contour lines for the second case of the analysis as shown in Figure 13-52.

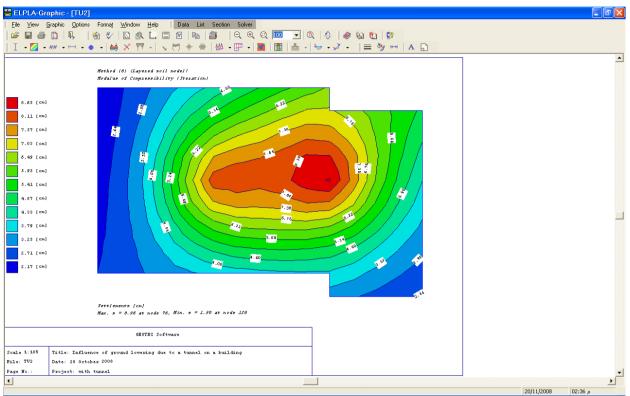


Figure 13-52 Settlements as contour lines for the raft (second case)

### 5.2 Plot a diagram of results

To plot a diagram of results, leave *ELPLA-Graphic* to *ELPLA-Section*. This is done by clicking on "Section" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Graphic* window. *ELPLA-Section* window appears, Figure 13-53.

*ELPLA-Section* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Section* window.

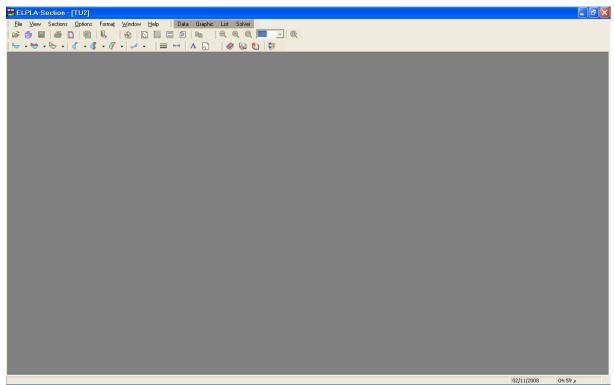


Figure 13-53 Opening screen of the sub program ELPLA-Section

To consider the results for the two cases together

- Choose "Combination from many projects" command from "File" menu of *ELPLA-Section*. The following list box in Figure 13-54 appears. Select the project "TU1" in the list of projects to be combined
- Click "Add project" button in "Combination from many projects" list box. Then open the project "TU2"
- Click "OK" button in the "Combination from many projects" list box

С	omb	ination from many pr	ojects	×
ſL	ist of I	projects to be combined		<u>o</u> k
	No.	File name of the project	Project Identification	
	1	E:\Omar\K6\TU2	With tunnel	Cancel
	2	E:\Omar\Analysis of	Without tunnel	
				A <u>d</u> d Project
				Remove Project
				New
				Help

Figure 13-54 "Combination from many projects" list box

To plot a section in *x*-direction

- Choose "Section in *x*-direction" command from "Sections" menu. The following option box in Figure 13-55 appears
- In "Section in *x*-direction" option box, select "Settlements" as a sample for the results to be plotted in a diagram
- Click "OK" button

Section in x-direction	
Select one item to draw	
Settlements	C Contact pressure q
C Moments mx	O Moments my
C Moments mxy	C Shear forces Qx
C Shear forces Qy	
C Modulus of subgrade reaction ks	Qk
C Additional soil settlements Ss	Cancel
C Principal moments hm1	
C Principal moments hm2	Help

Figure 13-55 "Section in *x*-direction" option box

The following dialog box in Figure 13-56 appears to specify the section in x-direction that is required to be plotted.

In this dialog box

- Type 7.5 in the "Section at *y*-coordinate" edit box to plot a diagram at the horizontal axis of the columns
- Click "OK" button

The settlements are now plotted in a diagram at a section across the columns of the raft in the two cases together as shown in Figure 13-57.

The user can repeat the following steps to display the critical results of contact pressure as shown in Figure 13-58 or displaying the critical results of bending moment in x-direction as shown in Figure 13-59. The critical results which are shown located at y = 7.5 [m].

