User’s Guide
for the program ELPLA

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

Version 9.2

Part A	Description of the program ELPLA
Part B	New features and enhancements in the program ELPLA
Part C	User’s Guide for the program ELPLA-Data
Part D	User’s Guide for the program ELPLA-Solver
Part E	User’s Guide for the program ELPLA-Graphic
Part F	User’s Guide for the program ELPLA-Section
Part G	User’s Guide for the program ELPLA-List
Part H	User’s Guide for the program ELPLA-Boring
Part I	GEOTEC-Editor

Program authors: M. El Gendy
A. El Gendy

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

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

Part A

Short description of the program package 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

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

Contents Page

1 Introduction

ELPLA (ELASTIC PLATE) is a program for analyzing raft foundations of arbitrary shape with the real subsoil model. The mathematical solution of the raft is based on the FE-Method. The program can analyze different types of subsoil models, especially the three-dimensional Continuum model that considers any number of irregular layers. A good advantage of this program is the capability to handle the three analyses of flexible, elastic and rigid foundations. In addition, the mesh of the rigid and flexible foundations can be constructed to be analogous to the finite elements mesh of the elastic foundation. Therefore the three analyses can be compared easily and correctly. ELPLA can also be used to represent the effect of external loads, neighboring foundations, tunneling and the influence of the temperature difference on the raft.

ELPLA is a 32-bit, graphical software product that operates under Microsoft Windows 9x/ NT/ ME/ XP. The common "what you see is what you get" of Windows applications makes it easy to learn how to use ELPLA, especially if you are already familiar with the Windows environment.

The program package ELPLA consists of 7 separate programs. These can run independently. The names and short descriptions of the separate programs are shown in Table A-1.

Table A-1 Names and descriptions of ELPLA programs

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

2 Calculation methods

In ELPLA 9 different numerical methods with 3 soil models are considered for analyzing raft foundations as follows:

1) Linear contact pressure
   (Simple assumption model)

2) Constant modulus of subgrade reaction
   (*Winkler*'s model)

3) Variable modulus of subgrade reaction
   (*Winkler*'s model)

4) Modification of modulus of subgrade reaction by iteration
   (*Winkler*'s model/ Continuum model)

5) Modulus of compressibility method for elastic raft on half-space soil medium
   (Solving system of linear equations by elimination)
   (Isotropic elastic half-space soil medium - Continuum model)

6) Modulus of compressibility method for elastic raft
   (Solving system of linear equations by iteration)
   (Isotropic elastic half-space soil medium and layered soil medium - Continuum model)

7) Modulus of compressibility method for elastic raft on layered soil medium
   (Solving system of linear equations by elimination)
   (Layered soil medium - Continuum model)

8) Modulus of compressibility method for rigid raft
   (Isotropic elastic half-space soil medium and layered soil medium - Continuum model)

9) Modulus of compressibility method for flexible raft
   (Isotropic elastic half-space soil medium and layered soil medium- Continuum model)

Beside the above 9 main methods ELPLA can also be used to analyze system of flexible, elastic or rigid foundations. Furthermore ELPLA can be used to analyze many other structural problems such as slab floors, grids, plane frame and plane stress.

3 Geometry and loads

It is possible to consider raft with any arbitrary shape including holes, Figure A-1. It is also possible to consider raft with variable thickness, Figure A-2. Loads on the raft can be applied independently on the mesh at any position. Loads may be defined in different types such as point loads, line loads and polygon uniform loads, Figure A-3.

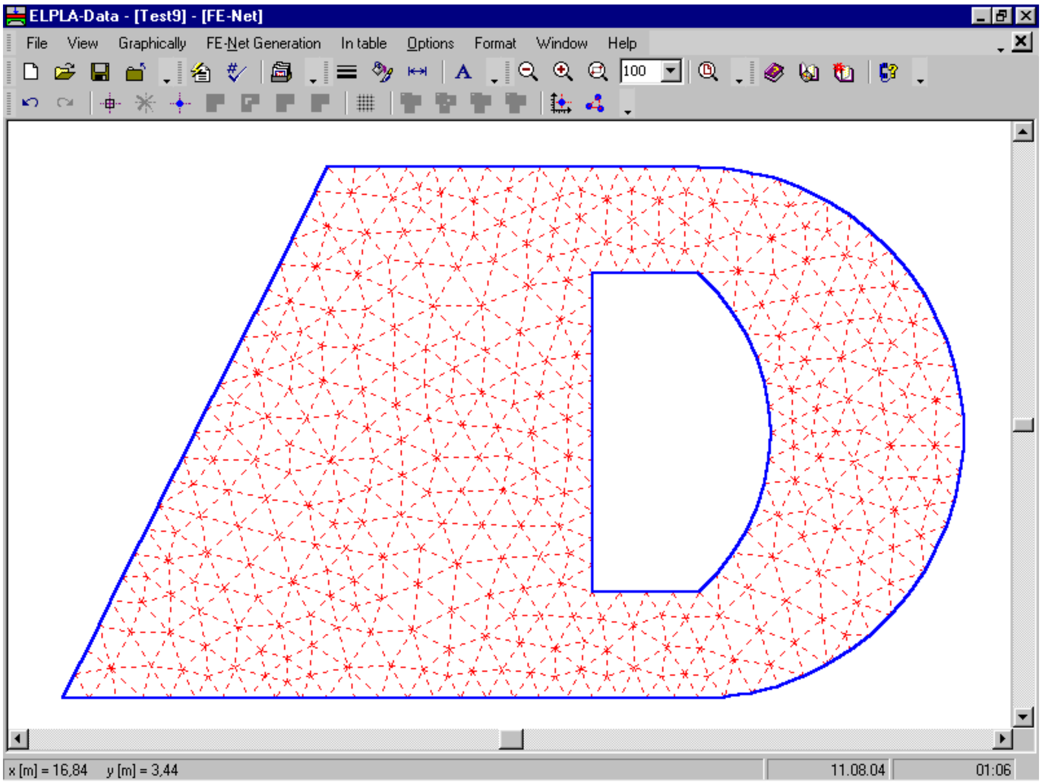


Figure A-1 Arbitrary shape of raft with hole

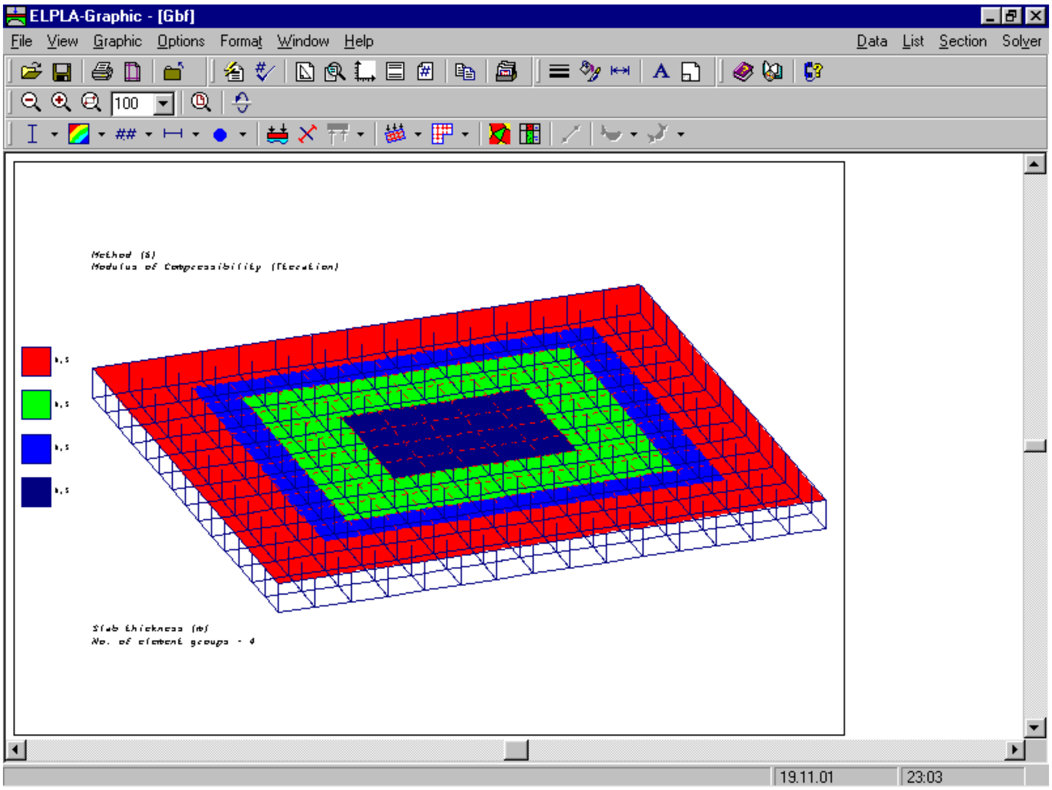


Figure A-2 Variable slab thickness

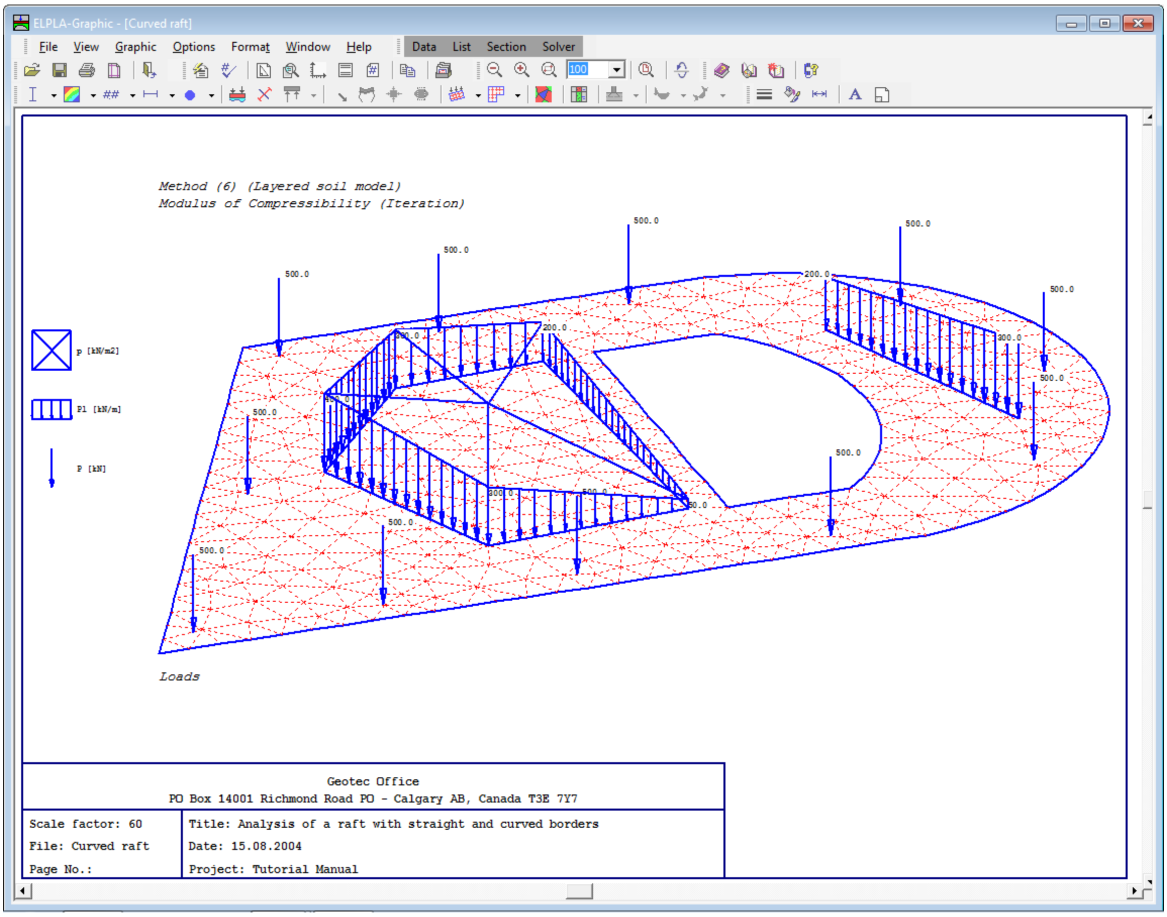


Figure A-3 Arbitrary type of loads

4 Boundary conditions

It is possible to define elastic or fixed rotations and displacements on the raft, Figure A-4. Also translational or rotational springs may be defined.

5 Soil

The soil is defined by a number of borings; each boring has multi-layers with different soil materials, Figure A-5. Variable thickness and discontinuous soil strata can be considered, Figure A-6. Loading and reloading of the soil modulus can be taken into account by the analysis, Figure A-7. Three different methods are used to determine the flexibility coefficients or the modulus of subgrade reaction:

1 Hand-Division of boring logs to nodes
2 Subareas method
3 Interpolation method (Figure A-5)

It is possible to draw soil layers by different symbols according to the German Standard DIN 4023 for easy identification. Also the limit depth of soil layers can be determined. Variable foundation levels can be considered in the analysis (Figure A-6).

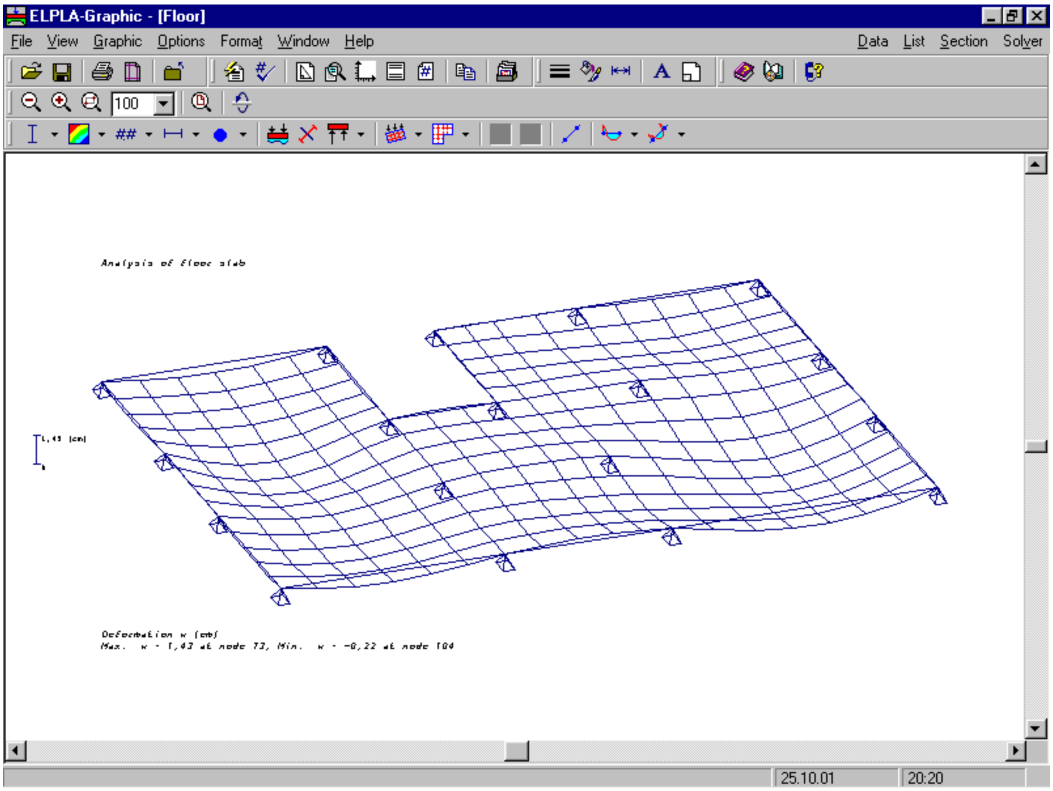


Figure A-4 Fixed displacements

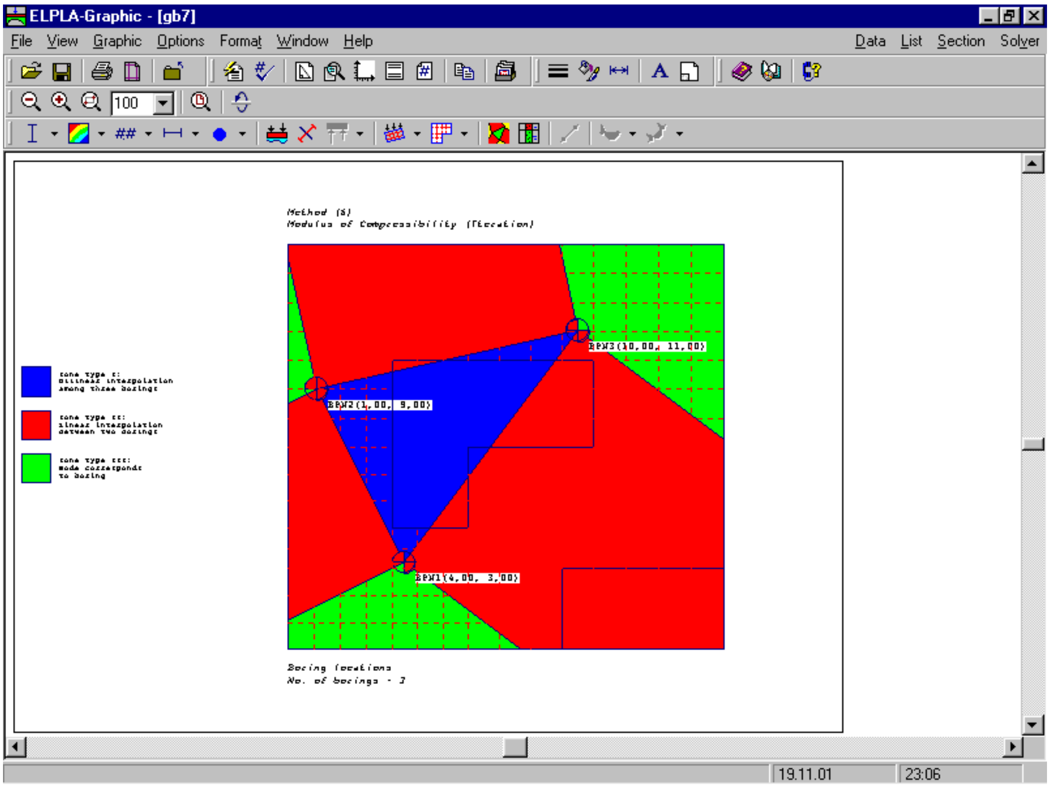


Figure A-5 The soil is defined by a number of borings

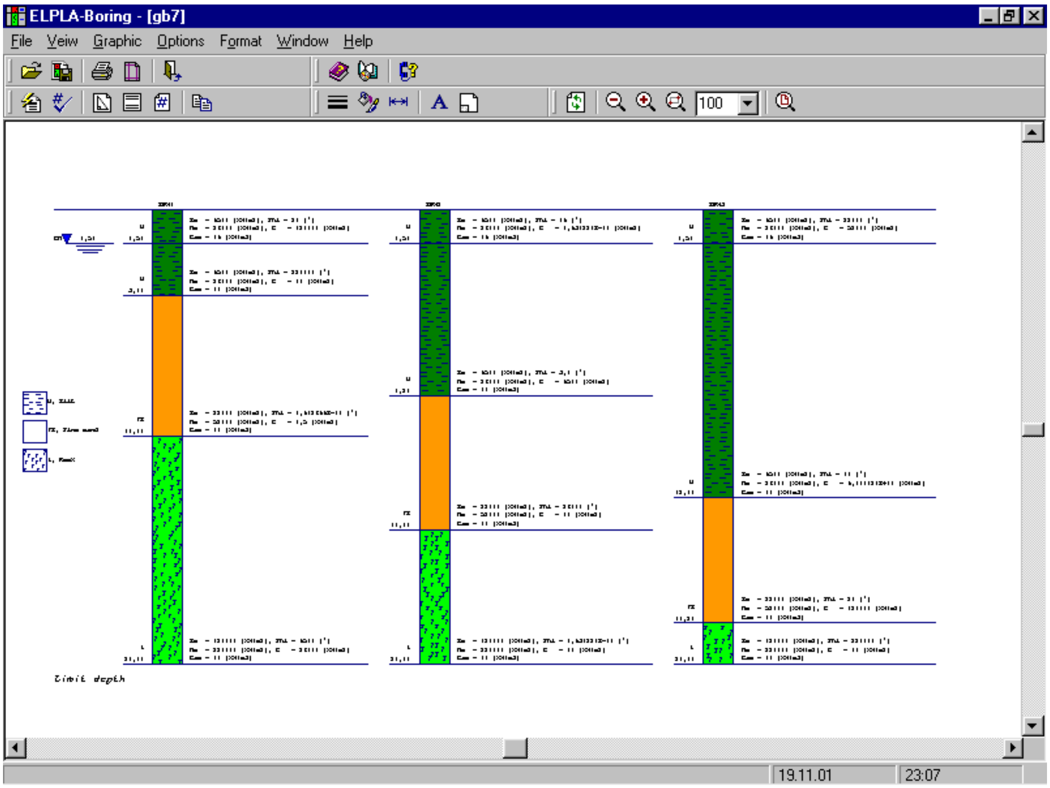


Figure A-6 Variable thickness soil strata

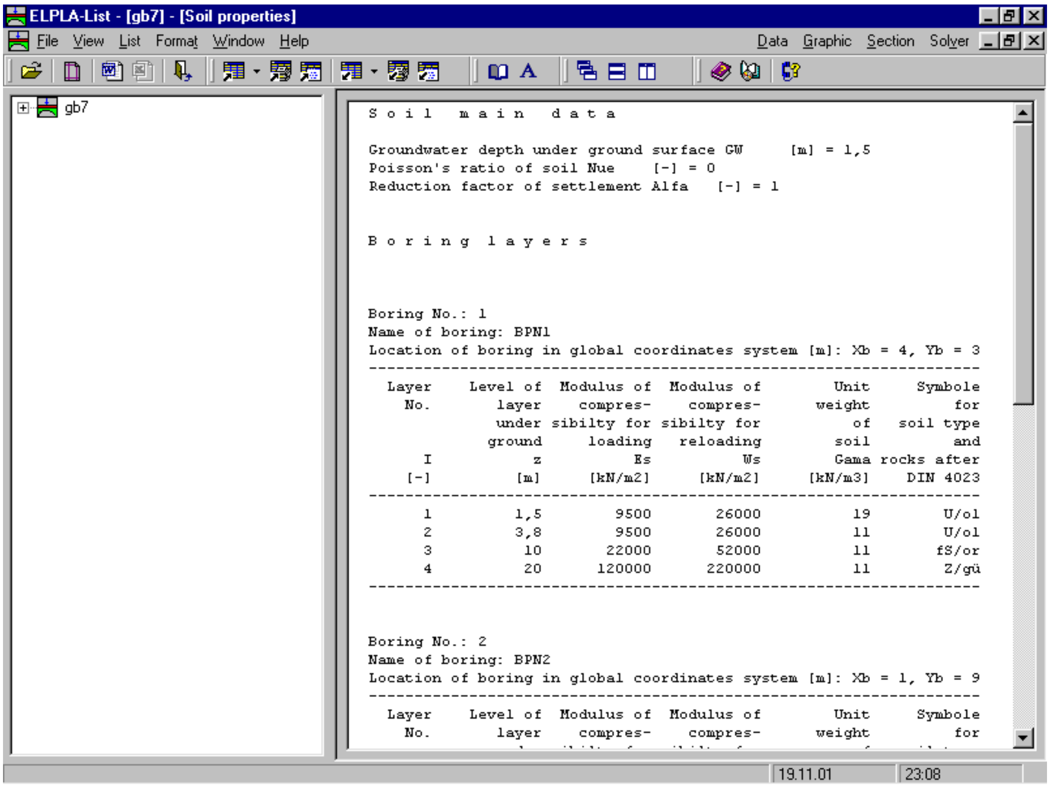


Figure A-7 Loading and reloading soil modulus are considered

6 Design of the slab

The design of the slab for determining reinforcement and punching stress can be carried out according to the following design codes:

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

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

- ACI — American Concrete Institute Building Code Requirements for Structural Concrete

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

7 Graphical drawing of data and results

You can display, plot and print data and results graphically using the sub program ELPLA-Graphic. It is possible to draw raft geometry, boring locations, soil profiles, loading, boundary conditions, settlement, deformation, contact pressure, moment, shear, modulus of subgrade reaction and reinforcement (Figure A-8 to Figure A-14).

The results and data can be presented graphically as follows:

- Data in the plan
- Data in isometric view
- Boring locations
- Boring logs
- Limit depth
- Arrangement of rafts including neighbor foundations
- Result values in the plan
- Distribution of results in the plan
- Results as contour lines
- Results in isometric view
- Results as circular diagrams
- Principal moments as streaks
- Support reactions as arrows
- Deformation
- Girders

The graphical drawing, if desired, can be saved as WMF-File, in which it can be exported into other Windows applications to prepare reports, slide presentations or add further information to the drawing.