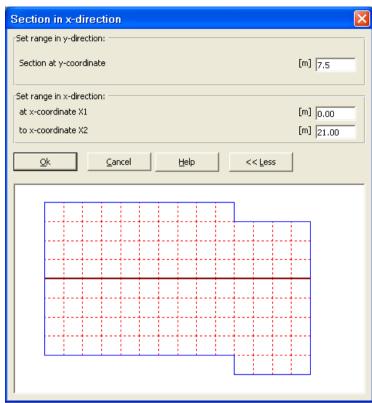


Figure 13-56 "Section in *x*-direction" dialog box

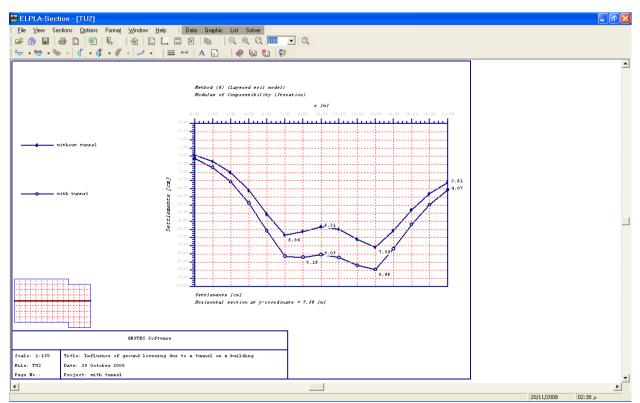


Figure 13-57 Diagram of settlements in x-direction at y = 7.5 [m] for the two cases

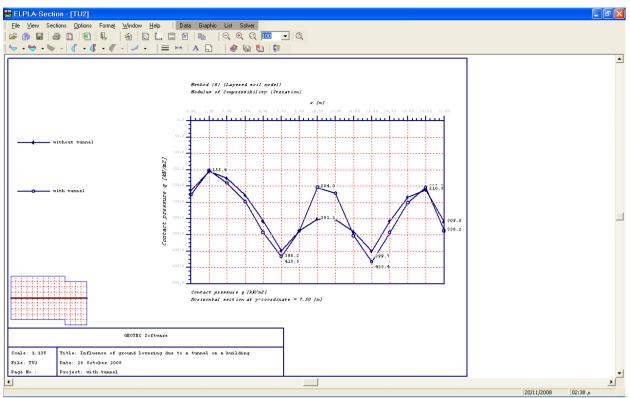


Figure 13-58 Diagram of contact pressure in x-direction at y = 7.5 [m] of the two cases

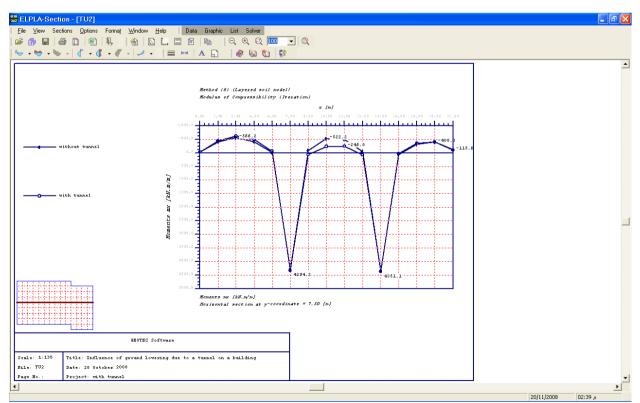


Figure 13-59 Diagram of bending moment  $M_x$  in x-direction at y = 7.5 [m] of the two cases

#### Index 6

## A

Analysis ......5, 6, 19. 36, 38

## B

Boiring fields	28
Boring log19, 20, 21,	

## C

Calculation	
Calculation methods	
Check of the solution	
Computation of all	
Contact pressures	
Contour lines	
Coordinates	
Cross section	

## D

Deformation	36
Diagram	42

# E

ELPLA-Data 6, 9, 10, 11, 15, 19,	25, 28, 31, 32, 39
ELPLA-Graphic	
ELPLA-Section	
ELPLA-Solver	
E-Modulus	

## F

FE-Net	
File6, 9, 15	, 25, 28, 29, 31, 32, 36, 39, 41
Flexibility coefficients	
Foundation	5, 6, 11, 12, 25, 26, 27, 30
Foundation properties	

## G

Generation parameters	
Girder locations	
Girders	15, 19
Graphic	
*	

## Ι

Internal forces	6, 36
Iteration process	37

## K

Key	
L	

Load vector	36
Loads	3, 29

## M

Main soil data	.20,24
Modulus of compressibility for loading	5
Modulus of compressibility for reloading	5

## N

Nodes	

## 0

Options6, 29
--------------

## P

Point loads	29, 30
Poisson's ratio	5, 26
Project identification	

## R

Remove nodes
--------------

## S

Save	8
Save project as	
Select nodes	
Slab type	11
Soil properties	
Soil stiffness matrix	
System symmetry	

## T

Tutorial manual		1
-----------------	--	---

# U

Unit weight	5, 13, 27
Unsymmetrical system	