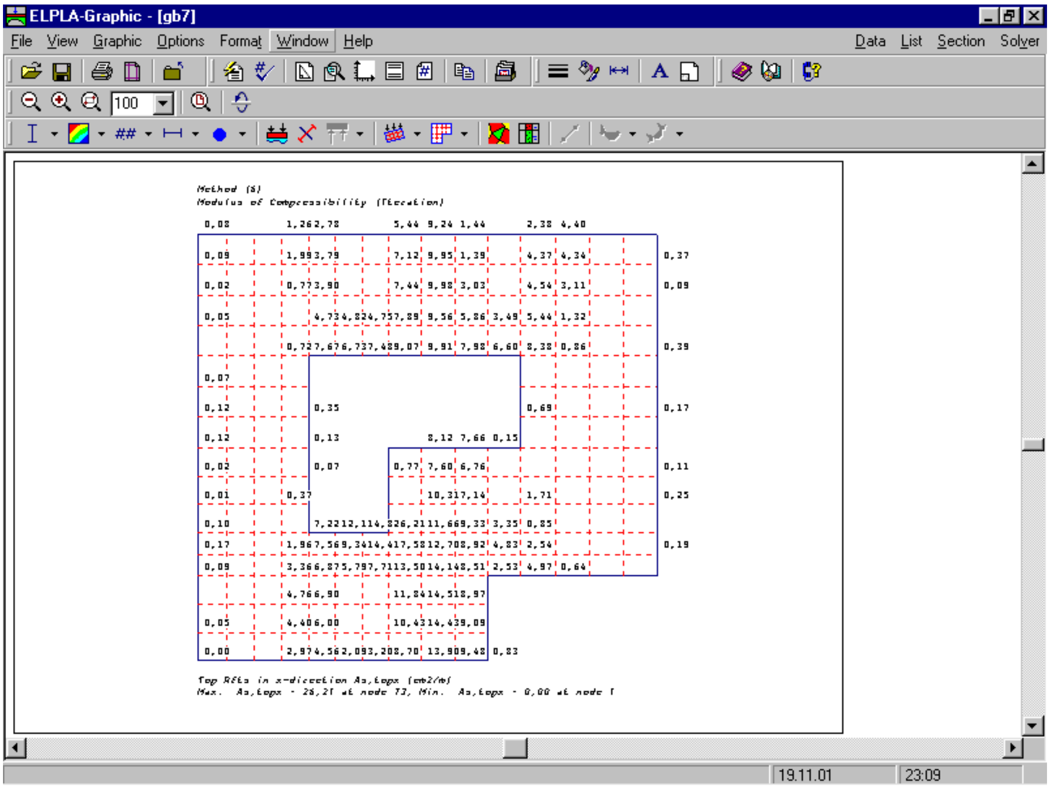


Figure A-8 Results can be tabulated on the mesh

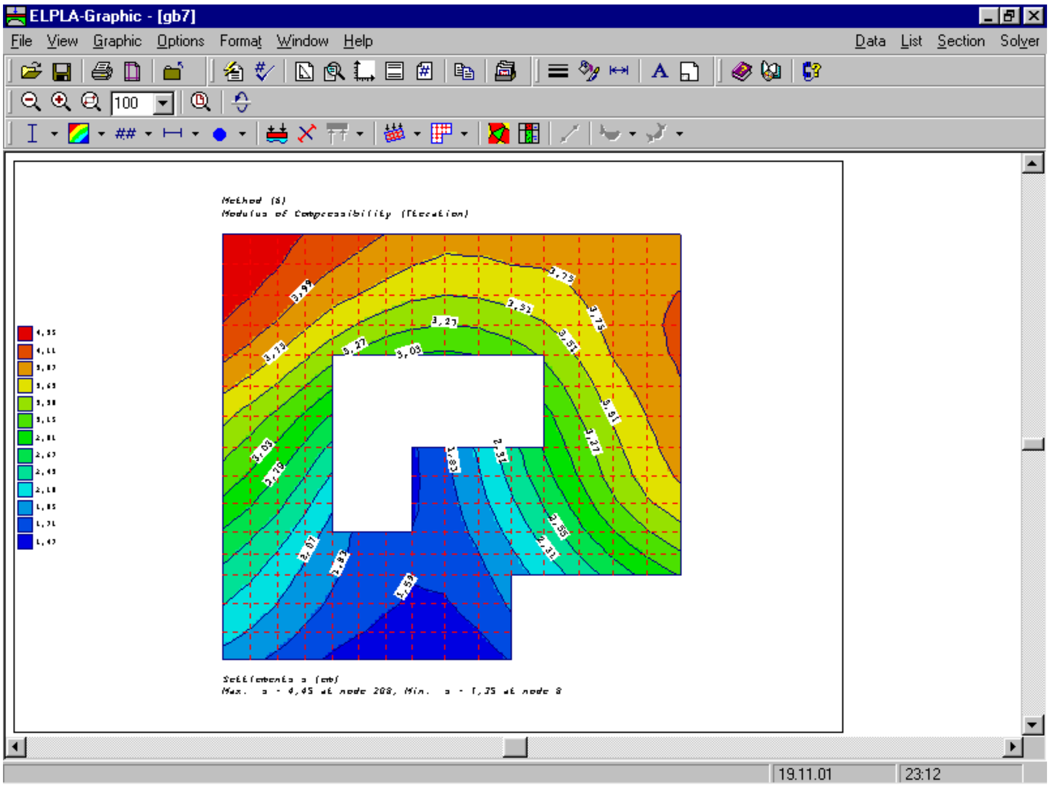


Figure A-9 Results can be contoured

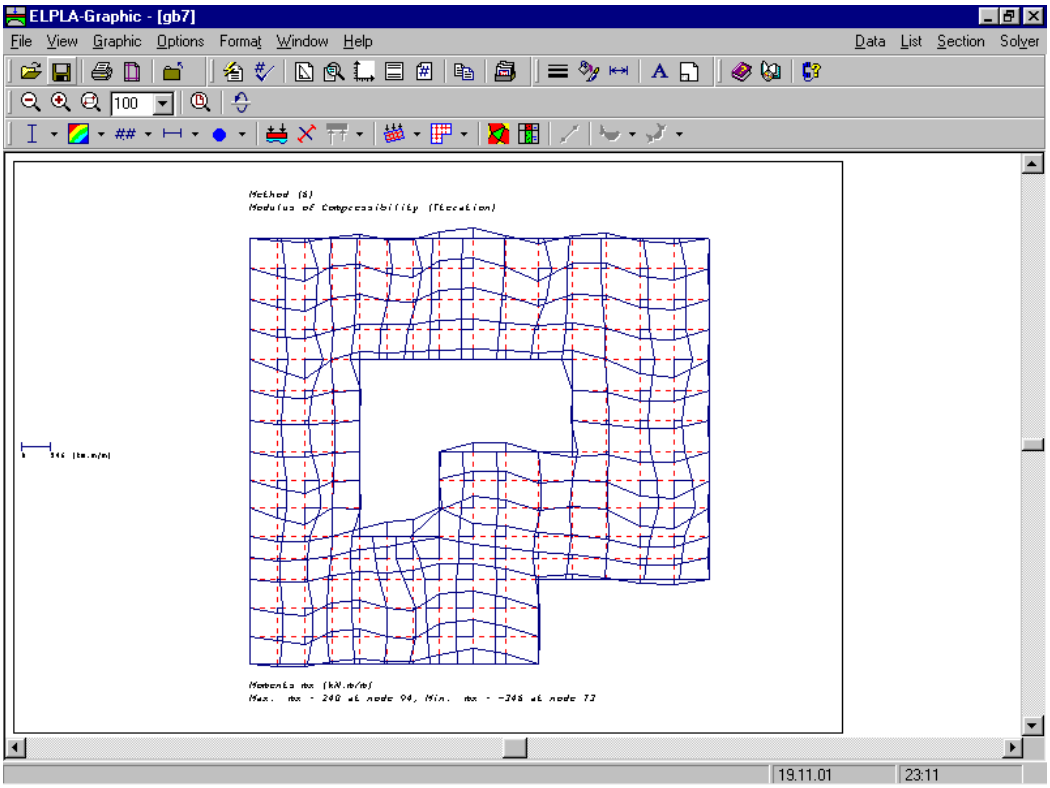


Figure A-10 Moment distribution on the raft can be plotted

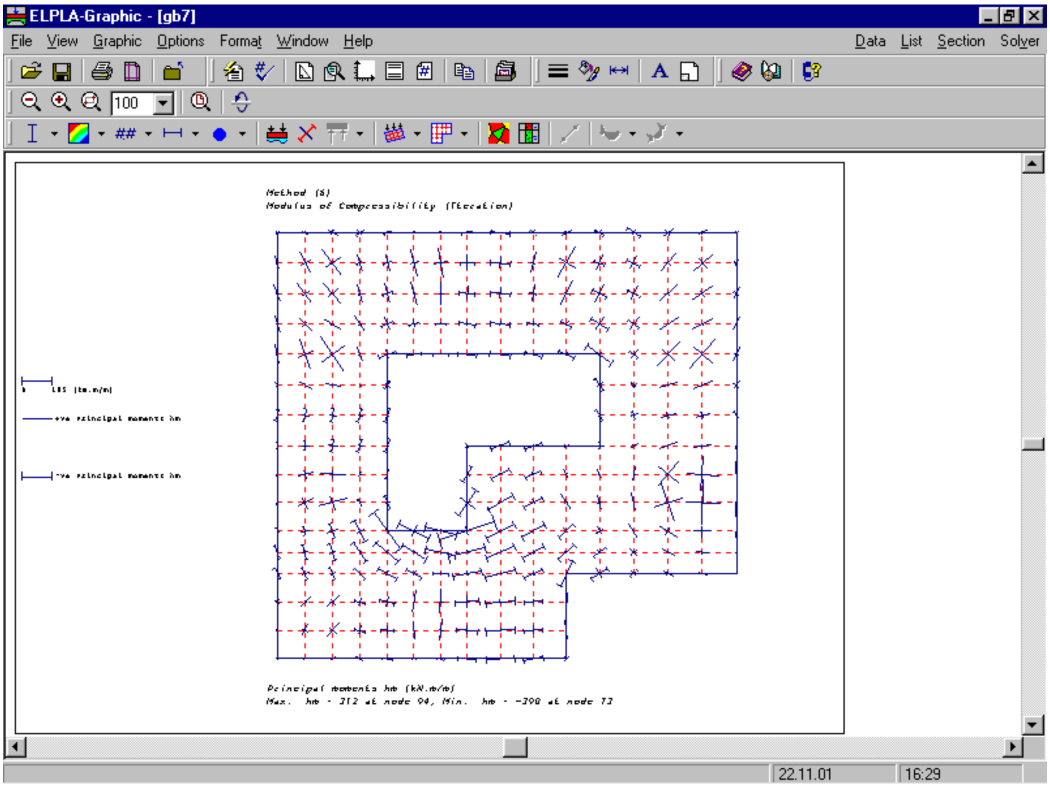


Figure A-11 Principal moments as streaks

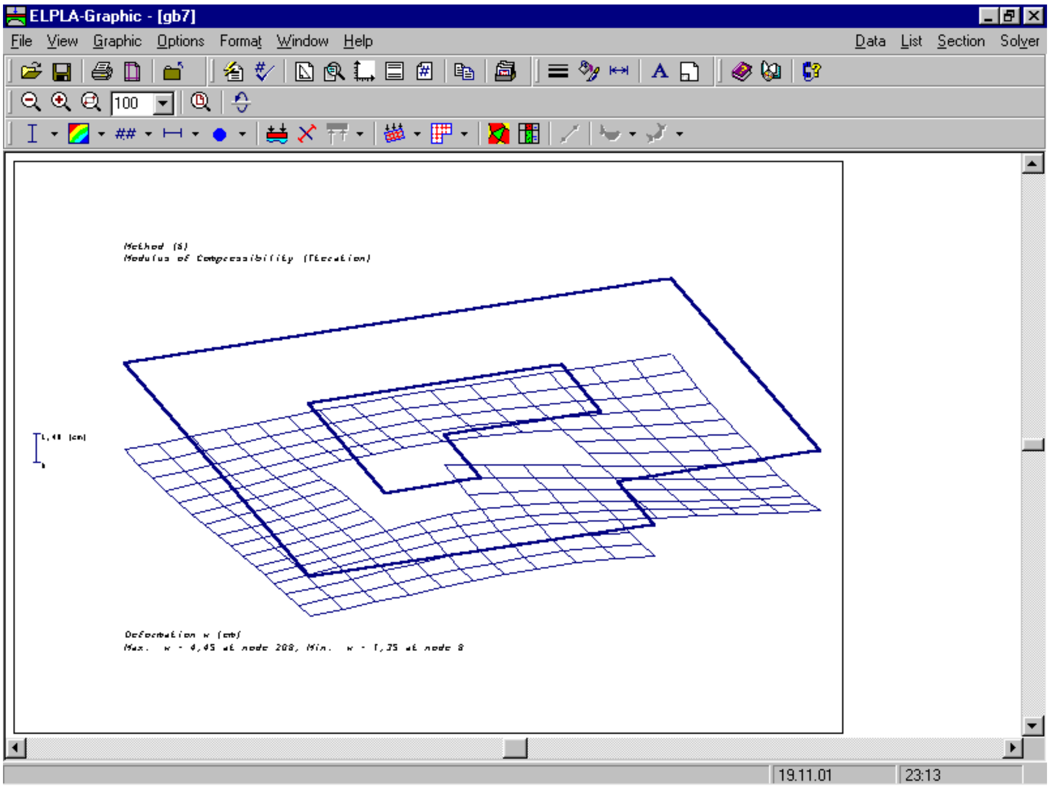


Figure A-12 Raft deformation can be plotted as a deformed mesh

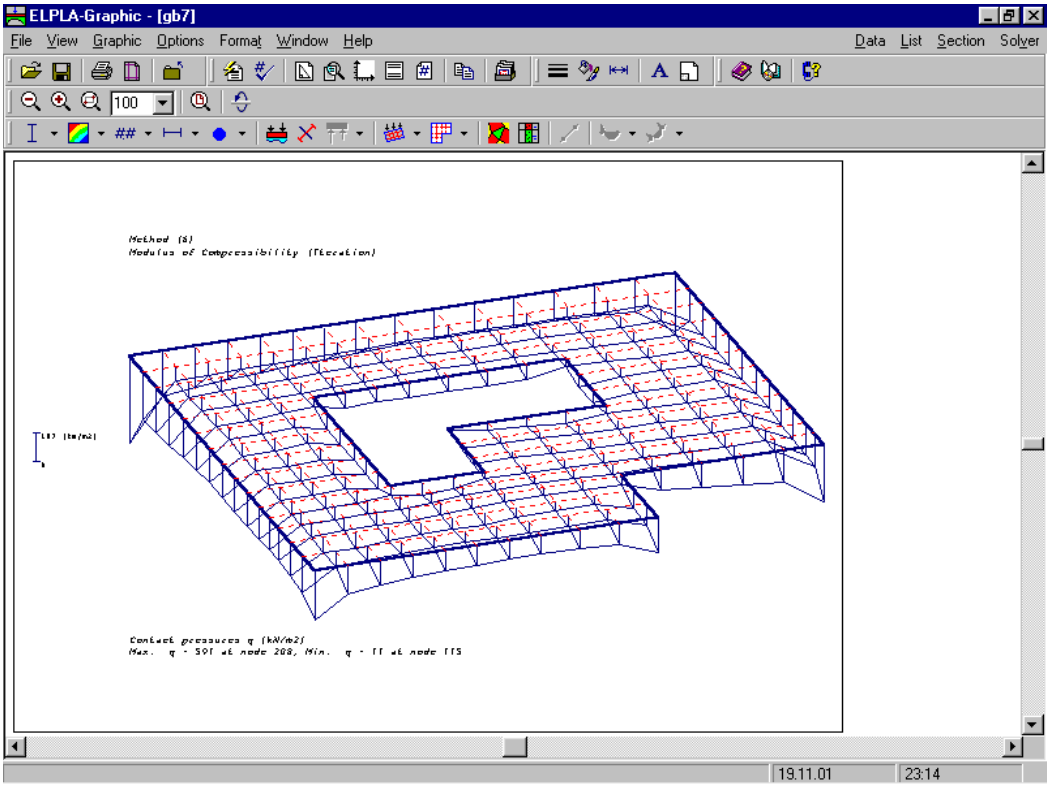


Figure A-13 Results can be plotted in isometric shape

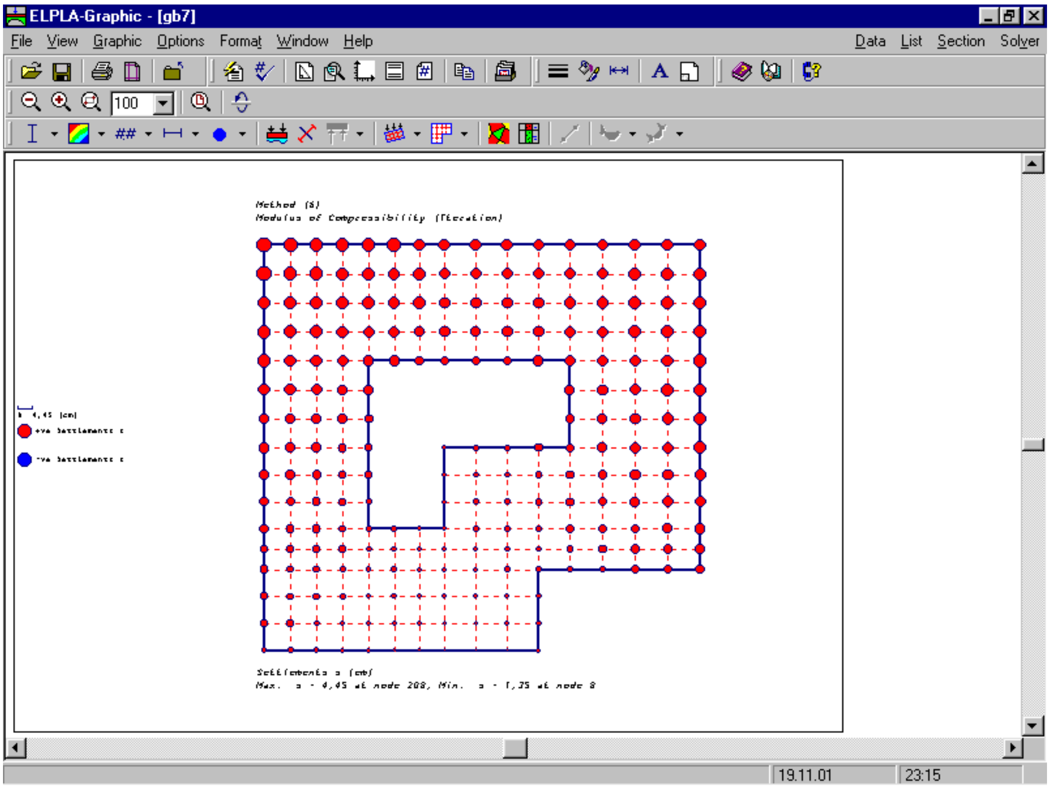


Figure A-14 Results can be plotted as circular diagram

8 Drawing sections

Further more you can display, plot and print results at specified sections graphically using the sub program ELPLA-Section. It is possible to draw settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement (Figure A-15 to Figure A-17). It is also possible to determine extreme values of the results from many load cases. The results can be presented graphically as follows:

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

Also drawing sections, if desired, can be saved as WMF-Format files, in which they can be exported to other Windows applications to prepare reports, slide presentations or add further information.

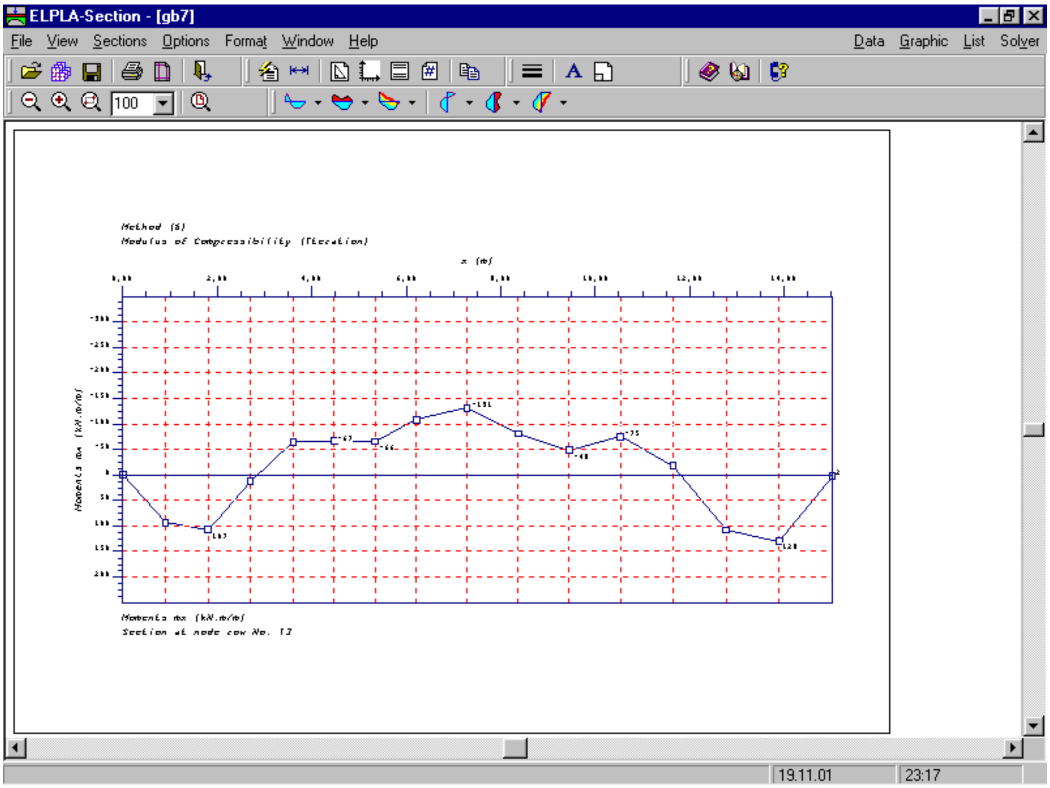


Figure A-15 Results can be plotted at specified section

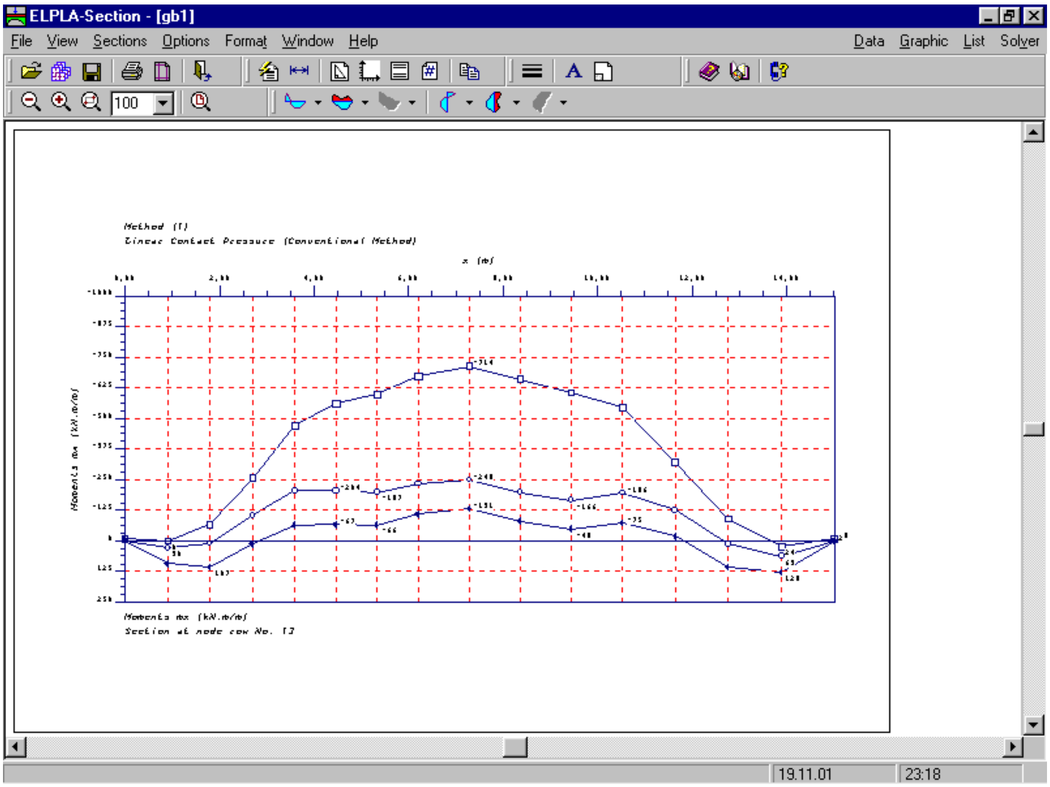


Figure A-16 Results from many projects can be plotted together

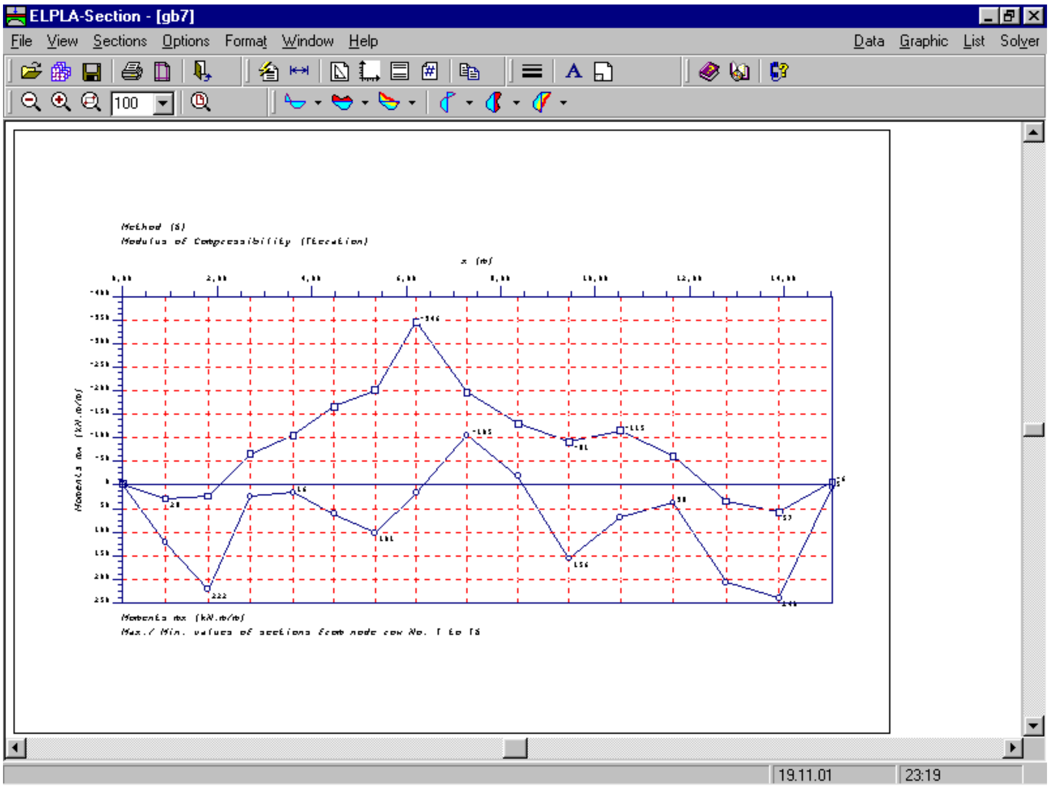


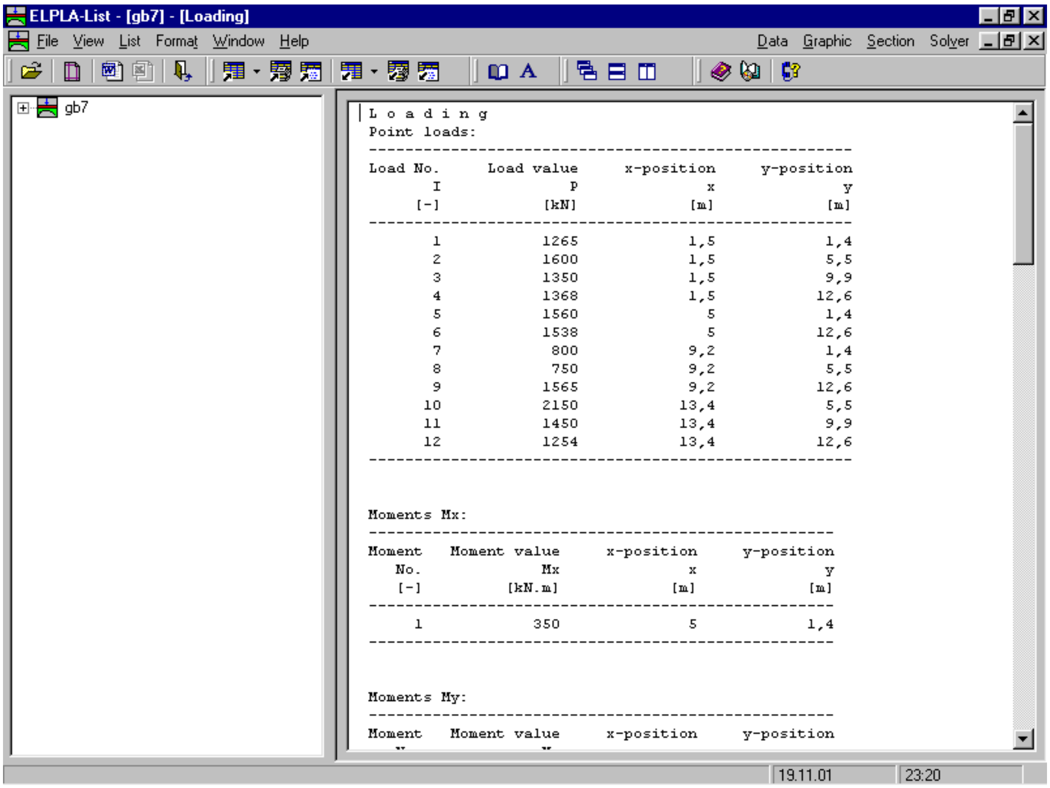
Figure A-17 Max. and Min. values can be calculated and plotted together

9 Tabulation of data and results

You can list data and results using the sub program ELPLA-List. Listing the data and results can be displayed first on the screen and then can be sent to the printer (Figure A-18 to Figure A-20). The results and data can be listed as follows:

- 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

The listed results and data, if desired, can be saved as ASCII-format Files, in which they can be exported to other Windows applications to prepare reports or add further information.



L o a d i n g

Point loads:

Load No. I [-]	Load value P [kN]	x-position x [m]	y-position y [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	5	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

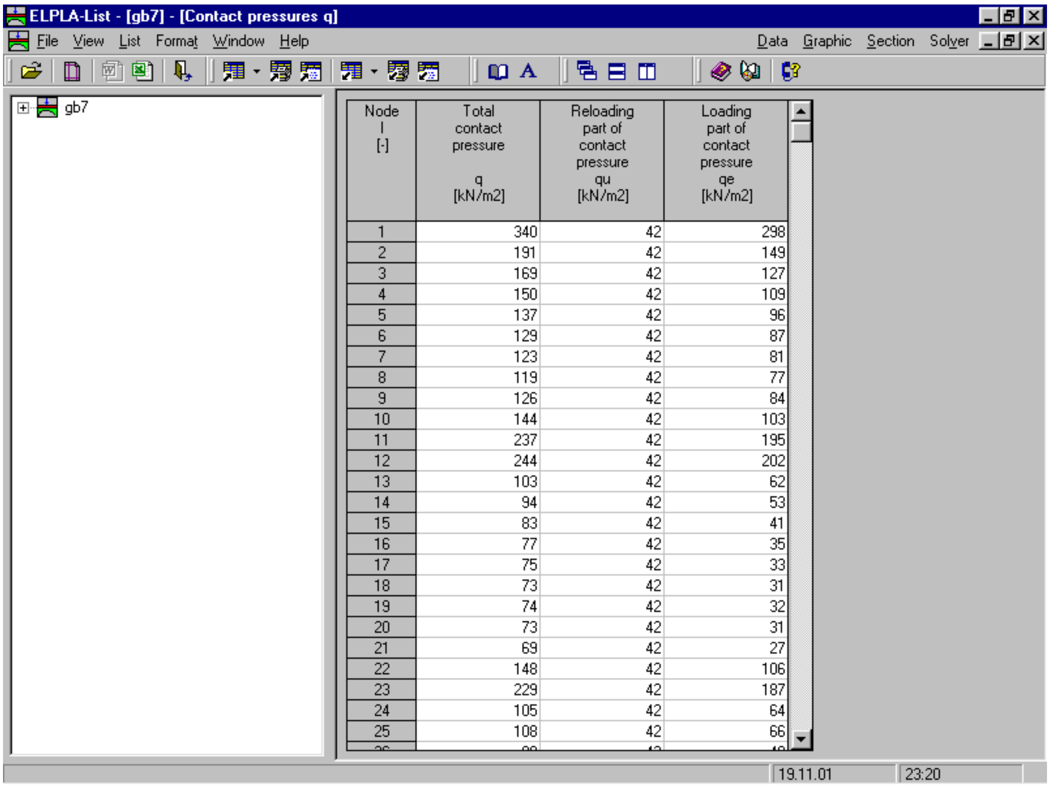
Moments Mx:

Moment No. [-]	Moment value Mx [kN.m]	x-position x [m]	y-position y [m]
1	350	5	1,4

Moments My:

Moment   Moment value   x-position   y-position

Figure A-18 Data can be tabulated



Node I [-]	Total contact pressure q [kN/m2]	Reloading part of contact pressure qu [kN/m2]	Loading part of contact pressure qe [kN/m2]
1	340	42	298
2	191	42	149
3	169	42	127
4	150	42	109
5	137	42	96
6	129	42	87
7	123	42	81
8	119	42	77
9	126	42	84
10	144	42	103
11	237	42	195
12	244	42	202
13	103	42	62
14	94	42	53
15	83	42	41
16	77	42	35
17	75	42	33
18	73	42	31
19	74	42	32
20	73	42	31
21	69	42	27
22	148	42	106
23	229	42	187
24	105	42	64
25	108	42	66

Figure A-19 Results can be tabulated

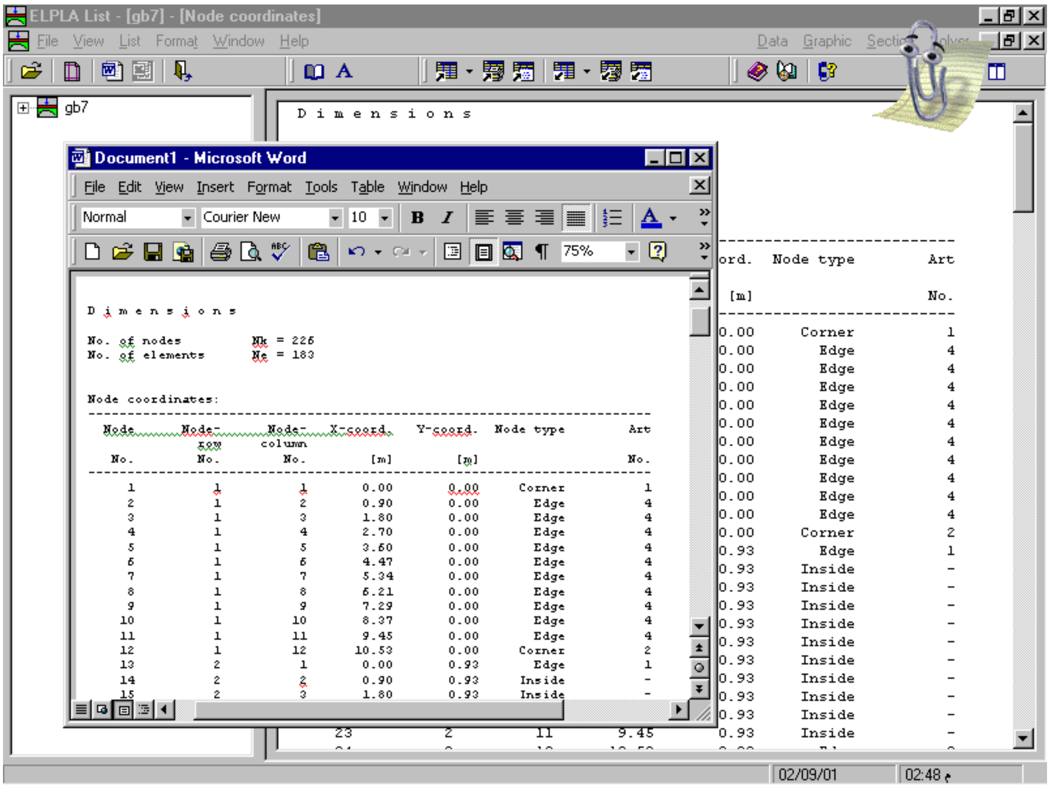


Figure A-20 Data can be imported to other text editor applications

10 Typical applications of ELPLA

* Soil-structure interaction problems
* Analysis and design of rafts
* Analysis of rigid rafts
* Analysis of flexible foundations
* Analysis and design of slab floors
* Determining the consolidation settlements
* Analysis and design of pile caps
* Determining forces on piles due to structure loads
* Settlement calculation of surface foundations
* Determining the settlement due to surcharge fills or surcharge concentrated loads
* Determining the surface settlement around rafts
* Determining the constant or variable modulus of subgrade reaction
* Effect of external loads or neighboring foundations
* Effect of temperature difference
* Effect of tunneling
* Analysis of system of flexible, elastic or rigid foundations
* Analysis of beams or grids by FE-Method
* Simulation of excavations and construction of embankments
* Determining the ultimate bearing capacity of the soil
* Determining the limit depth
* Eliminating negative contact pressure
* Design of slabs according to codes ACI, EC 2, DIN 1045 and ECP
* Determining the stress in soil

11 References

The program is a result of many extensive research works after many authors. Some of these references are:

[1] EL ARABI/ EL GENDY, M. (2001): On the Optimum Design of Foundation Systems
Suez Canal University, Faculty of Engineering, Port-Said
Port-Said Engineering Research Journal, November 2001

[2] EL ARABI/ EL GENDY, M. (2001): Effect of Openings on Raft Behavior
Suez Canal University, Faculty of Engineering, Port-Said
Port-Said Engineering Research Journal, December 2001

[3] CRUZ, L. (1994): Vergleichsuntersuchungen zur Bauwerk-Boden-Wechselwirkung an einer Hochhausgründungsplatte zwischen den nationalen Normen und den Eurocodes
Diplomarbeit, Universität Gesamthochschule Siegen

[4] EL GENDY, A. (1996): Structural analysis and design using finite element method
B. Sc. Project report, Suez Canal University, Port-Said, Egypt

[5] EL GENDY, M. (1994): Comparing examinations of the influence of calculation methods of basement slabs
PH.D Thesis, Suez Canal University, Egypt

[6] EL GENDY, M. (1998): An analysis for determination of foundation rigidity
Eighth International Colloquium on Structural and Geotechnical Engineering
Ain Shams University, Cairo, Egypt

[7] EL GENDY, M. (1998): An iteration method for design of slab on elastic foundation
Proceeding of the first International Conference on Civil Engineering
Helewan University, Cairo, Egypt

[8] EL GENDY, M. (1999): Effect of Girders on the Raft Rigidity
1st International Conference for Advanced Trends in Engineering
Minia University, Minia, Egypt

[9] EL GENDY, M. (2003): Numerical Modeling of Rigid Circular Rafts on Consolidated Clay Deposits
International Workshop on Geotechnics of Soft Soils-Theory and Practice
Noordwijkerhout, The Netherlands

[10] HERRMANN, R. (1994): Konstruktion und Bemessung von Bodenplatten
Nachweis von Grenzzuständen nach EC 2 Teil 1 und EC 7 Teil 1 / DIN V 1054–100
Seminar Universität Gesamthochschule Siegen

[11] IBRAHIM, F./ El GENDY, M./ EL SHERIFY (2002): Analysis of Plates on Compressible Subsoil
2nd International Conference for Advanced Trends in Engineering
Minia University, Minia, Egypt

[12] KANY, M. (1974): Berechnung von Flächengründungen, 2. Auflage
Verlag Ernst & Sohn, Berlin

[13] KANY, M./ EL GENDY, M. (1993): Vergleichende Untersuchung über numerische Modelle für die Berechnung von Gründungsplatten
Theorie und Praxis numerischer Modelle in der Bodenmechanik, Sonthofen
Herausgeber: TU Graz

[14] KANY, M./ EL GENDY, M. (1995): Computing of beam and slab foundations on three Dimensional layered model
Proceeding of the Sixth International Conference on Computing in Civil and Building Engineering, Berlin

[15] KANY, M./ EL GENDY, M. (1996): Sicherheitsuntersuchungen bei Flächengründungen nach EC 7/ DIN 1054
Forschungsbericht an IFBT, Berlin

[16] KANY, M./ EL GENDY, M. (1996): Unterlagen zu den TAW-Seminaren
"Berechnung von Flächengründungen", Nürnberg

[17] KANY, M./ EL GENDY, M. (1997): Analysis of system of footing resting on irregular soil
Proceeding of the XIVth International Conference on Soil Mechanics and Foundation Engineering, Hamburg

[18] KANY, M./ EL GENDY, M. (1999): Berechnung von großen Systemen starrer Sohlplatten
Bauingenieur, Bd. 74, Nr. 11, S. 471-478

[19] KANY, M./ EL GENDY, M. (2000): Einfluss der Bauwerkssteifigkeit auf das Fundamentsystem
2. Kolloquium Bauen in Boden und Fels
Technische Akademie Esslingen, Ostfildern, Germany

[20] KANY, M./ EL GENDY, M. (2002): Berechnung von Fundamenten auf nichtlinearem Baugrund
3. Kolloquium Bauen in Boden und Fels
Technische Akademie Esslingen, Ostfildern, Germany

[21] OHDE, J. (1942): Berechnung der Sohldruckverteilung unter Gründungskörpern
Z. Bauingenieur, S. 99 ff. und S. 102 ff.

Part B

New features and enhancements in the program *ELPLA* 9.1

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

Web site: http://www.elpla.com
e-mail: geotec@elpla.com

Table of Contents

Page

1 Enhancements in *ELPLA* 9.1 (MUI)

1.1 GEOTEC Office Applications with Multilingual User Interface

The new English-language versions of GEOTEC Office applications with Multilingual User Interface (MUI) are now available. The Multilingual User Interface Pack is a set of language XML resource files that can be added to the English version of GEOTEC Office applications. MUI Pack allows the user interface language to be changed according to the preferences of individual users to one of the available supported languages. MUI Pack provides a single version of GEOTEC Office applications to which users can add one or more MUI Packs providing local user interface and help files. Now, three languages are already available in *ELPLA* 9.1; English, German and Arabic.

The major benefits of the new MUI Pack are:

- Allows user interface switching between languages
- Easy to update with new languages
- Language-specific updates do not affect all languages
- Languages are XML based resources that make it easier to users to add their own languages.

1.2 GEOTEC Office Language Settings

Now the user can define the language of the user interface and help system used in GEOTEC Office applications. To select or change GEOTEC Office language setting, start “GEOTEC Office Language Settings” tool by clicking on the program icon in the Windows ***Start-Menu> GEOTEC Office> GEOTEC Office Tools***. The language setting tool (Figure B-1) appears.

- In the “Display menus and dialog boxes in” list box, user can change the language of the menus and dialog boxes used in the GEOTEC Office applications. After selecting a new language, the user must quit and restart any Office applications he is currently using.

- In the “Display Help in” list box, the user can change the language of the Help system used in the GEOTEC Office applications.

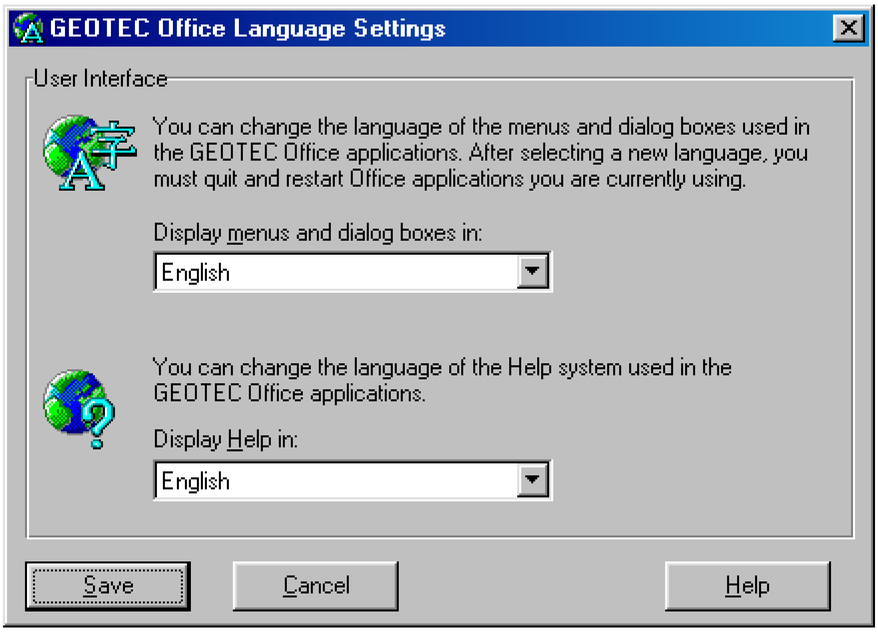


Figure B-1 “GEOTEC Office Language Settings” Tools

1.3 *ELPLA* Quick Tour

When you starting *ELPLA* for the first time, the “*ELPLA* Quick Tour” program (Figure B-2) appears. The tour program gives a quick access to the main contents of the *ELPLA* package as shown in Table B-1.

Table B-1 *ELPLA* Quick Tour Contents

Contents	**Description**
Tutorial	Taking the user step by step through some simple examples. These examples will help the user to become familiar with the most important functions of *ELPLA*.
Verification Examples	Verify the mathematical models used in program *ELPLA* by comparing *ELPLA* results with closed form or another published results. Illustrate how to use *ELPLA* for analyzing foundation by different subsoil models.
User's Guide	Display the complete User’s Guide in a PDF-Format.
Start *ELPLA*	Start *ELPLA* to create a new project

In the "Show this screen each time *ELPLA* starts" check box, you can choose whether you need to start the Quick Tour program each time when *ELPLA* starting or not. Also, you can start the “*ELPLA* Quick Tour” program any time by clicking on the program icon in the Windows ***Start-Menu> GEOTEC Office> Welcome to ELPLA***.

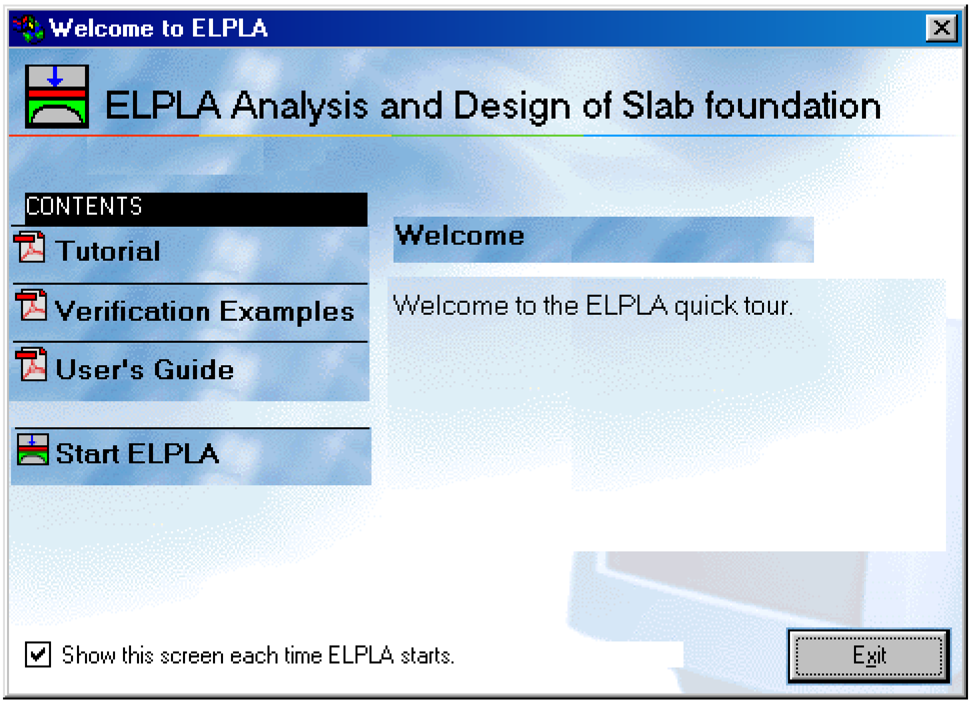


Figure B-2 "*ELPLA* Quick Tour" Program

1.4 Generating Circular Slab with Curved Element

A new template for generating second-order curved elements, Figure B-3, is now available for circular slab. As shown in Figure B-4, the new mesh refinement provides a better distribution of the results around the center of the circle, which reduces the local error in the elements around the center.

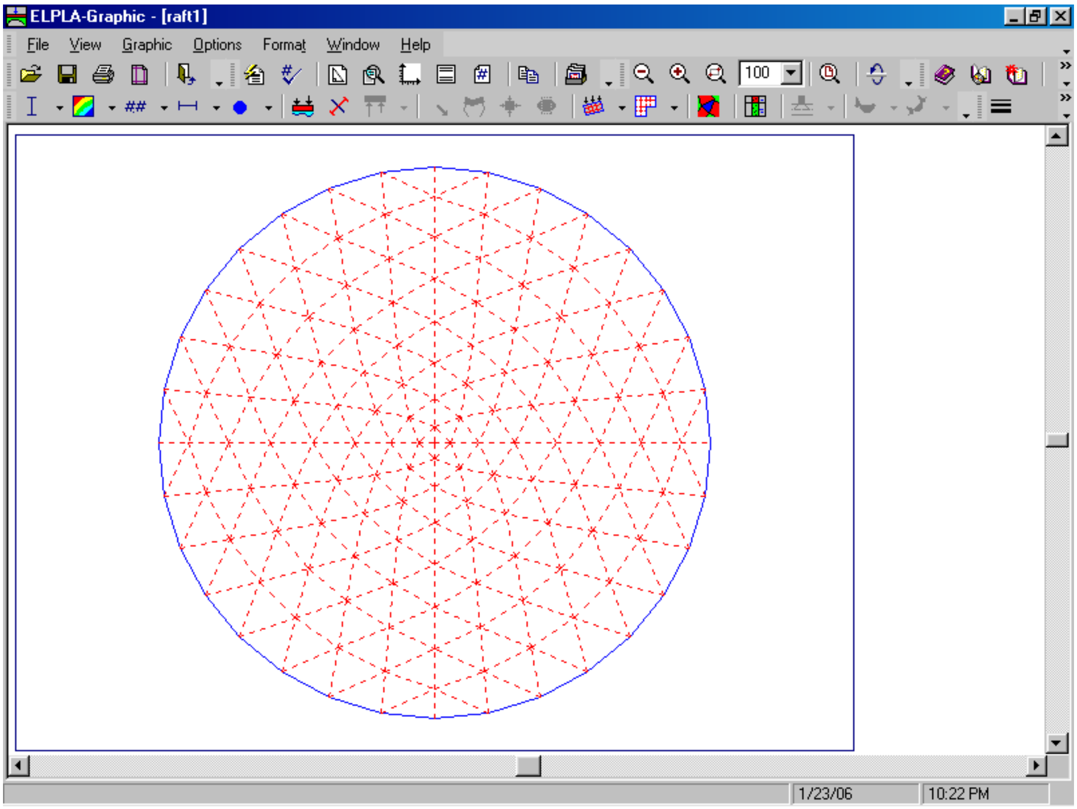


Figure B-3 Circular slab with curved elements

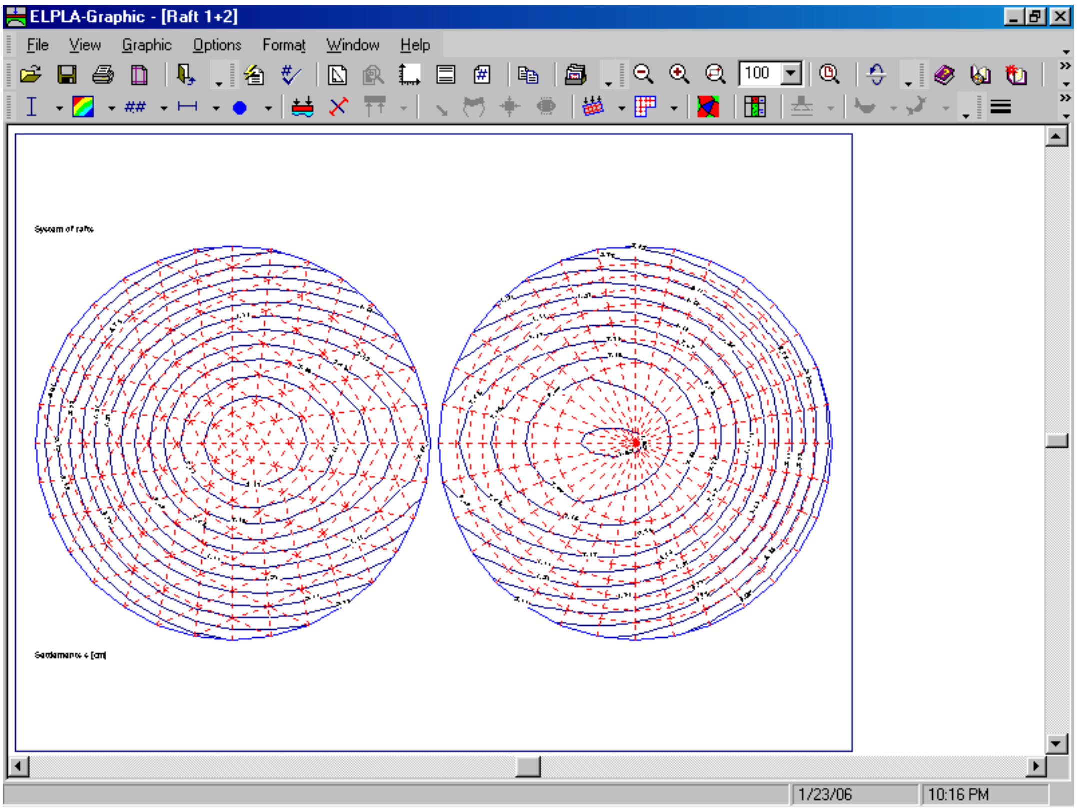


Figure B-4 Contour distribution for curved elements to the left and old elements to the right

1.5 Analysis of plane frame and plane stress

Besides the four different analyses available in the program *ELPLA* to analyze isolated raft, system of rafts, slab floors and grid, a new two analyses: Analysis of plane frame and plane stress are added (Figure B-5 and Figure B-6).

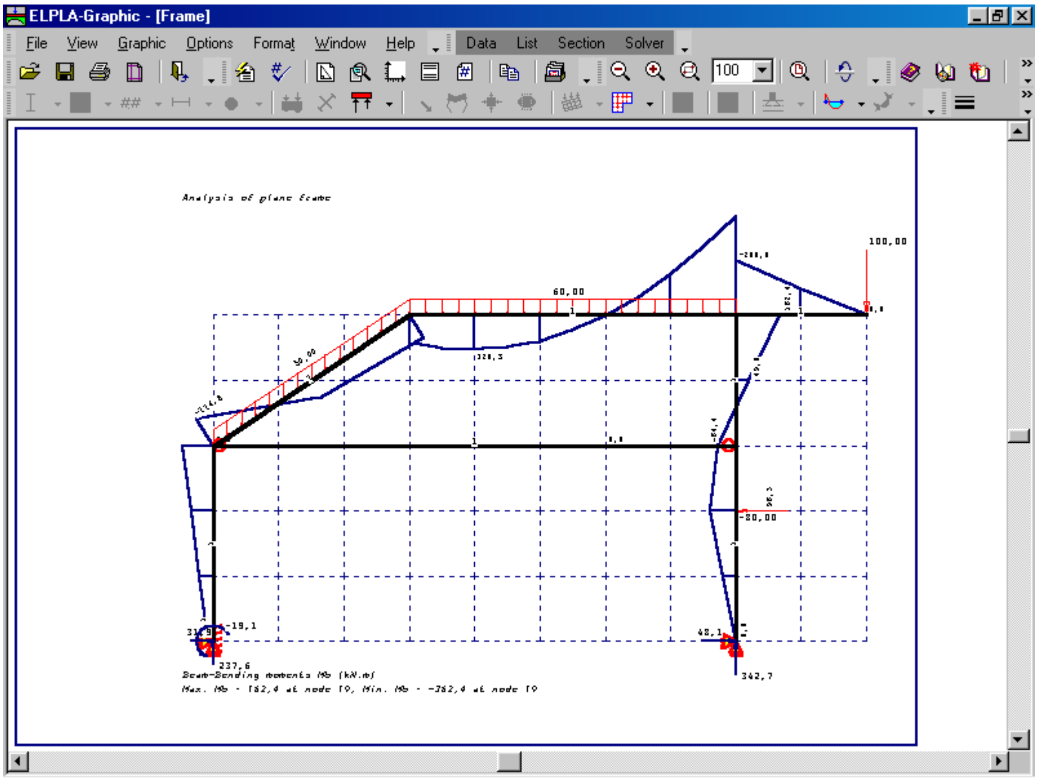


Figure B-5 Analysis of plane frame

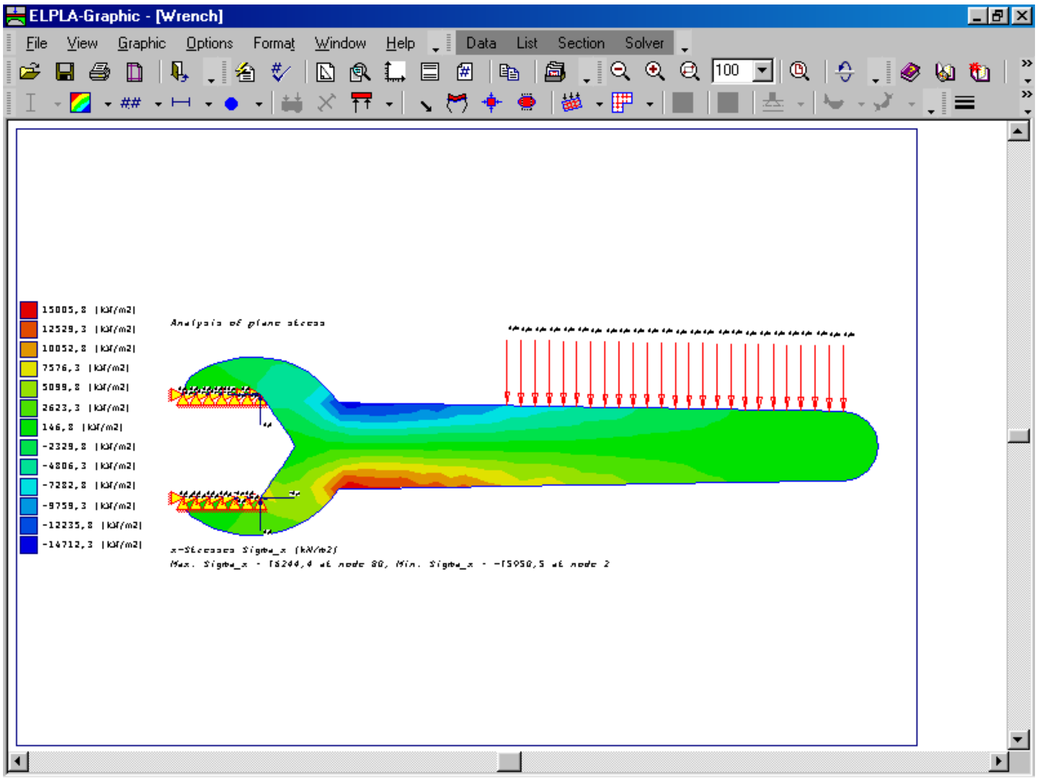


Figure B-6 Analysis of plane stress

2 Two-Dimensional Frame Problems

2.1 Introduction

The analysis of Two-Dimensional frame problems is now available in *ELPLA*. This section describes the frame modeling used for analysis this type of problems. It is recommended to read this section and to understand the procedures used by the program before starting to create any practical problem analysis.

2.2 Coordinate Systems

There are two different coordinates for Two-Dimensional frame problems; global coordinate system and local coordinate system (Figure B-7). 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 for the user to define correctly the problem.

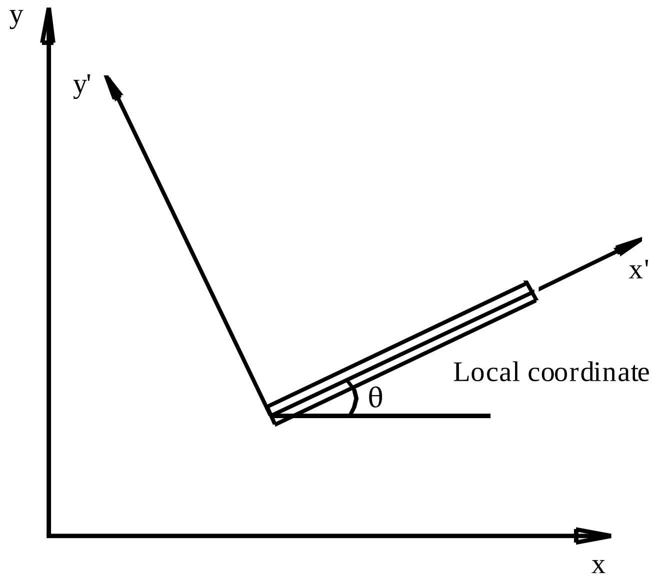


Figure B-7 System Coordinates

2.3 Element Loads

As shown in Figure B-8, *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 while element loads are applied in three different cases as follow:

a. Self weight: A vertical uniform load distributed along the length of the element.
b. Snow load: A vertical uniform load distributed along the horizontal projection of the element.
c. Wind load: A uniform load distributed along the length of the element with a direction perpendicular to the element (local x` axis).

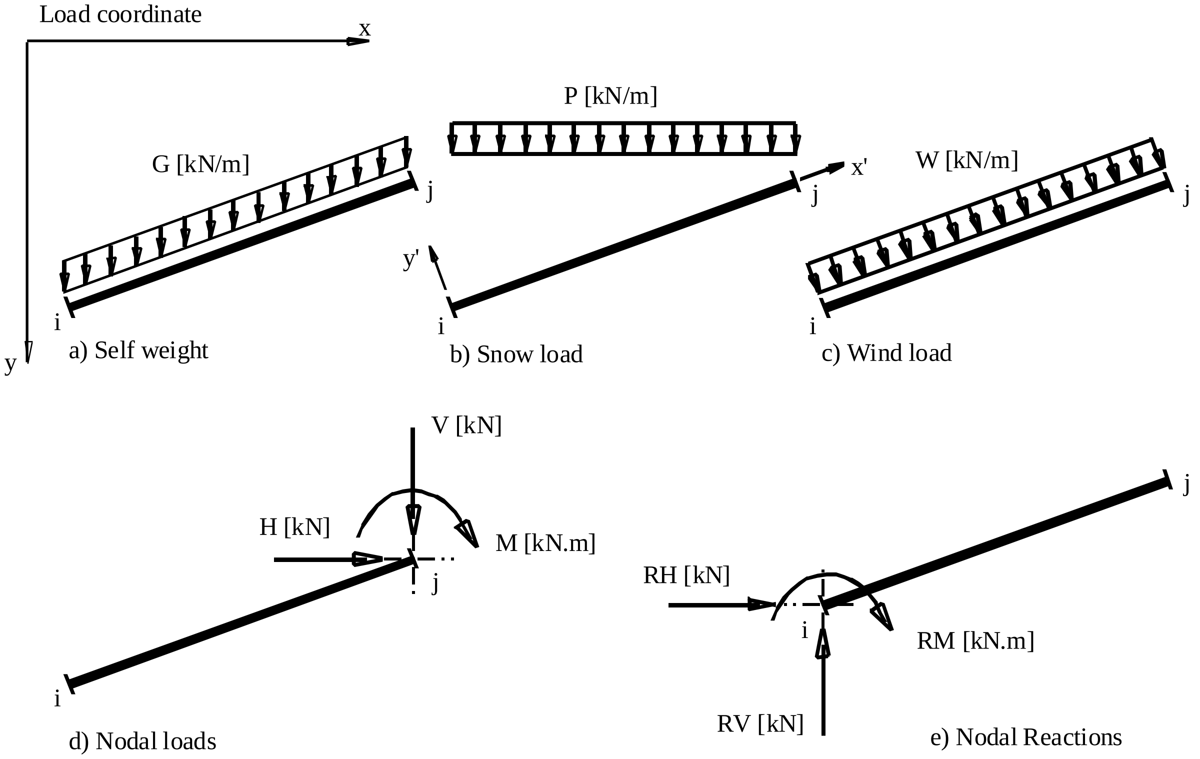


Figure B-8 Cases of element loads, nodal loads and nodal reactions with directions

2.4 Graphical output

The graphical output of results such as displacements, rotations and internal forces (bending moments, shear forces and normal forces) are drawn in locale coordinate.

GEOTEC Office Package

I Introduction

GEOTEC Office is a package for geotechnical and design engineering. The package contains the following programs and tools:

Program	Description
ELPLA	Analyzing footings, rafts, piled rafts, pile groups and foundation groups. ELPLA considers different soil models, especially the three dimensional continuum model that considers any number of irregular layers. The program is able to handle the three solutions of flexible, elastic and rigid foundations. ELPLA can also be used to represent the effect of external loads, neighboring foundations, tunneling and the influence of temperature change on the slab.
BOHR	Drawing borehole logging of soil layers by different symbols according to the German specification code DIN 4023.
TIEF	Analyzing single piles, pile walls, simple problems of rigid pile groups and rigid piled rafts.
GeoTools	Analyzing different problems in geotechnical engineering.
Self-Adaptive Mesh Wizard	Generating finite element mesh with better element/ node distributions.
GEOTEC-Editor	A simple word processing program for editing GEOTEC Office output.

II Installing GEOTEC Office

GEOTEC Office is distributed on a CD-ROM. The CD-ROM contains an installer program that installs the GEOTEC Office software on your computer.

To install GEOTEC Office, 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 GEOTEC Office on your computer.

Figure 1 GEOTEC Office Installer form

- Click “Next” button in the form of Figure 1 to install GEOTEC Office on your computer.

GEOTEC Office Installer program begins execution. Follow the instructions given by the GEOTEC Office Installer program, Figure 2.

- Click “Next” button in the form Figure 2 to install GEOTEC Office software to the specified destination folder.

You can specify the folder for GEOTEC Office files. By default, GEOTEC Office suggests the (..\Program Files\GEOTEC Office). However, you can indicate a different folder name. GEOTEC Office creates the folder name you specify.

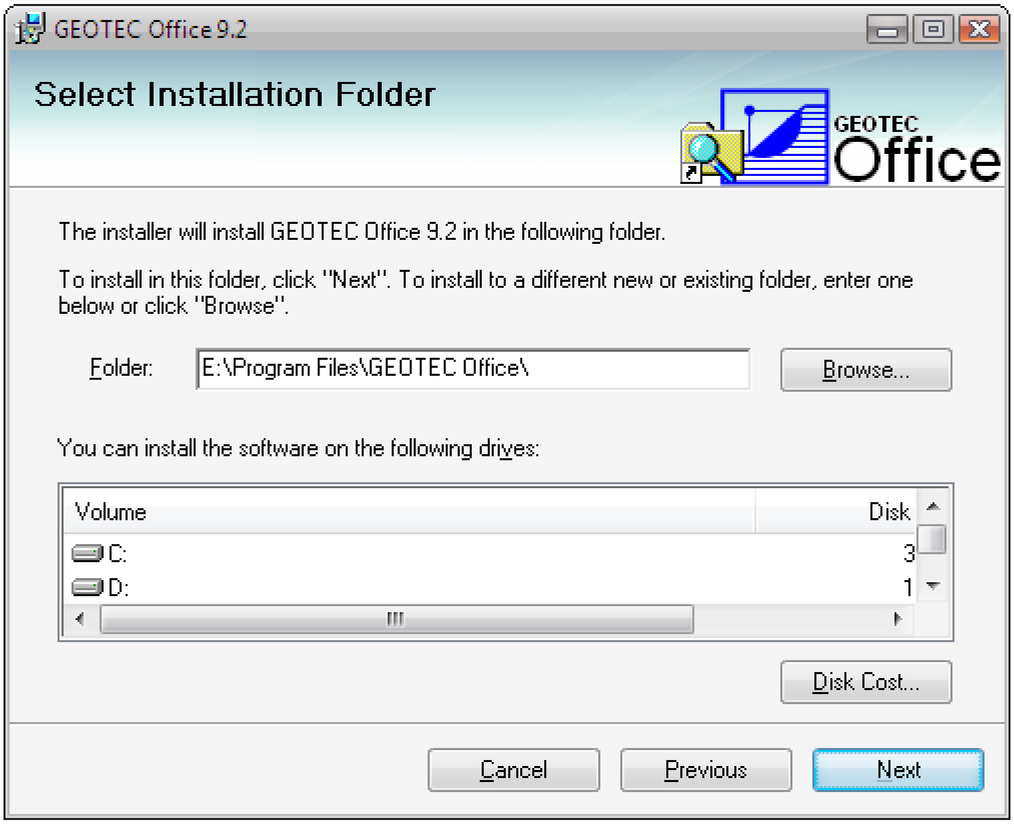


Figure 2 “Select Installation Folder” form

After select the installation folder, GEOTEC Office Installer program will be ready to install GEOTEC Office software on your program (Figure 3).

- Click “Next” button in the form of Figure 3 to start the installation.

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

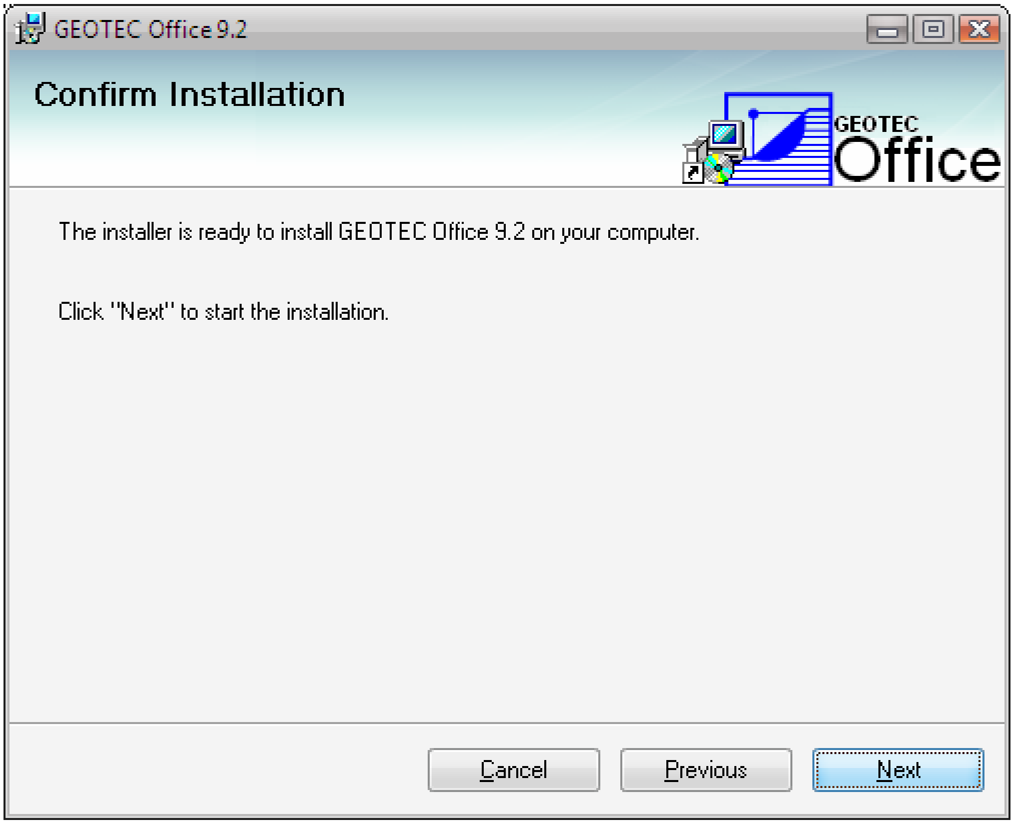


Figure 3 “Confirm installation” form

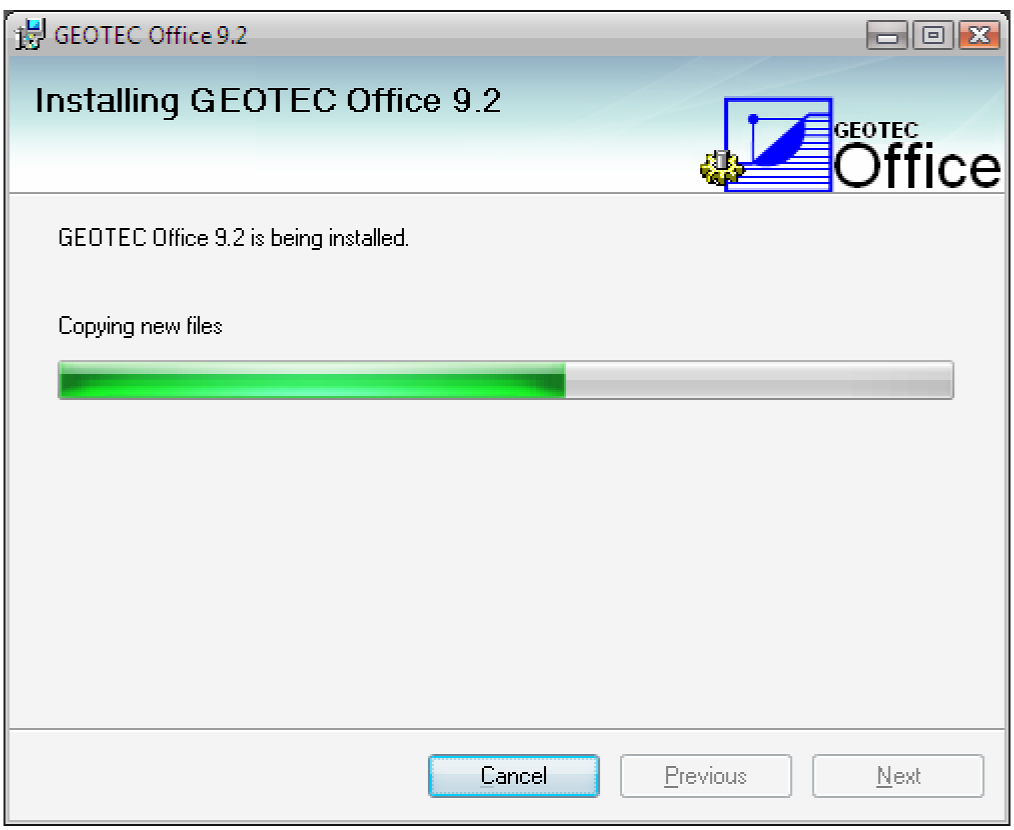


Figure 4 “Installing GEOTEC Office” form

GEOTEC Office installation will be completed and a message appears to inform you that the GEOTEC Office Installer was completed successfully (Figure 5).

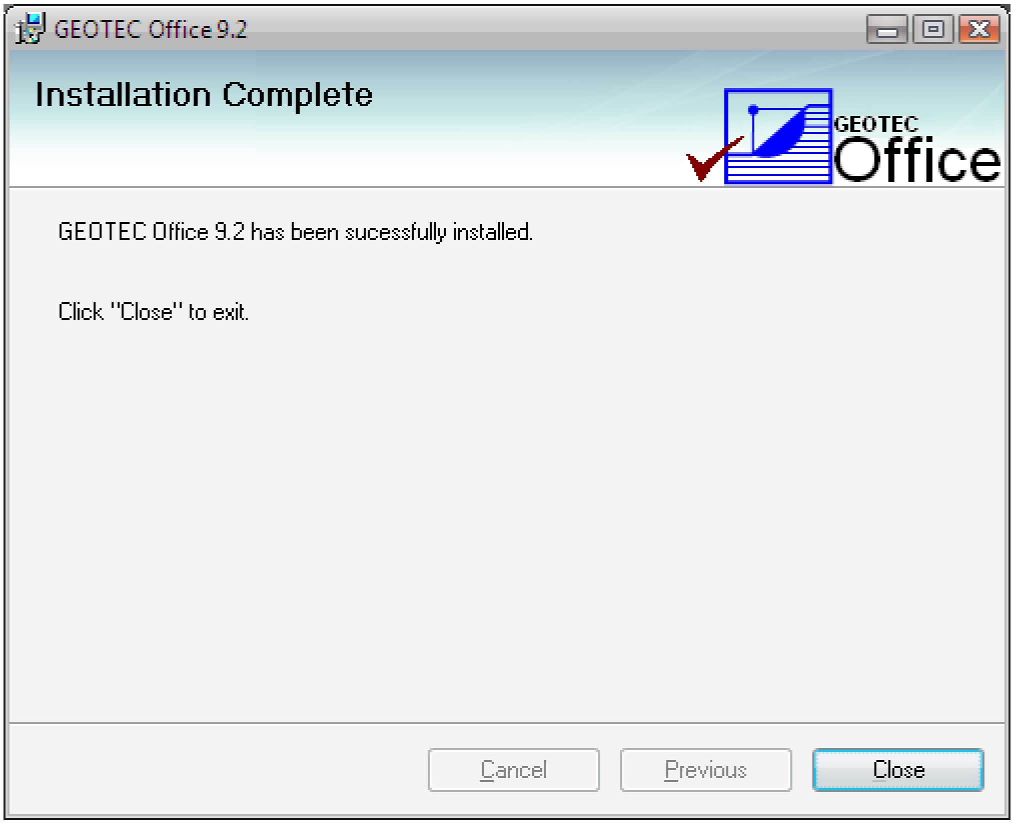


Figure 5 Final Installer message

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

Part B

New features and enhancements in 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

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

Contents Page

1 Preface

The program ELPLA is a part of the program group GEOTEC. The original program ELPLA 4.0 was developed first under the operating system MS DOS and then it was new revised for the operating system MS Windows 95 in order to use the possibilities of the user interface offered in it. The program runs also under Windows XP/ Vista/ 7/ 8.

ELPLA has a significant enhancement. The new features and enhancements are the result of feedback from users over the last years. The major enhancement in Version 9.0 is the ability to carry out a consolidation analysis beside elastic settlement analysis using the different calculation methods that are available in ELPLA. The program ELPLA also uses different types of finite elements, allowing the user to analyze any irregular shape of slabs with arc boundaries. Many improvements have been carried out in the generation of the flexibility matrix to allow the user solving large size problems faster.

The most important enhancements in ELPLA of versions 8.0 and 9.0 are explained in the next paragraphs.

2 Enhancements in version ELPLA 9.0

2.1 Design for punching

It is possible to design the slab floor or raft foundation for punching due to concentrated loads and reactions from columns, piles or supports. The design for punching is carried out according to 4 different codes: ACI, DIN 1054, EC2 and ECP.

2.2 Soil models

Now both layered soil model and isotropic elastic half-space soil model are available for all calculation methods. The isotropic elastic half-space soil model is added for the following methods:

Method 4: Modification of Variable Modulus of Subgrade Reaction
Method 6: Modulus of Compressibility (Iteration)
Method 8: Modulus of Compressibility for rigid raft, Figure B-1
Method 9: Modulus of Compressibility for flexible foundation

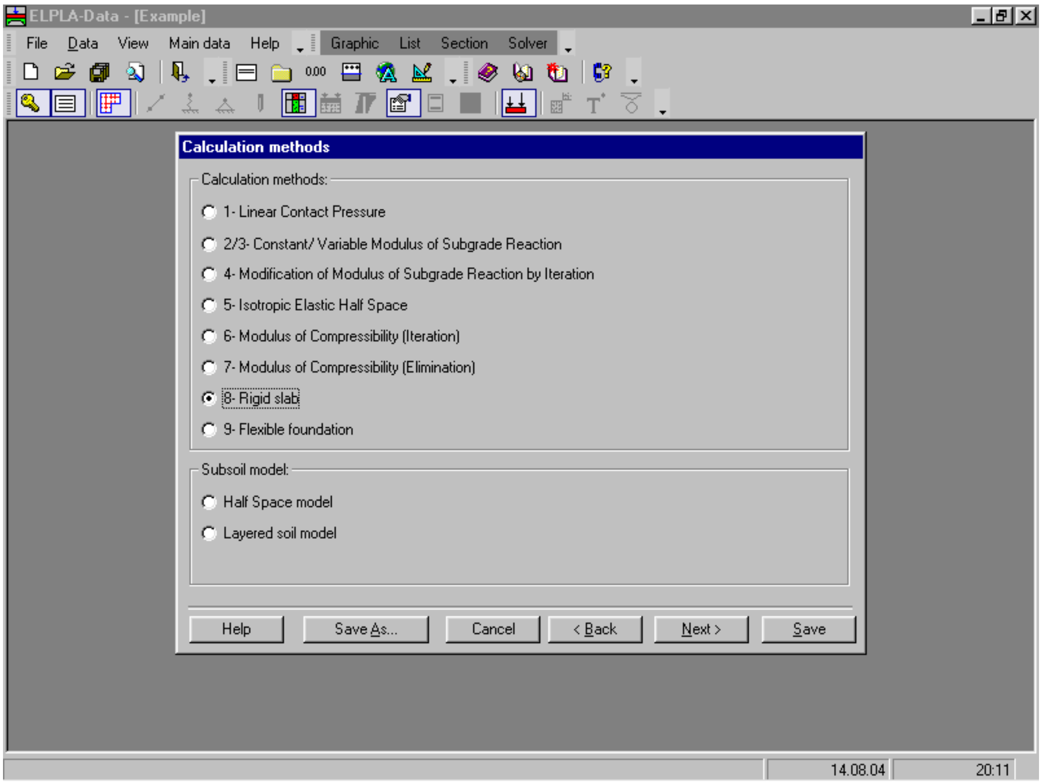


Figure B-1 Menu "Calculation method"

2.3 Bearing capacity factors

The bearing capacity factors used to determine the ultimate bearing capacity can optionally be defined according to different codes and authors. These factors are required to carry out the nonlinear analysis of the soil. The bearing capacity factors are defined according to Figure B-2:

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

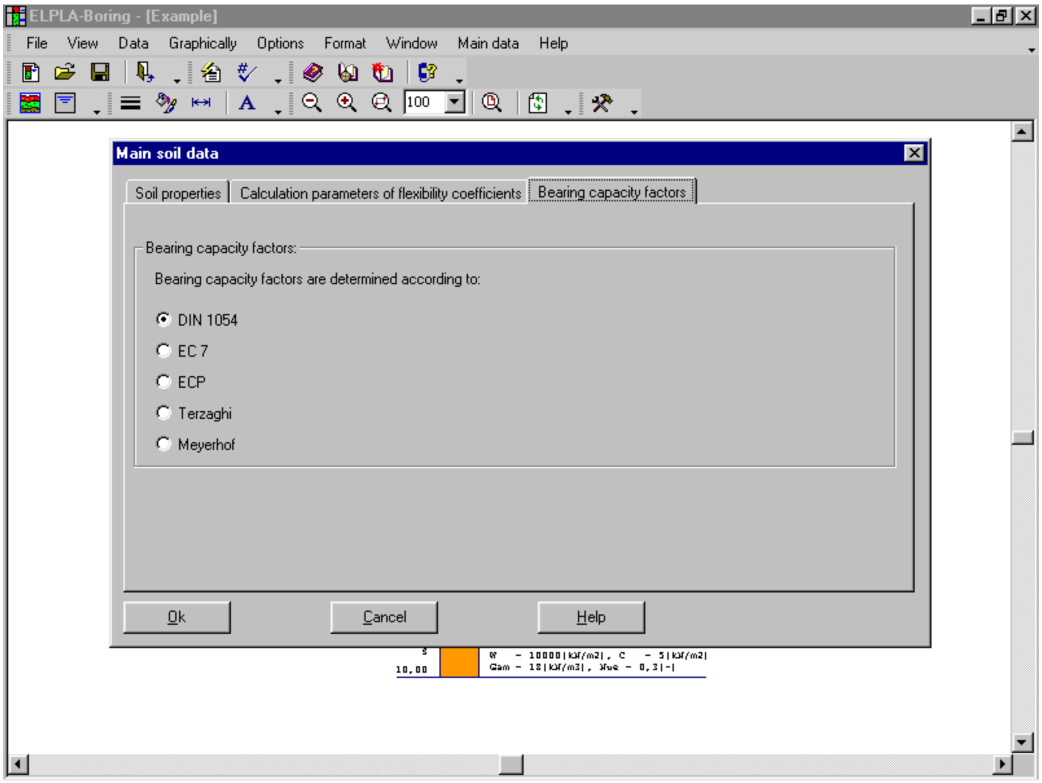


Figure B-2 Menu "Bearing capacity factors"

2.4 Flexibility coefficients for interior nodes

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

Now it is possible to determine the flexibility coefficient of the interior node due to a uniform load at that node, Figure B-3:

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

2.5 Flexibility coefficients for exterior nodes

Earlier versions of ELPLA determine flexibility coefficients for both interior and exterior nodes by assuming uniform loaded areas on these nodes. This assumption needs to use the principle of superposition for determining the flexibility coefficients. Now it is possible, optionally to convert the loaded areas on exterior nodes to point loads (Figure B-3). By this way the program doesn’t need to use the principle of superposition in the analysis, making it much faster than the old analysis. The new way of analysis is consequently faster and more efficient for problems that contain a large finite element mesh.

2.6 Limit distance

If the distance between two nodes is too large, the settlement of a node due to a load on the other will be small enough to be neglected. To reduce the time required for determining the flexibility coefficients for great rafts, a limit distance between node i and j for determining the flexibility coefficient c(i, j) may be defined (Figure B-3).

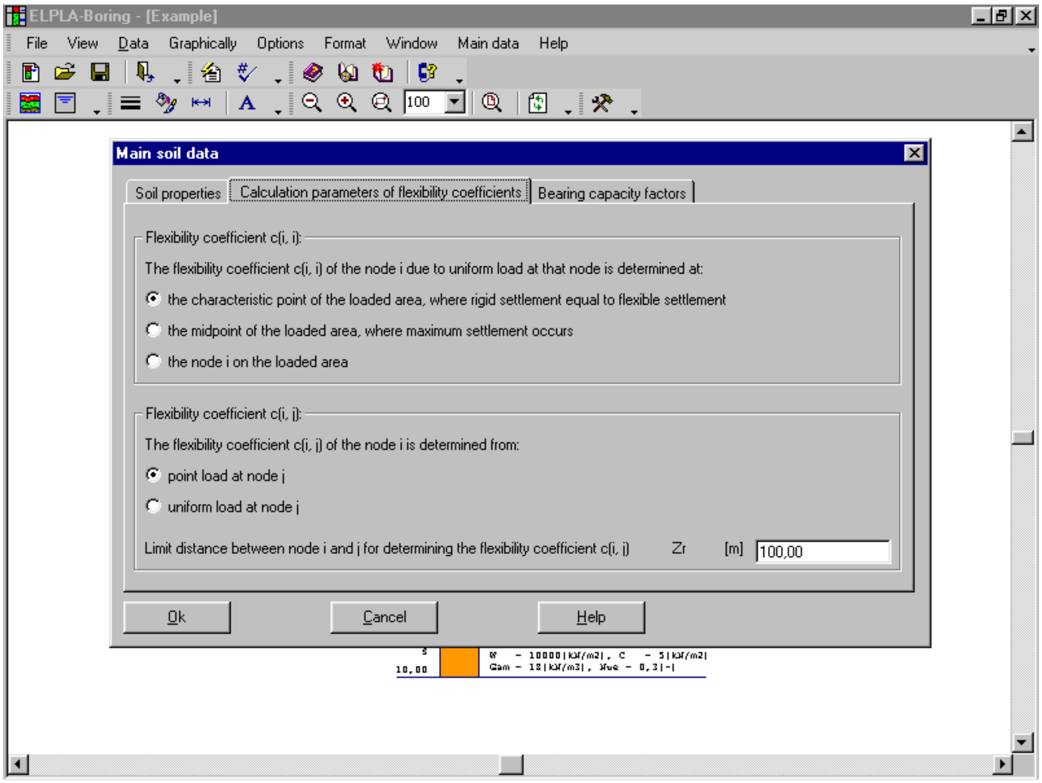


Figure B-3 Menu "Flexibility coefficients"

2.7 Soil properties

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

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

This option enables ELPLA to analyze rafts on consolidated clay deposits by the different calculation methods that are available in ELPLA (Figure B-4). Also the user doesn't need to convert a soil parameter to another. When defining soil properties by the Modulus of Elasticity E, the *Poisson*'s ratio ν_s can be different for every layer.

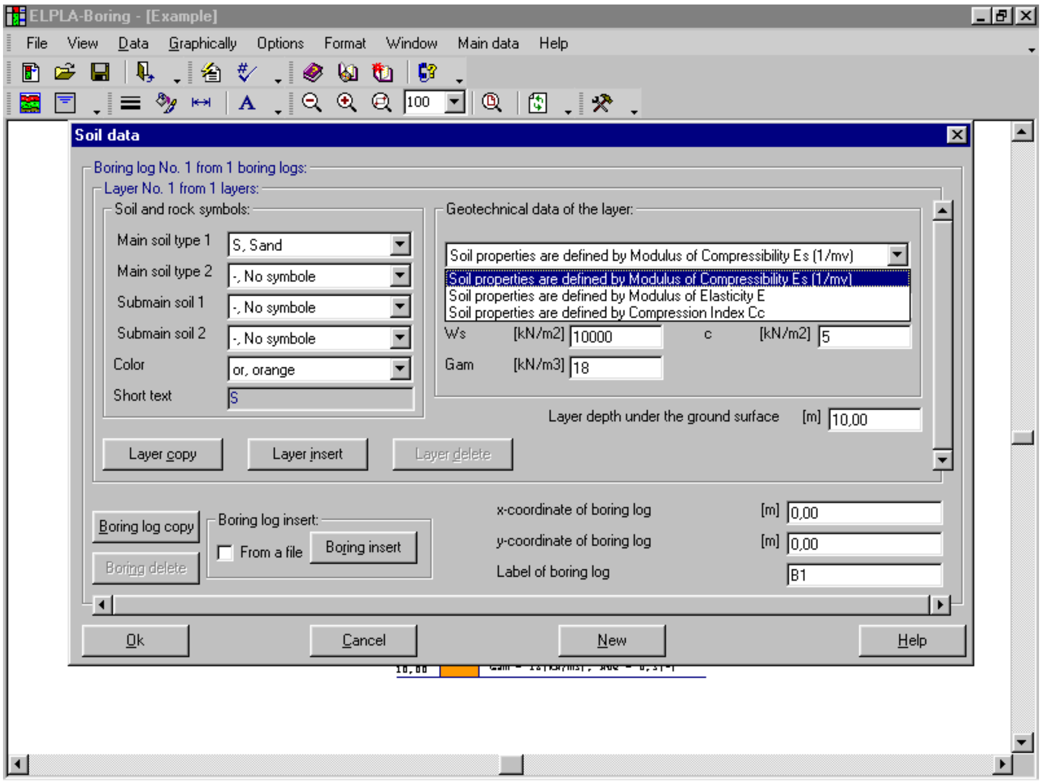


Figure B-4 Menu "Soil data"

2.8 FE-Net

Different element types are developed to generate the FE-Net of the slab according to Grid-based approach for both triangular and rectangular elements and according to *Delaunay*'s triangular generation for triangular elements.

A user-friendly embedded program for mesh generation is developed. The essential features in this embedded program are:

- generating the FE-Net for square, rectangular and irregular slabs using 6 different types of nets, Figure B-5

- generating the FE-Net for circular and ring slabs using 8 different types of nets, Figure B-6

- It can represent an irregular slab with openings and arc boundaries using a refine mesh, Figure B-7

- It is possible to use combined rectangular, quadratic and triangular finite elements at the same time for the slab, Figure B-8

- It is possible to define reference points and lines on the slab, Figure B-9. Reference points and lines are used to define the positions of girders, supports, piles, etc on the slab. Each time that the user generates the mesh, nodes of the FE-Net are passed automatically through these references. This provides the flexibility to make changes in the finite element mesh without having to redefine the positions of girders, supports, piles, etc

- You can refine the mesh in a specified region such as around supports to present the concentration of stress, moment and settlement in this region, Figure B-10

- It is possible to optimize the dimension of FE-Net by making all elements having nearly the same area using the option "smoothing the mesh"

- FE-Net can be displayed in separated elements, Figure B-11

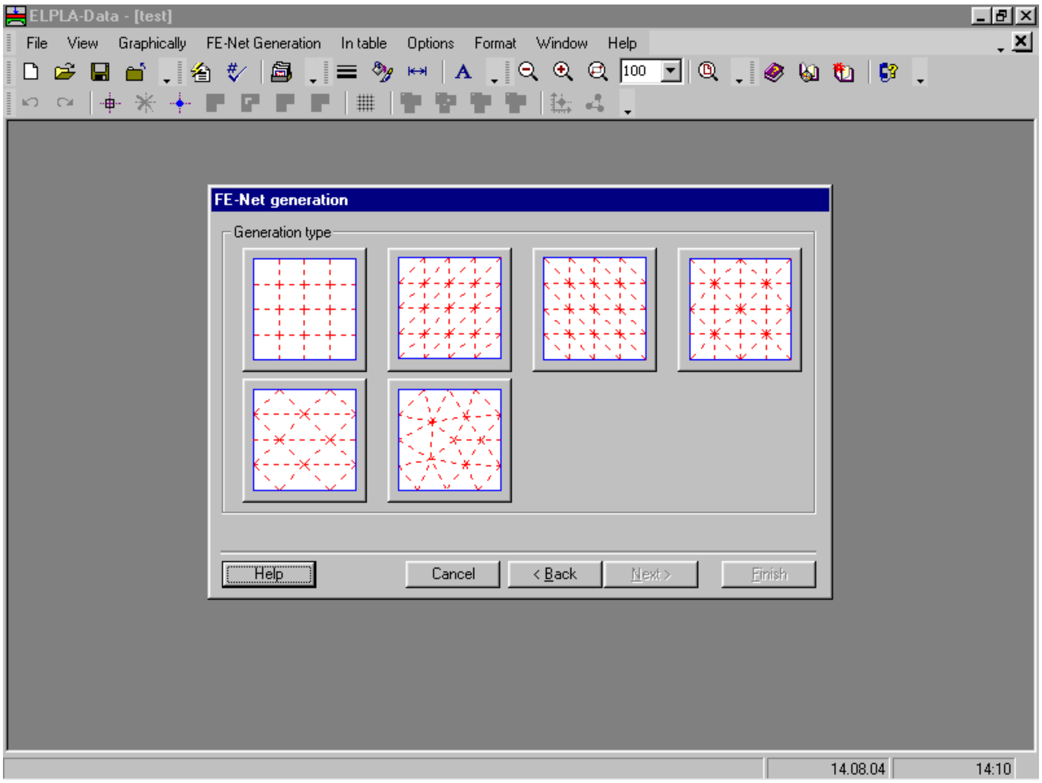


Figure B-5 Generation type for square, rectangular and irregular slabs

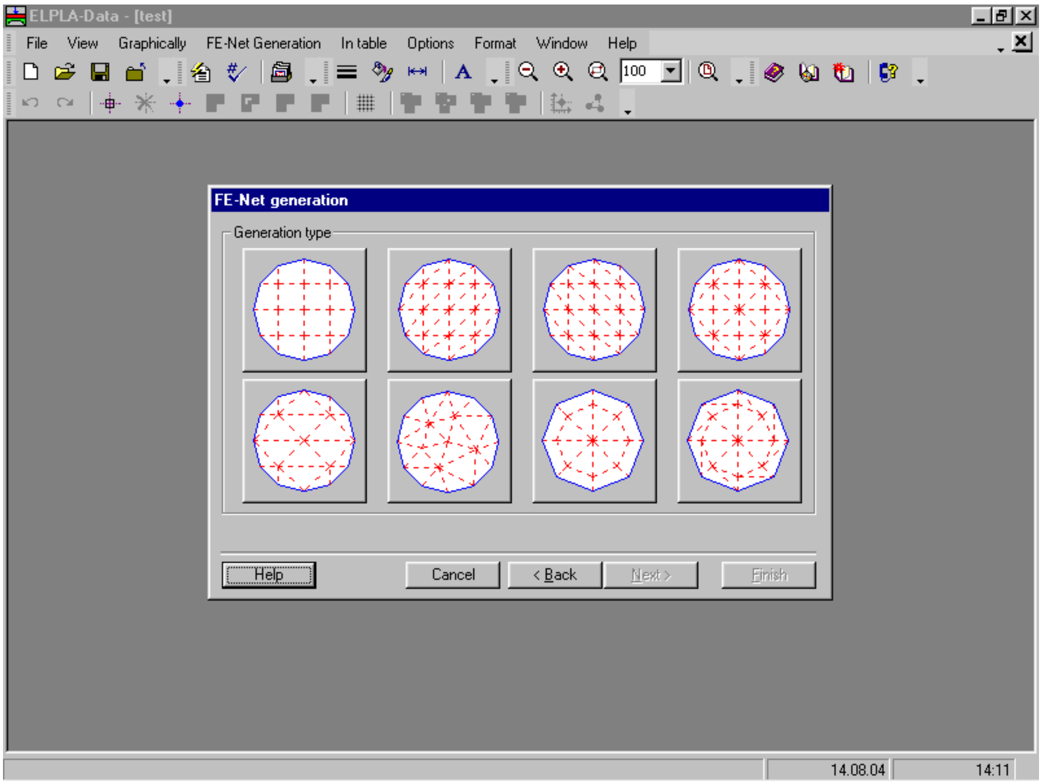


Figure B-6 Generation type for circular and ring slabs

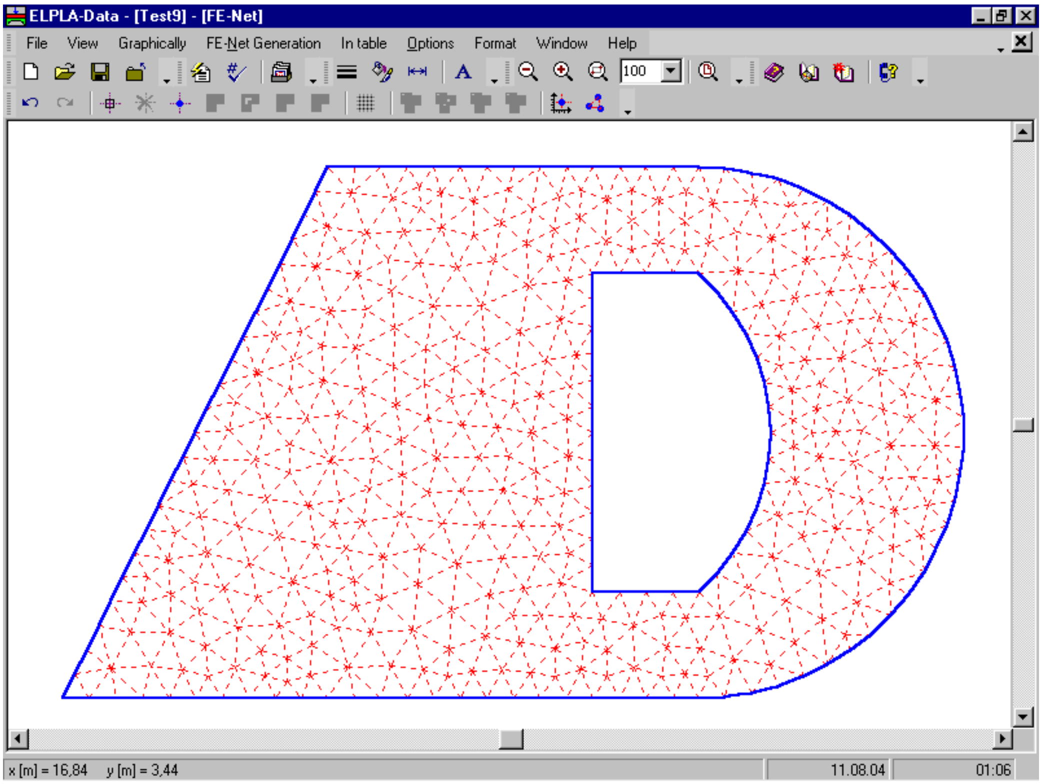


Figure B-7 Irregular slab with triangular finite elements

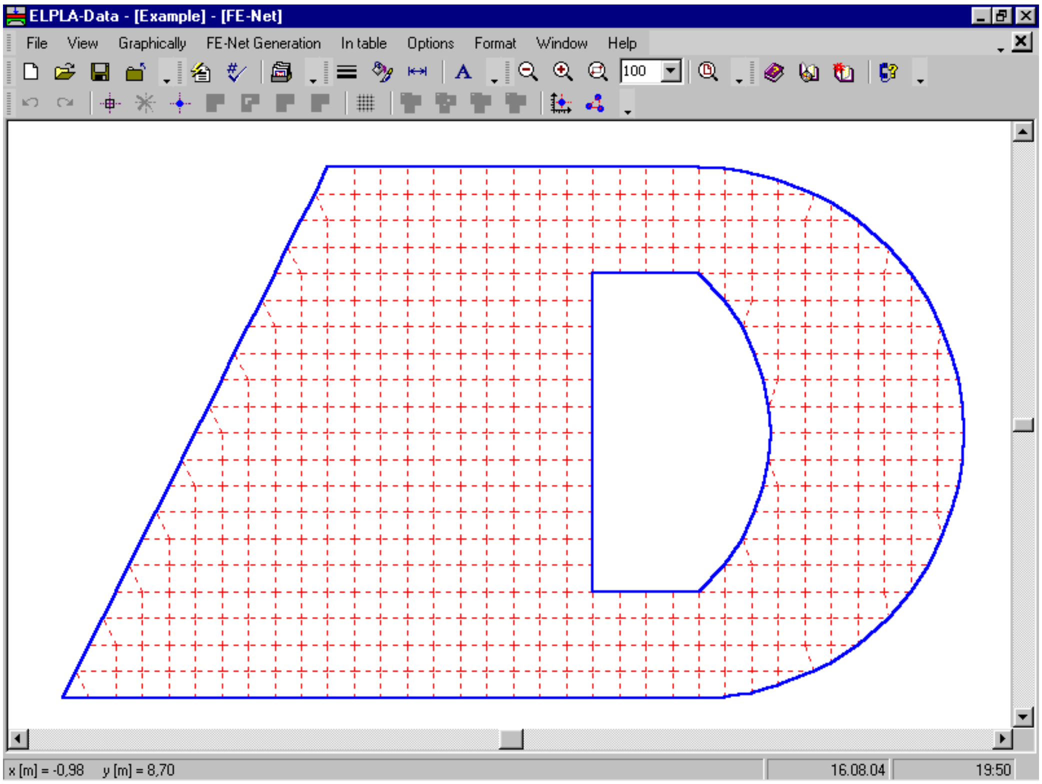


Figure B-8 Irregular slab with combined rectangular, quadratic and triangular finite elements

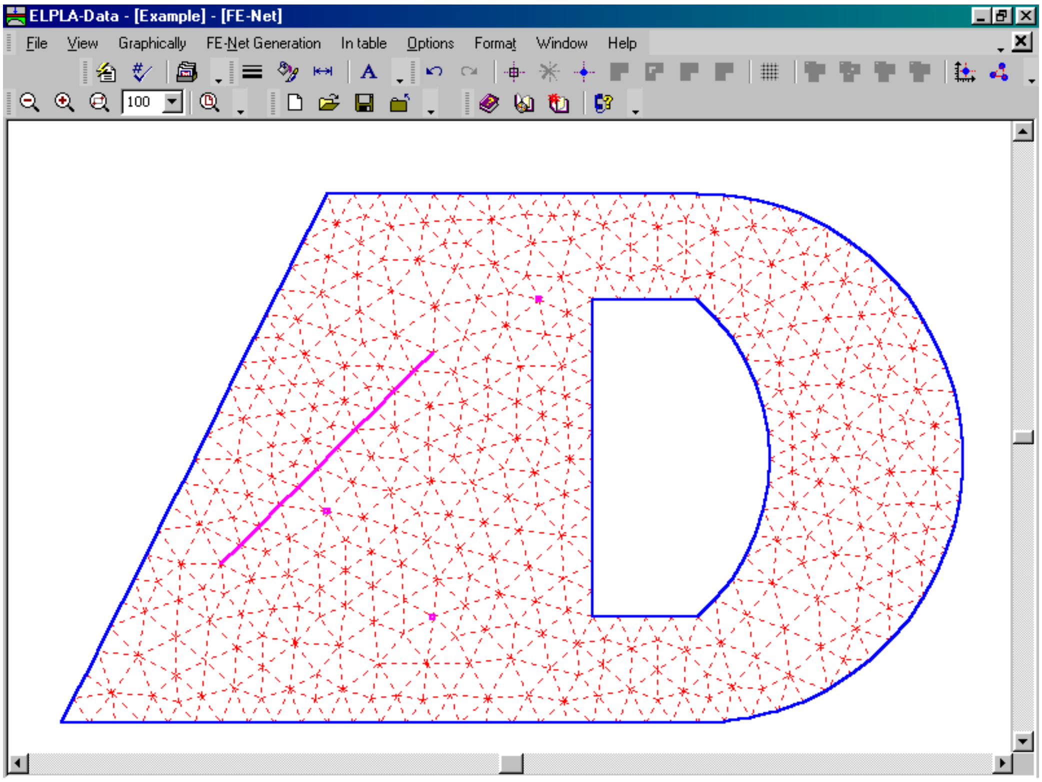


Figure B-9 Reference points and lines on the FE-Net

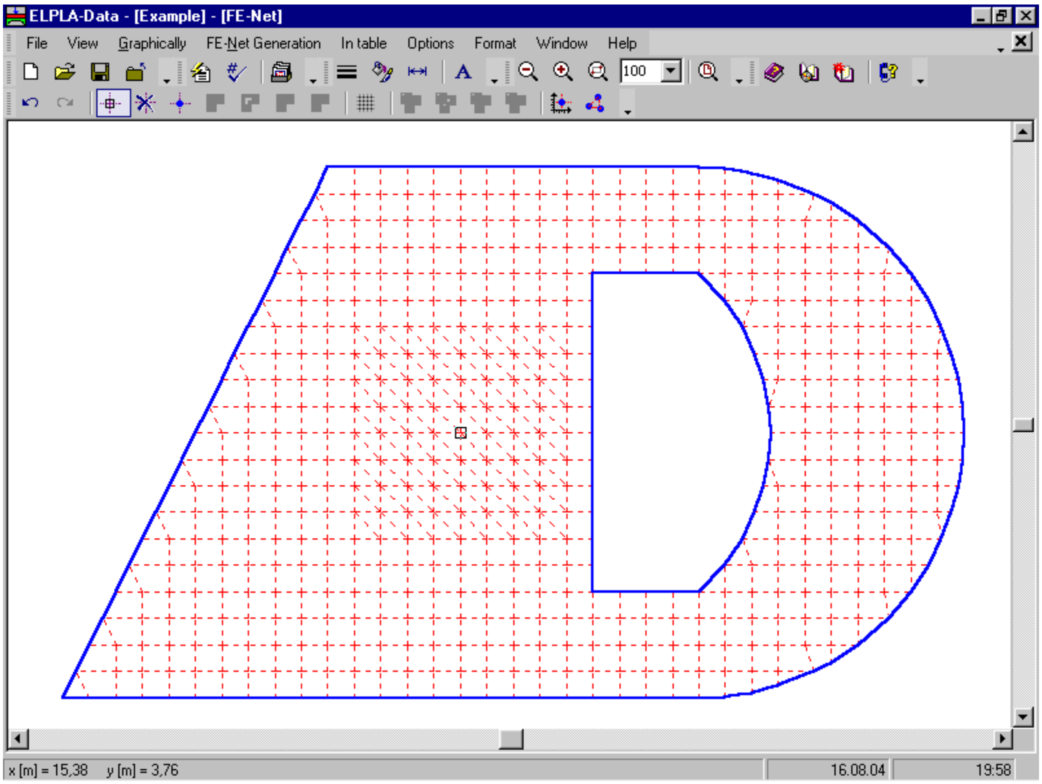


Figure B-10 Refining the mesh around a specified node

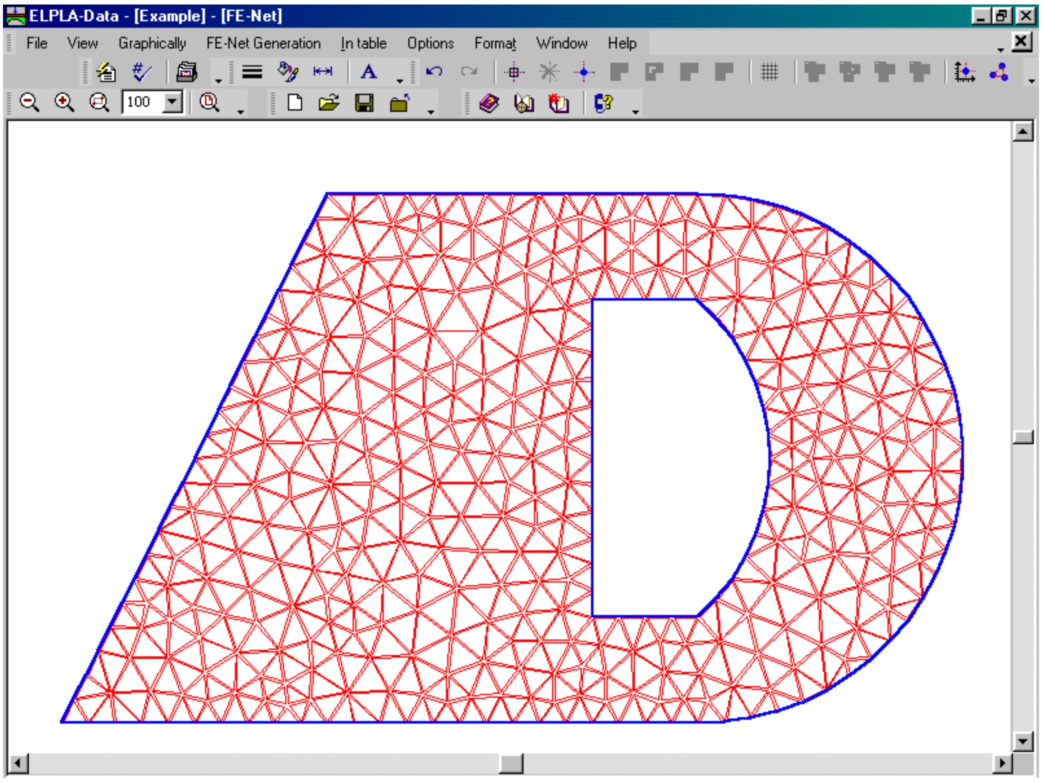


Figure B-11 FE-Net in separated elements

2.9 Unit systems

It is possible to set different unit systems such as SI-system or English-system without changing the real value of any previously defined data, Figure B-12.

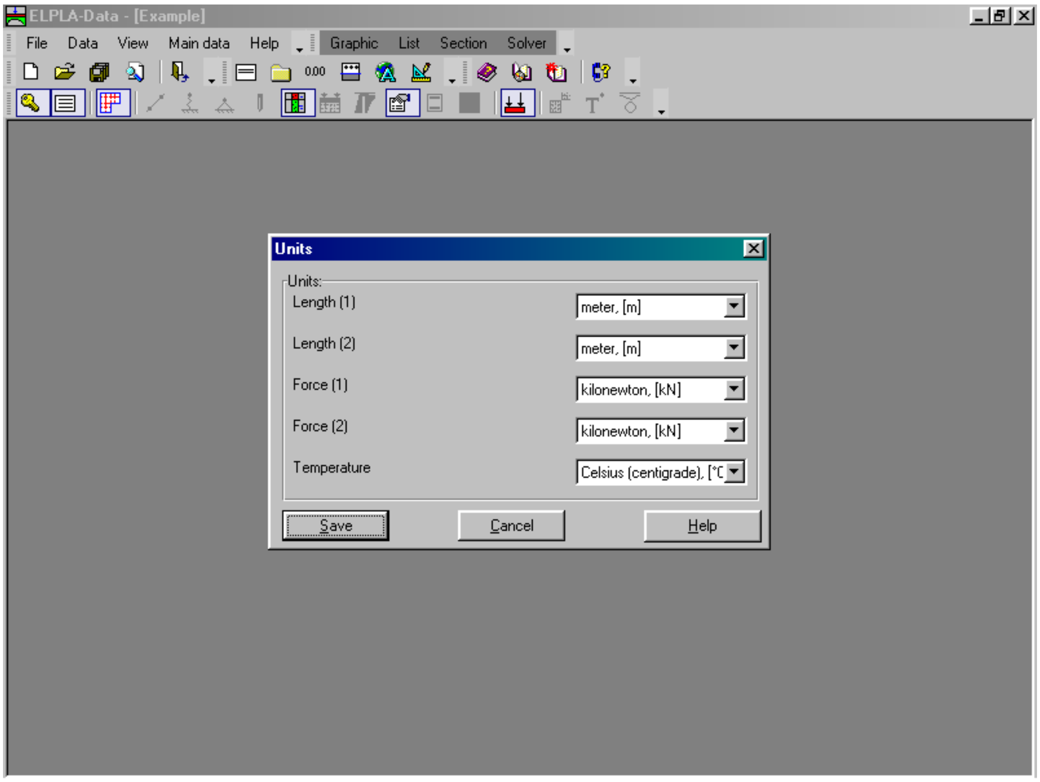


Figure B-12 Menu "Setting unit system"

2.10 Creating tasks through Wizard assistance

Simple step-by-step Wizard assistance in different tasks is possible, such as creating input data, redefining existing data or generating the net of finite elements. The Wizard assistance simplifies the process of creating and redefining the data using the standard and familiar "wizard" interface. A Wizard is a series of forms in a special window helping you through a task. The Wizard is used throughout Windows and by many Windows applications. The Wizard interface is ideal if you want to know as little about the task as possible. With the Wizard you simply click the "Next" button few times to carry out the task. Figure B-13 shows an example for Wizard assistance when defining the calculation method.

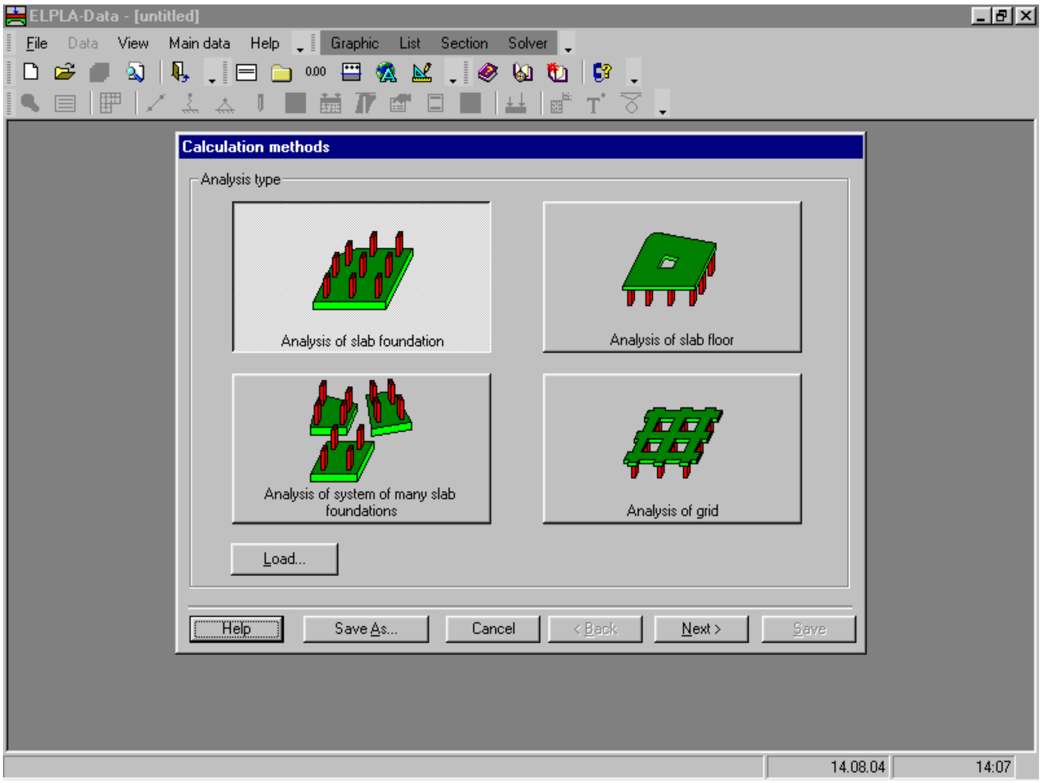


Figure B-13 Menu "Calculation methods" through Wizard assistance

2.11 Undo and Redo of commands

It is possible to undo and redo all commands in ELPLA-Data when defining a project. You may wish to undo the effects of a selected command and return to a previous state, Figure B-14.

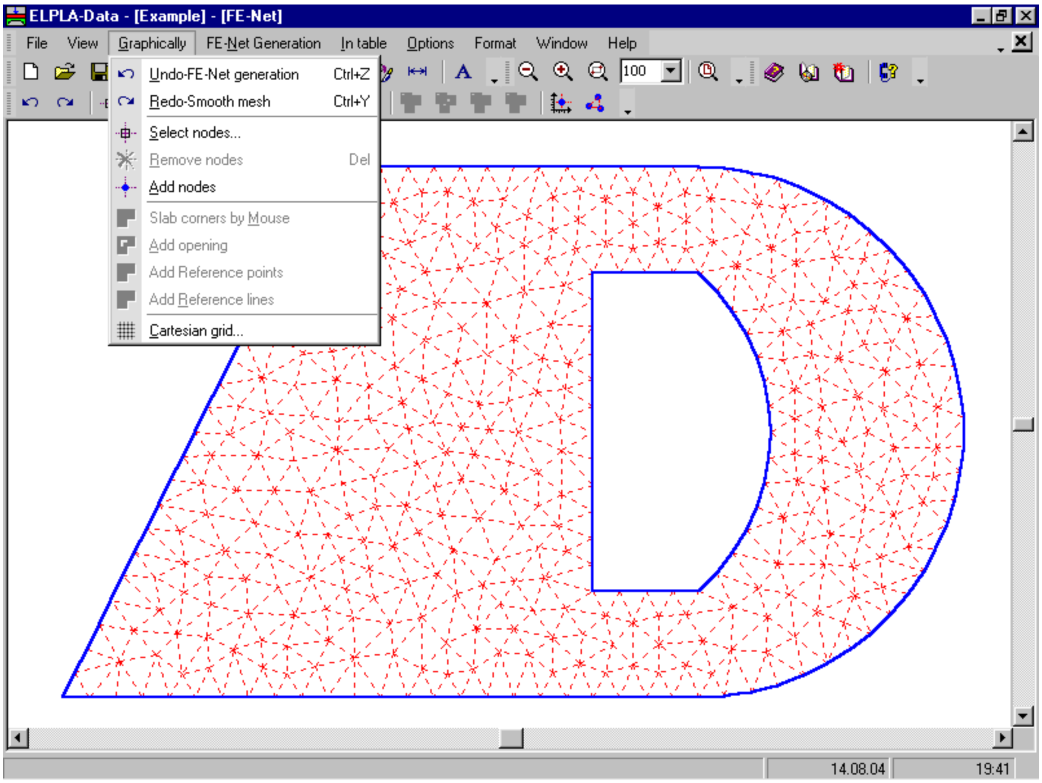


Figure B-14 Undo and redo the commands when defining FE-Net

2.12 Compressing data files into one file

The files of input data, intermediate results or final results for a project can now optionally be stored in one compressed file. Instead of storing hundreds of data files, you can now have one automatically compressed file for each project. This makes it easier to send projects to other persons or simply manage your own files. It also reduces the amount of disk space required to store all of your data files. The compressed file is ZIP-compatible, allowing you to manually extract the data files using WinZip or other data compression tools if you wish, Figure B-15.

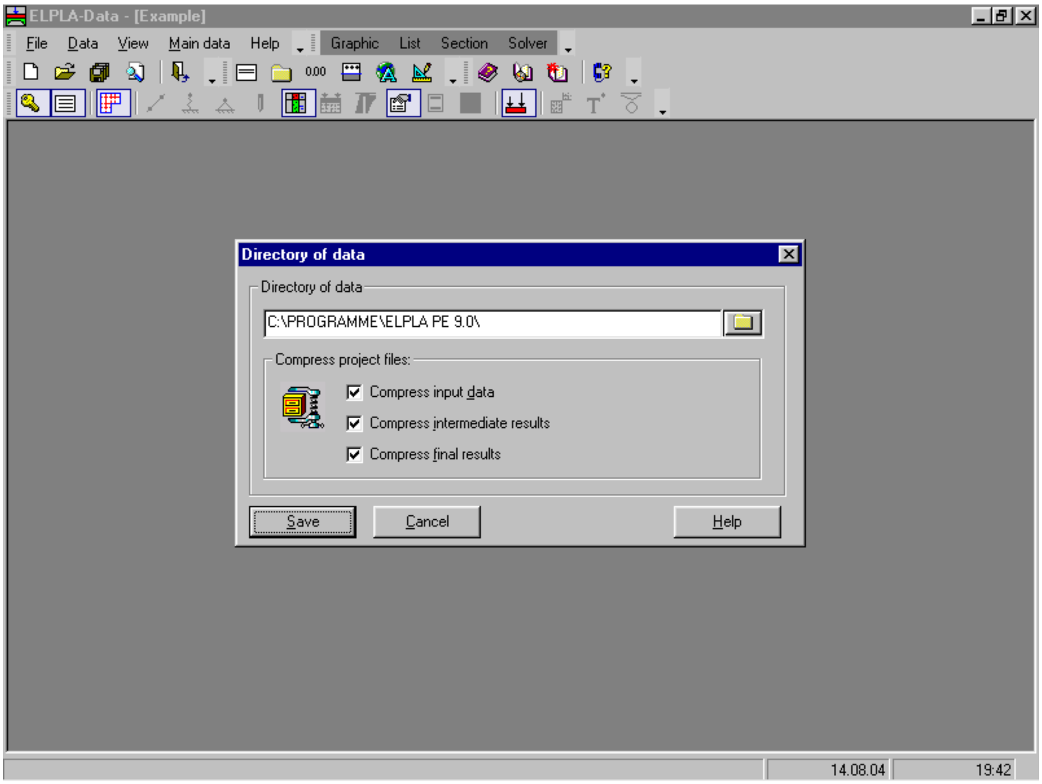


Figure B-15 Menu "Compressed project files"

2.13 Floating toolbars and icon menus

Many menu items and toolbars have been modified to make the commands easier to use:

- New toolbars have been added to ELPLA for quicker access to commonly used commands. These toolbars can be placed anywhere in ELPLA-window or can be hidden from view, Figure B-16

- Most of Menus and Popup-Menus in ELPLA have icons to easy identify the command, Figure B-17

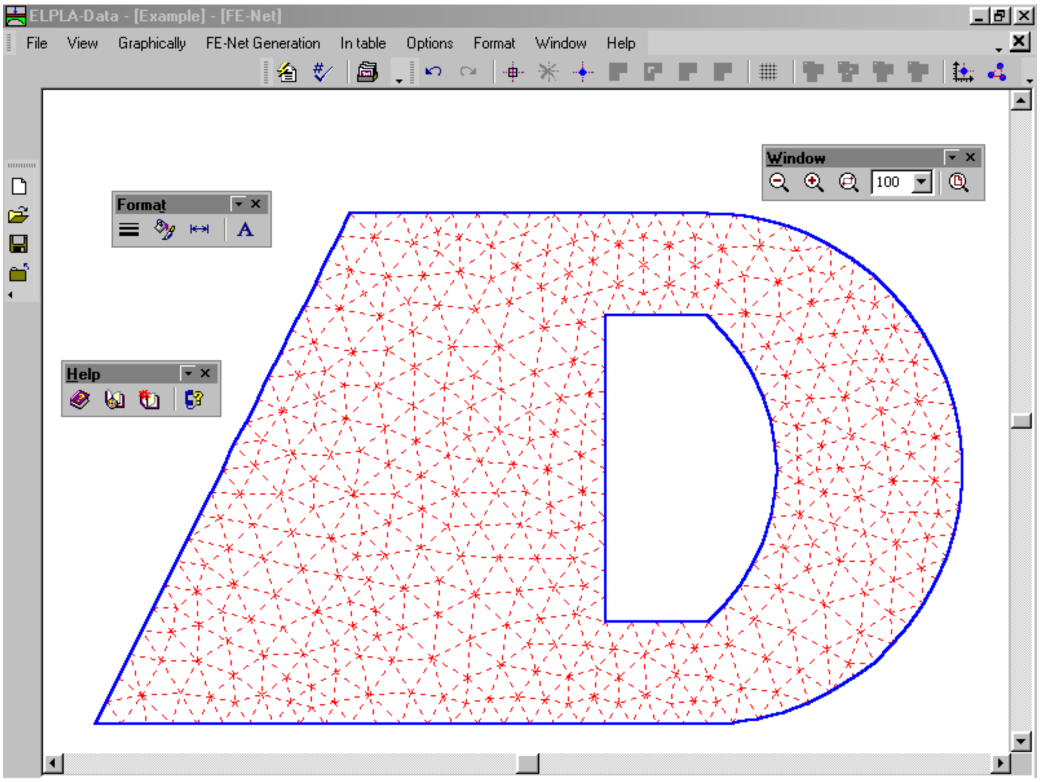


Figure B-16   Toolbars can be placed anywhere in the ELPLA-window

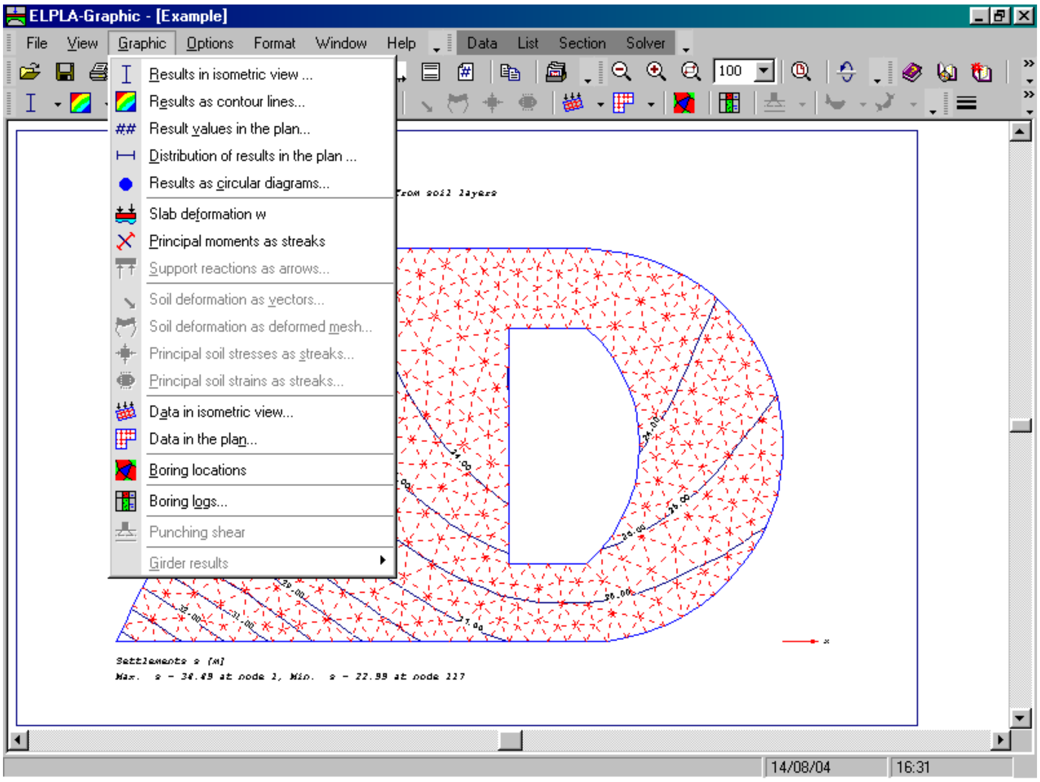


Figure B-17   Menu "Graphic" with icons

2.14 Graphic output

- Line formats setting for ELPLA-Data and ELPLA-Graphic are the same, Figure B-18
- Max. ordinates setting for ELPLA-Data and ELPLA-Graphic are the same, Figure B-19
- Fill colors setting for ELPLA-Data and ELPLA-Graphic are the same, Figure B-20
- The boundaries between contour regions are improved to be very smooth, Figure B-21
- It is possible to draw girders in isometric view with actual size, Figure B-22
- Each girder type has a specific color to easy identify the girder system, Figure B-22

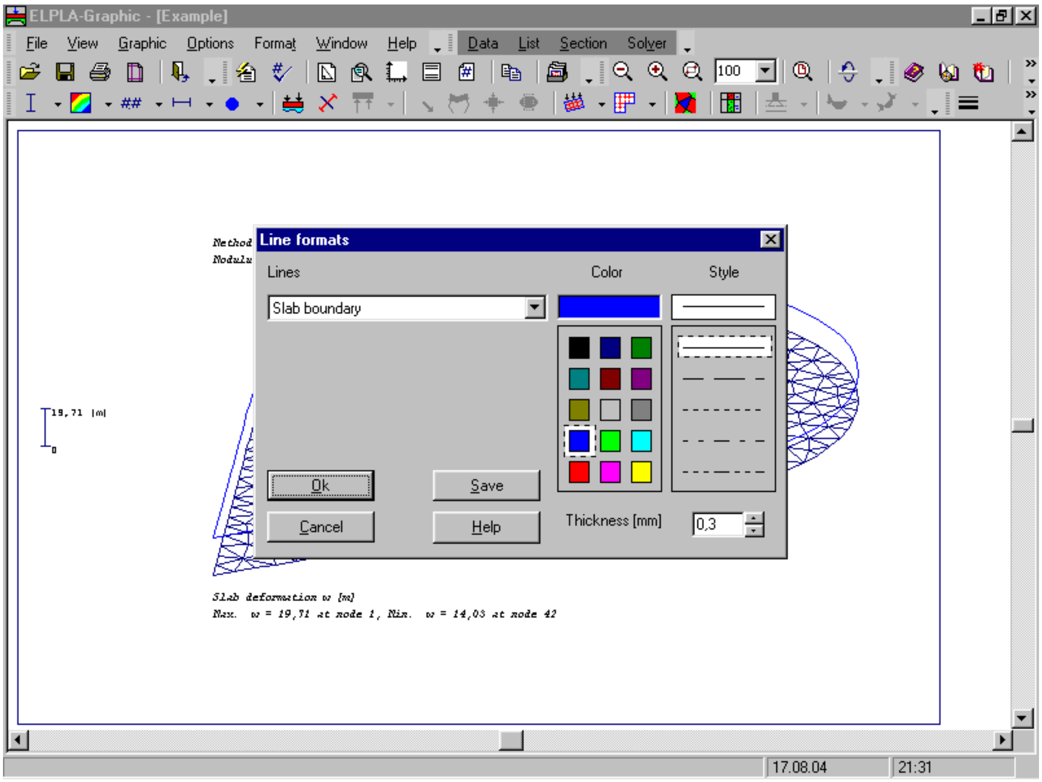


Figure B-18 Line formats for Data and Graphic are the same setting

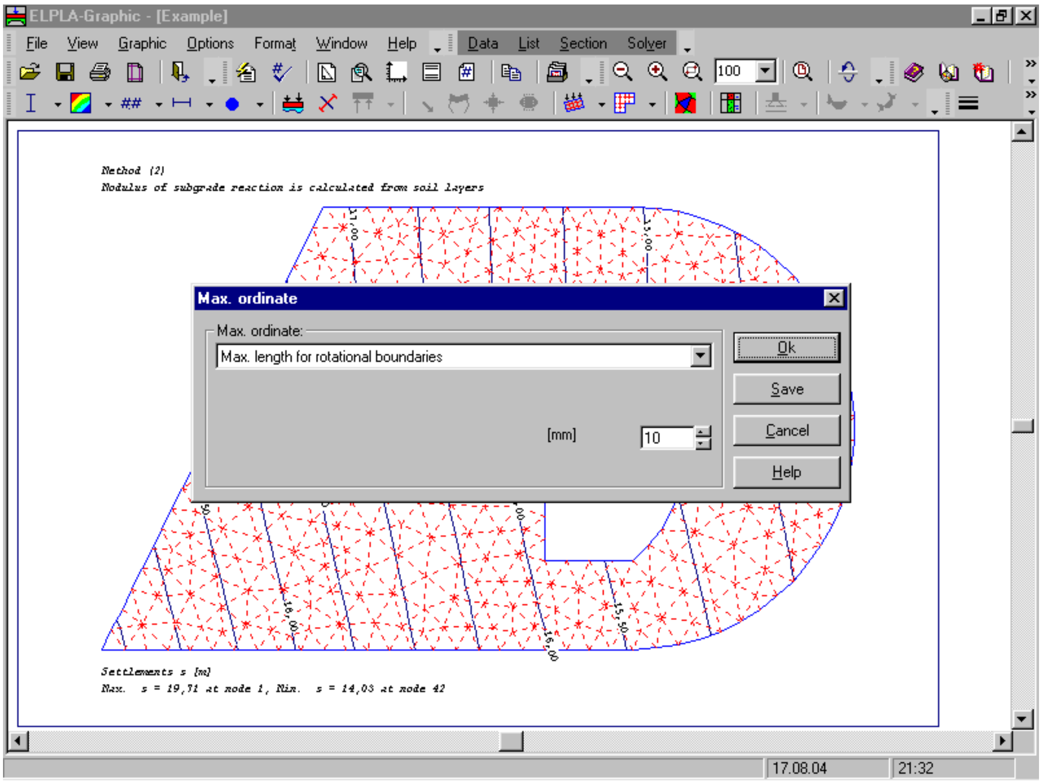


Figure B-19 Max. Ordinates for Data and Graphic are the same setting

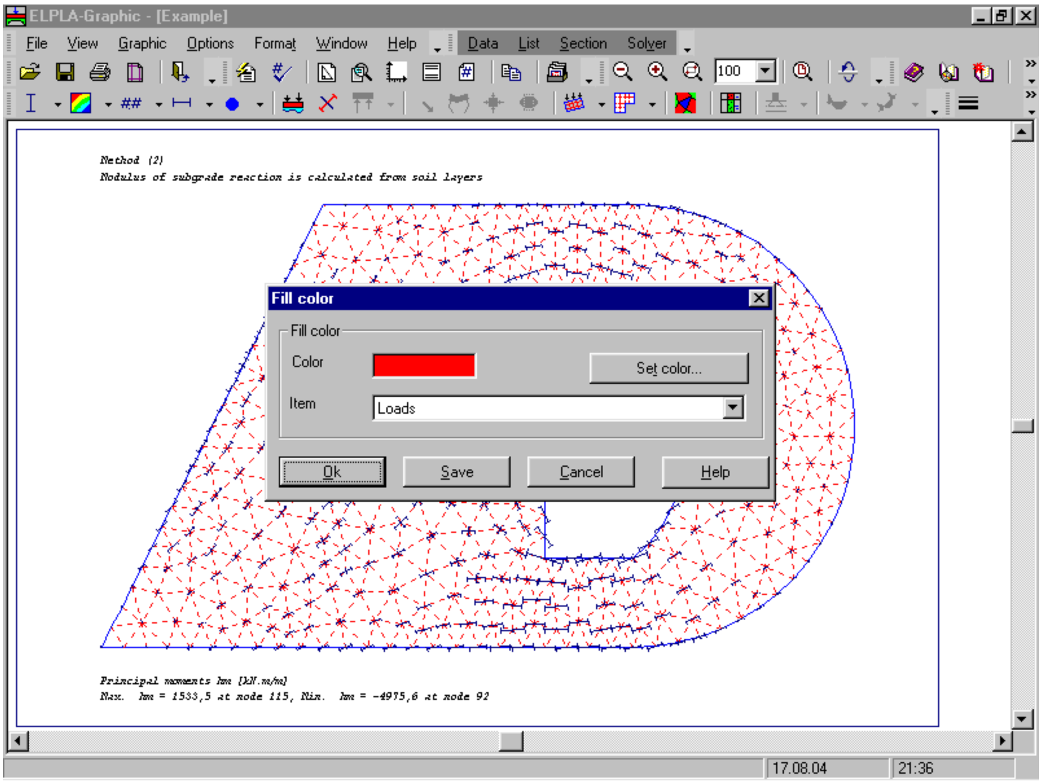


Figure B-20 Fill colors for Data and Graphic are the same setting

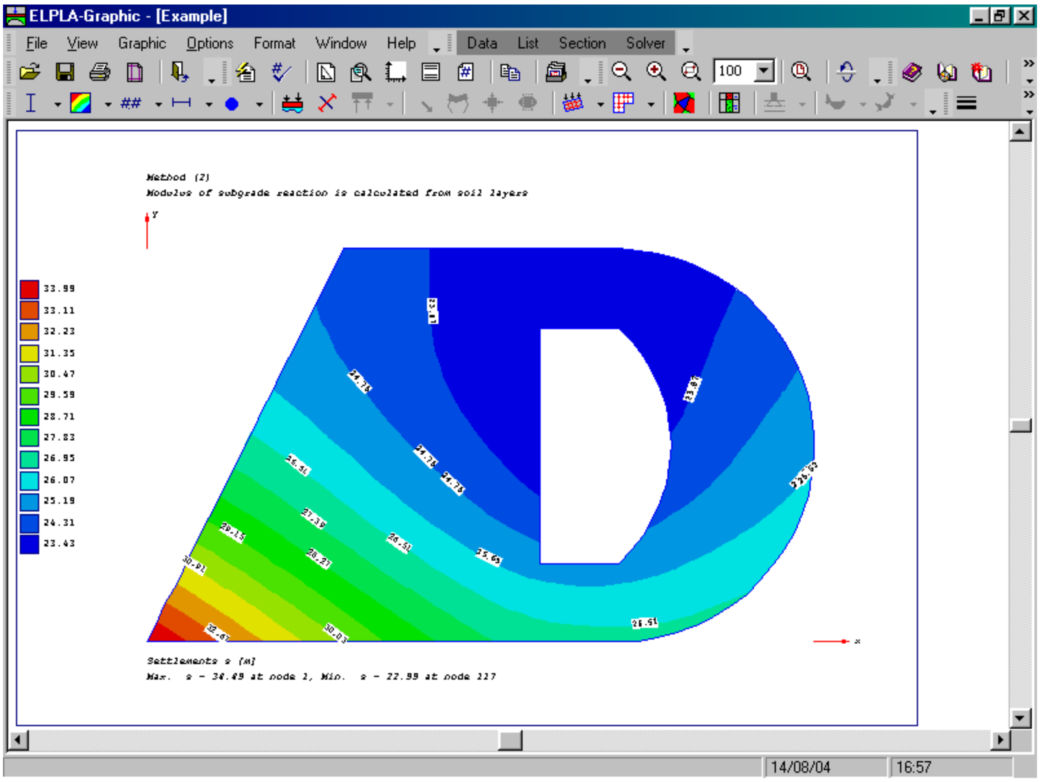


Figure B-21 Boundaries between contour regions are very smooth

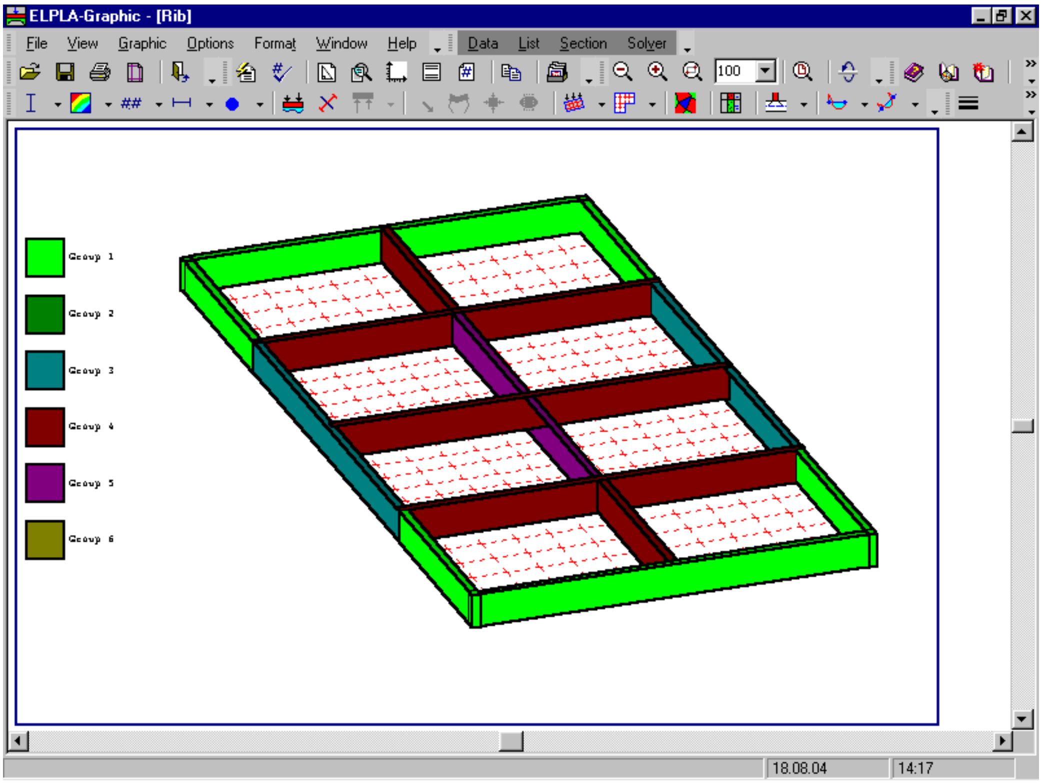


Figure B-22 Girder system in isometric view with actual size

2.15 Diagrams

- It is easy to define the required section where a plan of the raft with the chosen section is displayed when defining that section, Figure B-23

- It can be drawn with the diagram, a legend shows the plan of FE-Net with line indicating the chosen section, Figure B-24

- It is possible to plot a diagram at any section of the raft, Figure B-24

- It is possible to export diagrams from ELPLA-Section to MS Excel

- It is possible to define a diagram in ELPLA-Graphic by Mouse and to send it to ELPLA-Section, Figure B-26

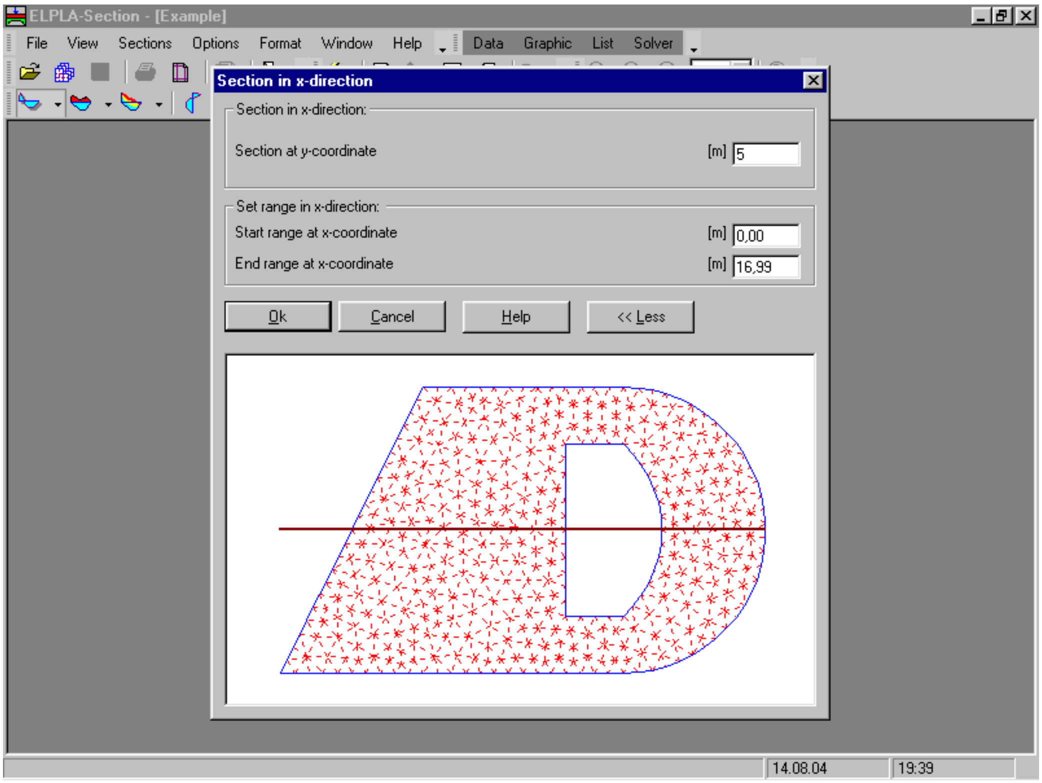


Figure B-23 Menu "Section in x-direction"

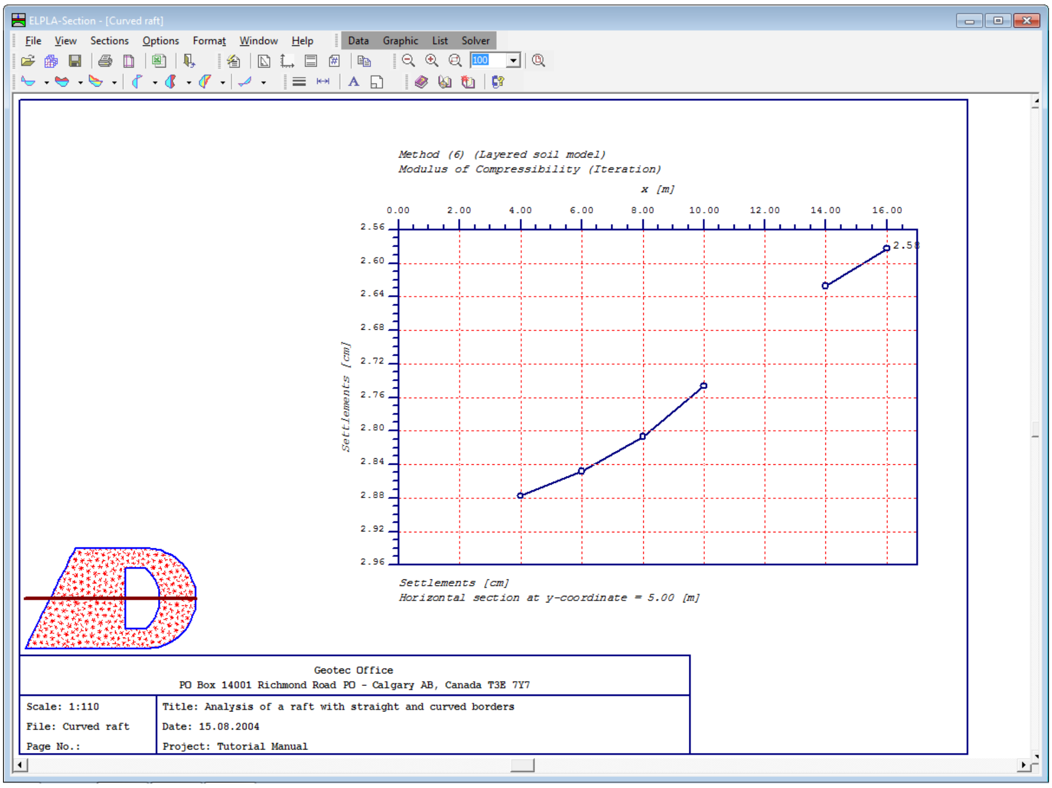


Figure B-24 Diagram with a legend showing a plan of FE-Net

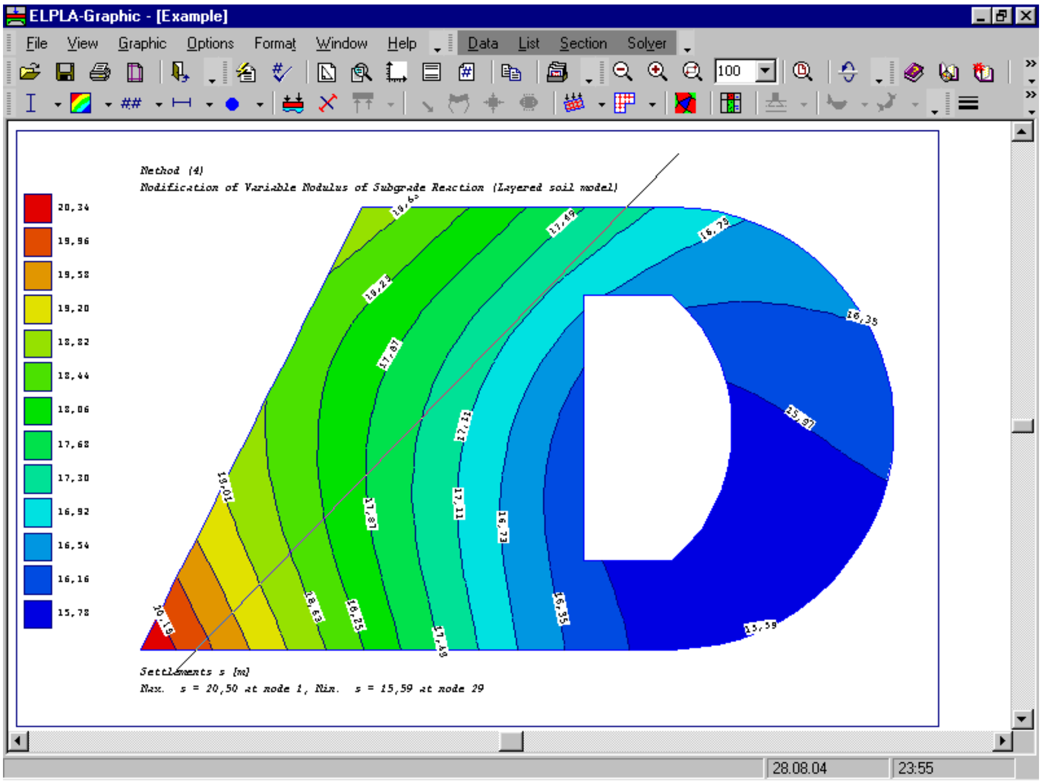


Figure B-26 Defining a diagram in ELPLA-Graphic

2.16 Loads

- It is possible to define a polygon load with variable ordinates to represent dam or embankment loads, Figure B-27

- It is possible to define a line moment to represent moments from walls or line supports, Figure B-27

- Point load is never applied in reality. If a point load represents a column load on a mesh of refine finite elements, the moment under the column will be higher than the real moment. To take the effect of the load distribution through the slab thickness, the column load must be distributed outward at 45 [°] from the column face until reaching the center line of the slab. Now it is possible, to overcome this problem by converting the point load to an equivalent uniform load over an appropriate area, Figure B-28

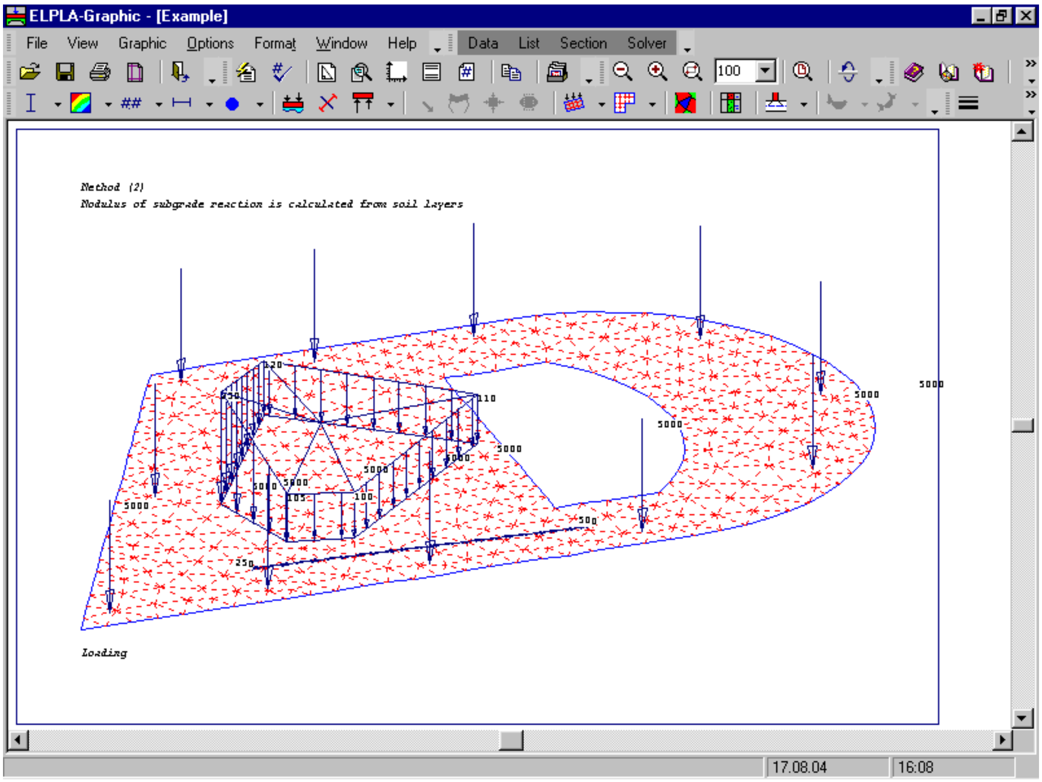


Figure B-27 Polygon load with variable ordinates and line moment

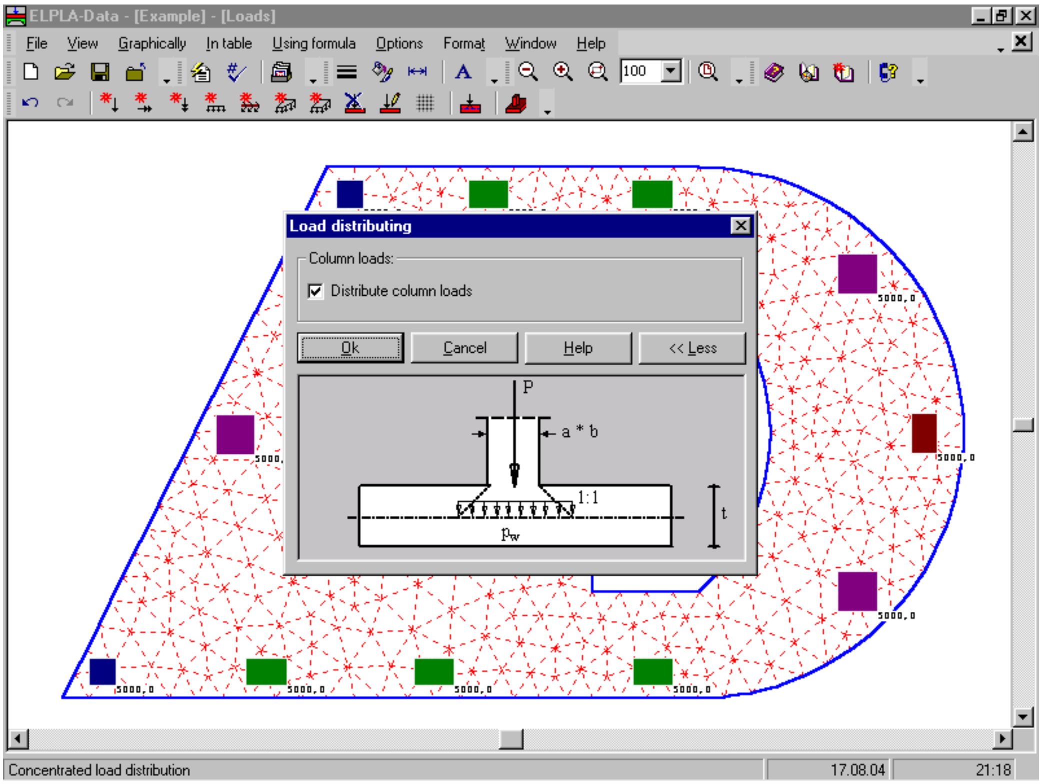


Figure B-28 Distributing the column point load over an appropriate area

2.17 Column cross-section and punching area

- It is possible to define column dimensions that are used to design columns for punching and to convert column concentrated load to equivalent distributed load, Figure B-29

- Column cross-section can be drawn with column load, Figure B-29

- It is possible to draw column cross-sections in colored groups to easy identify the column capacity, Figure B-30

- Area of punching can be displayed according to the specified code of design with column cross-section, Figure B-31

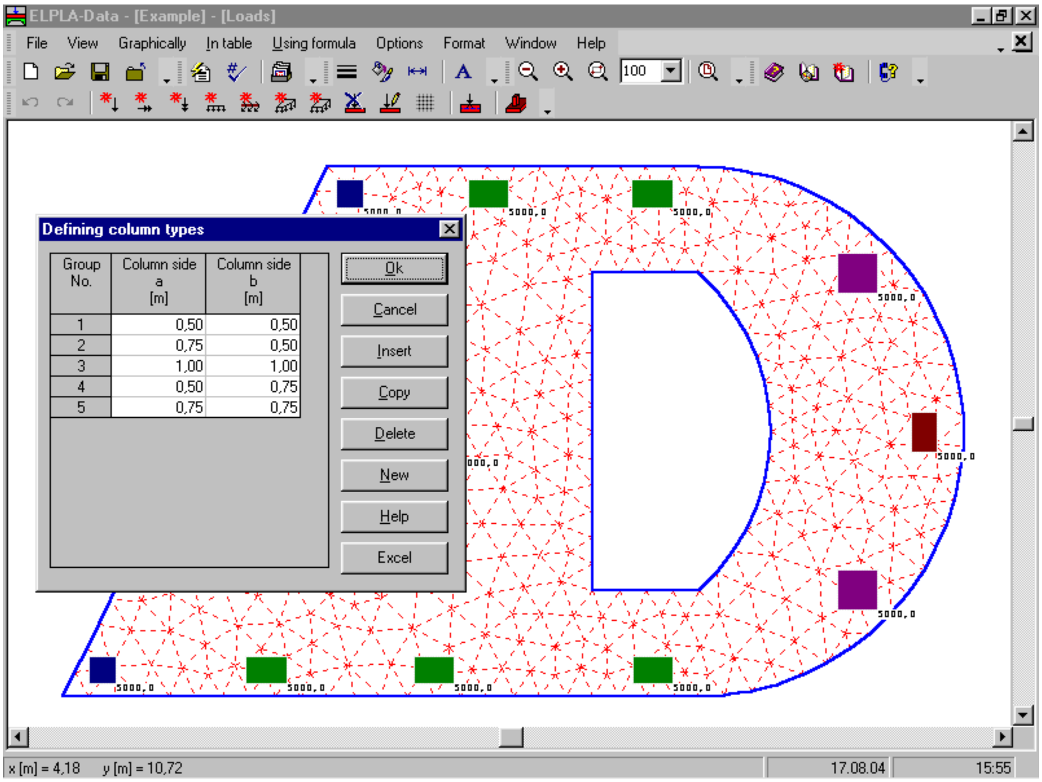


Figure B-29 Column cross-sections with column loads

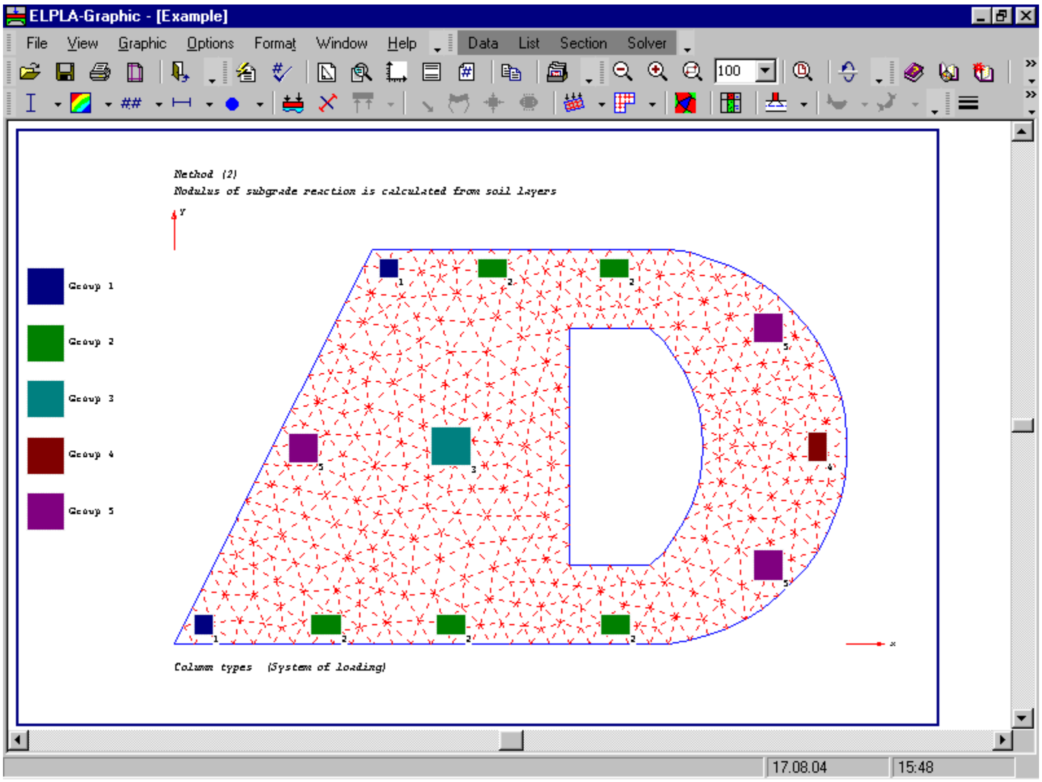


Figure B-30 Column cross-sections in colored groups

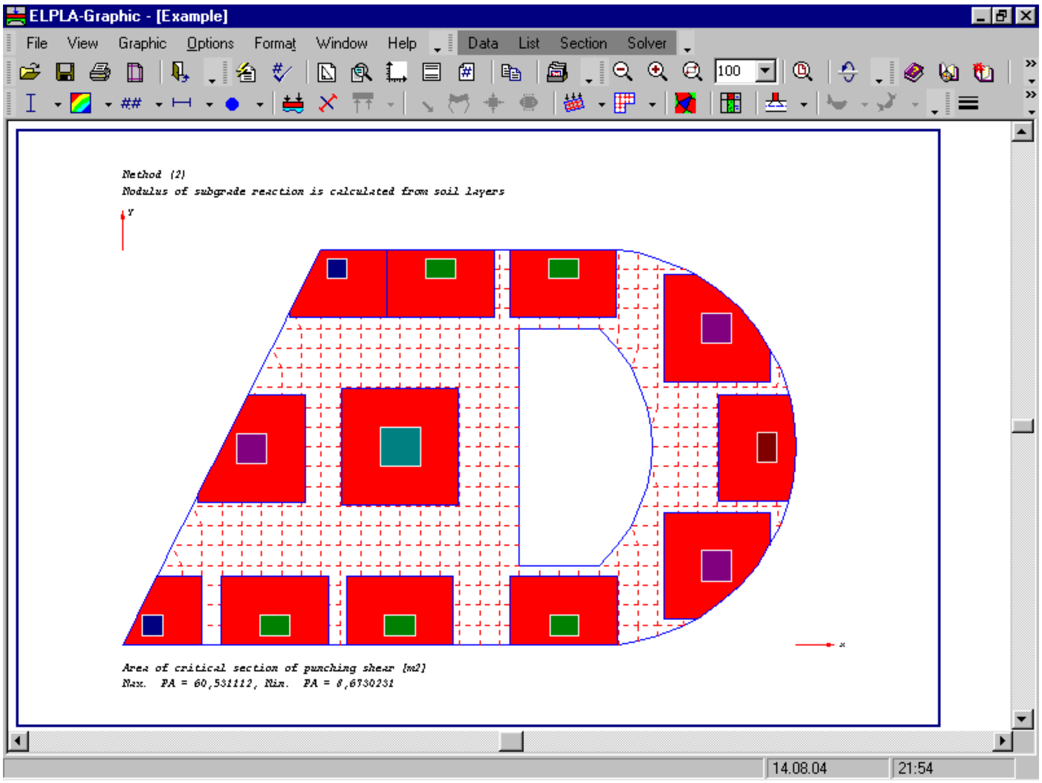


Figure B-31 Punching areas according to ACI with column cross-sections

2.18 Output list

- It is possible to open many projects at the same time in one view in the program ELPLA-List to compare between their results, Figure B-32

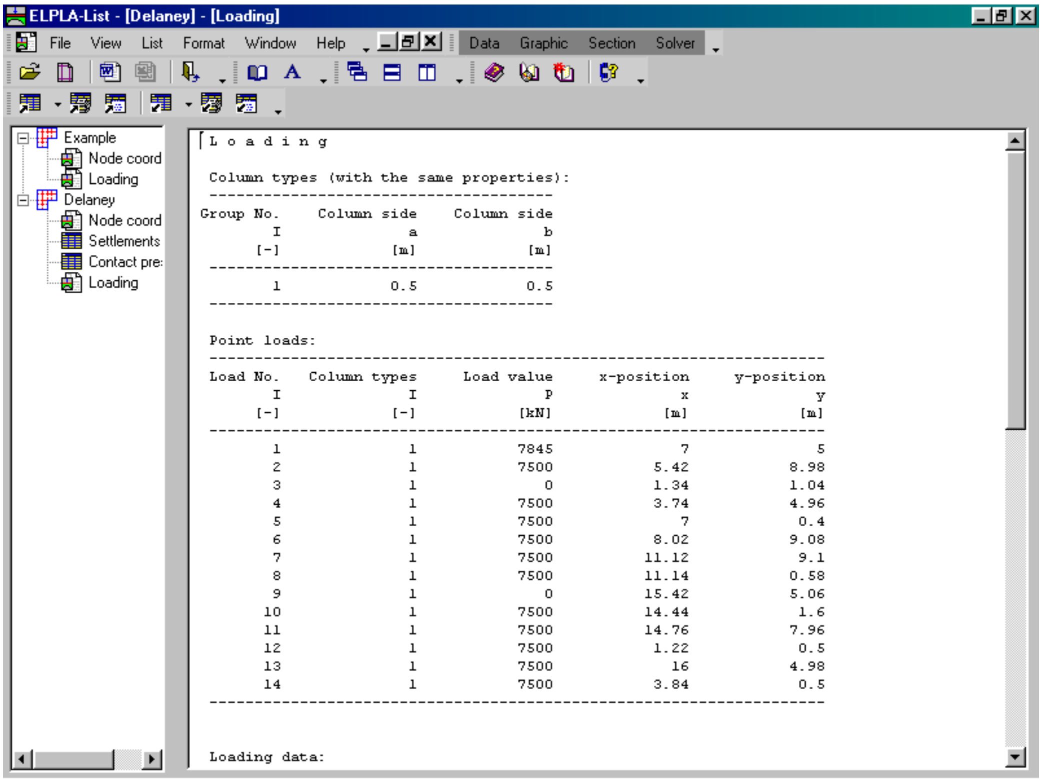


Figure B-32 Two projects are opened at the same time in the program ELPLA-List

3 Enhancements in version ELPLA 8.1

3.1 Analysis of grid

Besides the three different analyses available in the program ELPLA to analyze foundations and slab floors, a new analysis "Analysis of grid" is added (Figures B-33 and B-34).

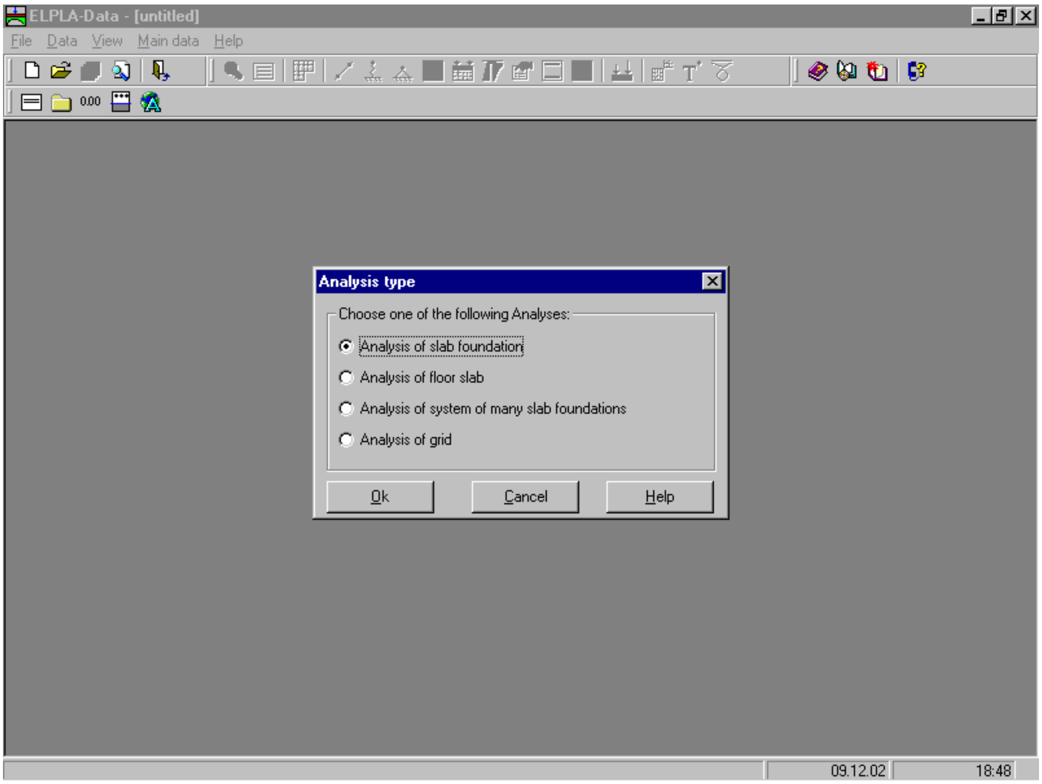


Figure B-33 Menu "Analysis type"

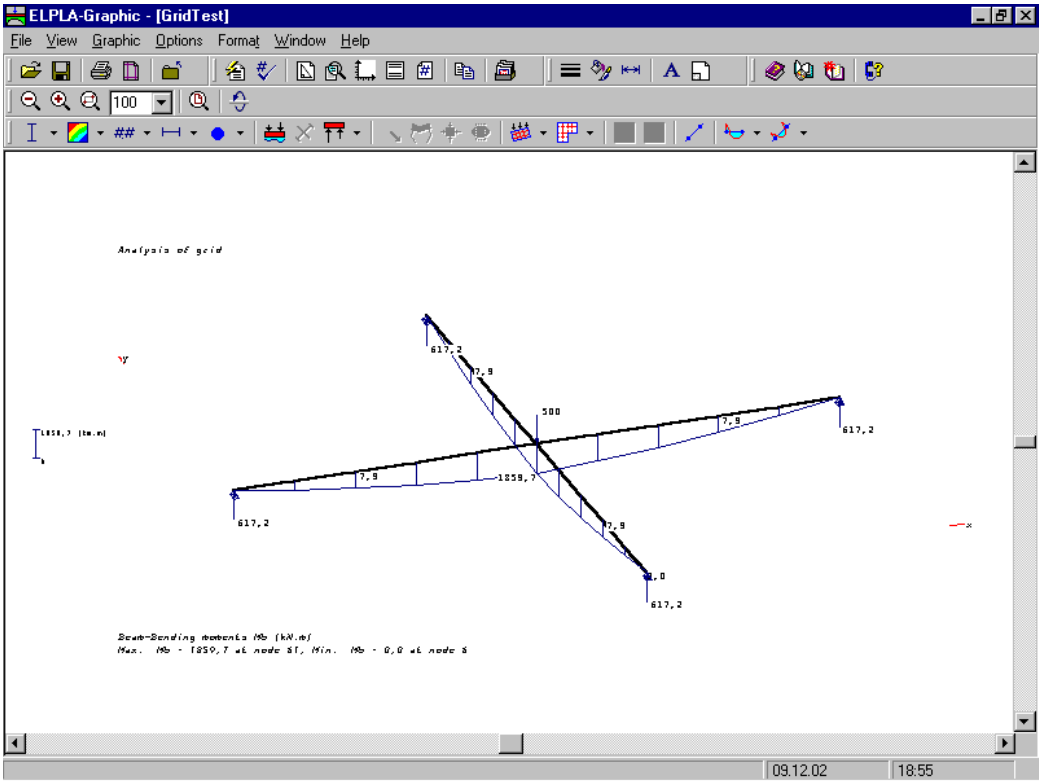


Figure B-34 Beam-Moments

3.2 Determining stresses, strains and displacements in soil

You can determine the stresses, strains and displacements in soil under the foundation in a net in z-direction (Figures B-35 to B-38).

ELPLA can display results of stresses, strains and displacements in different forms such as:

- Soil deformation as deformed mesh, Figure B-39
- Soil deformation as vectors, Figure B-40
- Principal soil stresses as streaks, Figure B-41
- Principal soil strains as streaks, Figure B-42

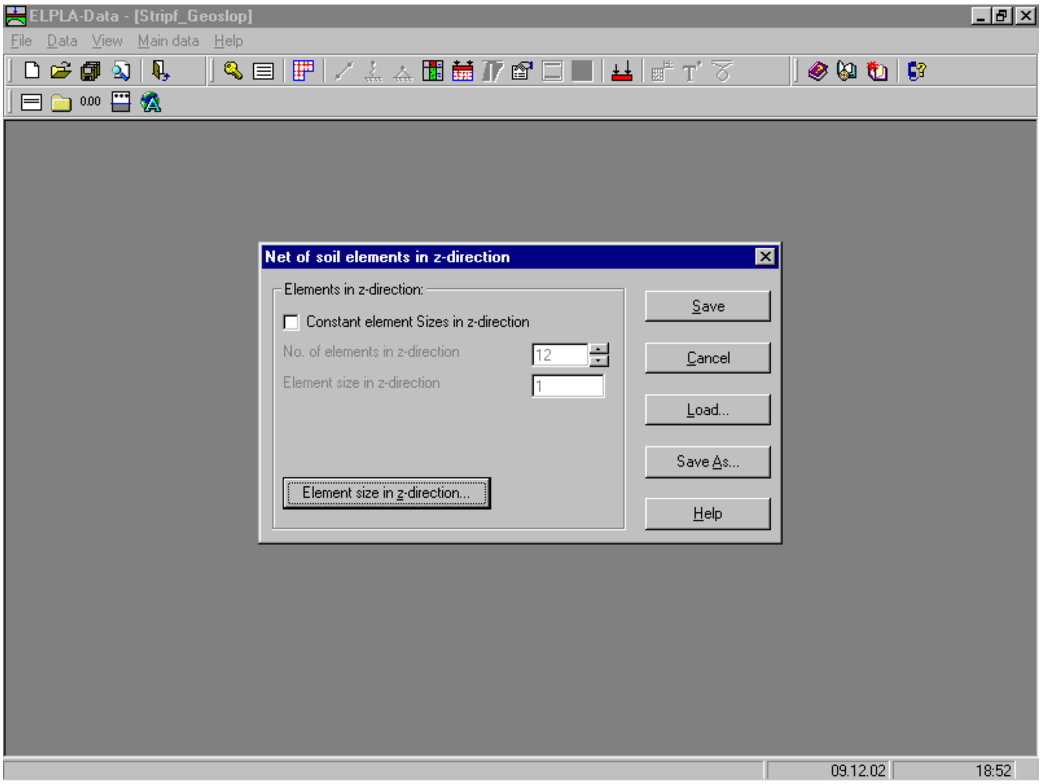


Figure B-35 Menu "Net of soil elements in z-direction"

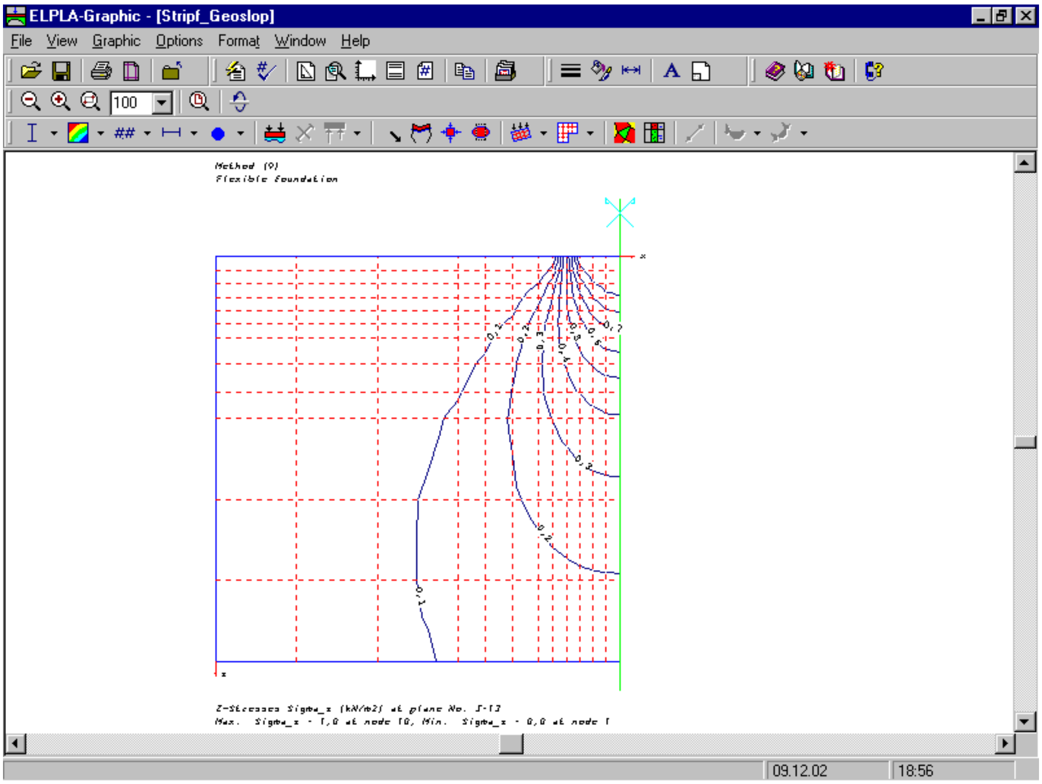


Figure B-36 Z-Stresses Sigma_z

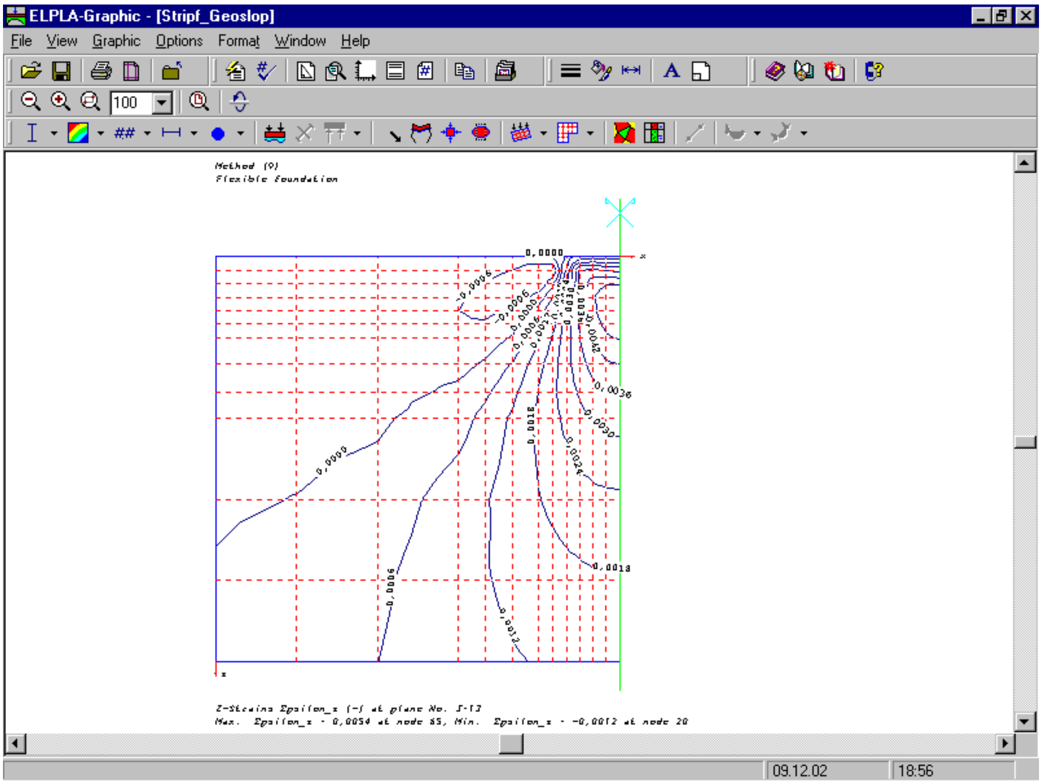


Figure B-37 Z-Strains Epsilon_z

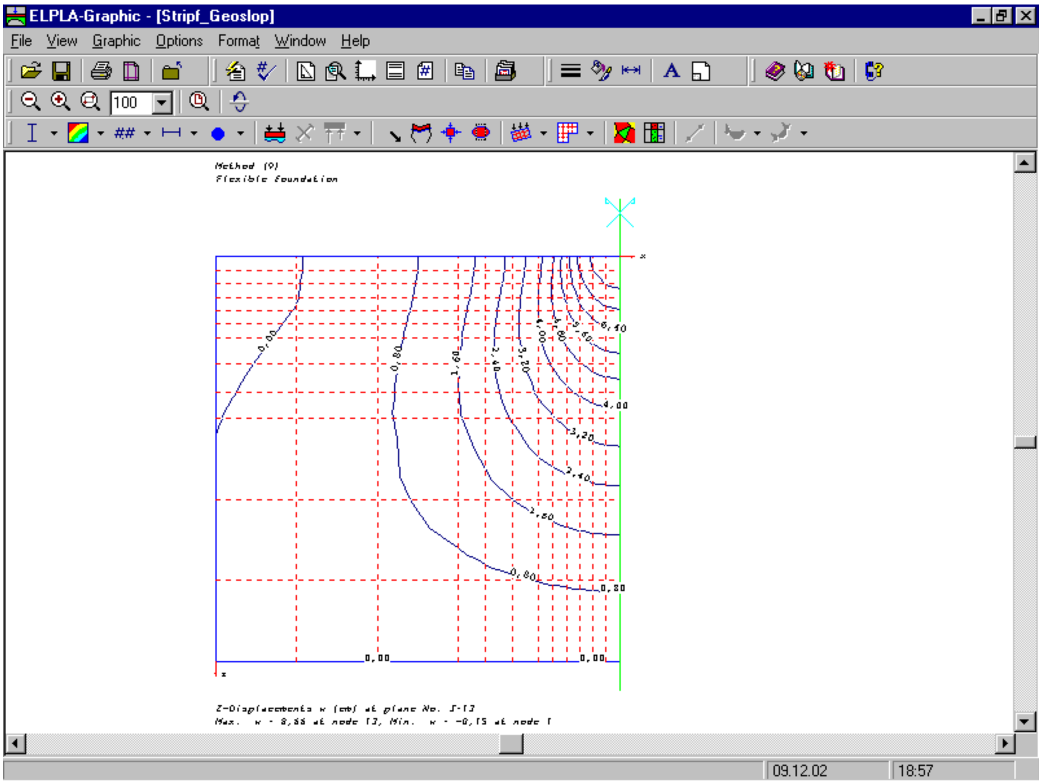


Figure B-38 Z-Displacements w

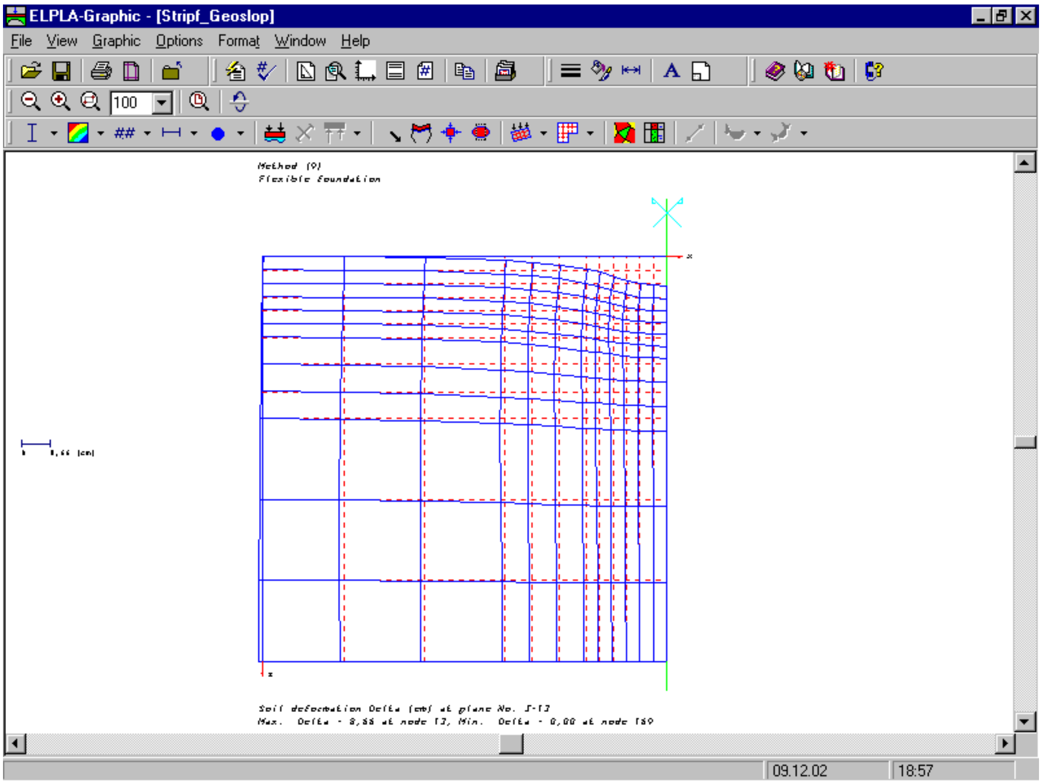


Figure B-39 Soil deformation as deformed mesh

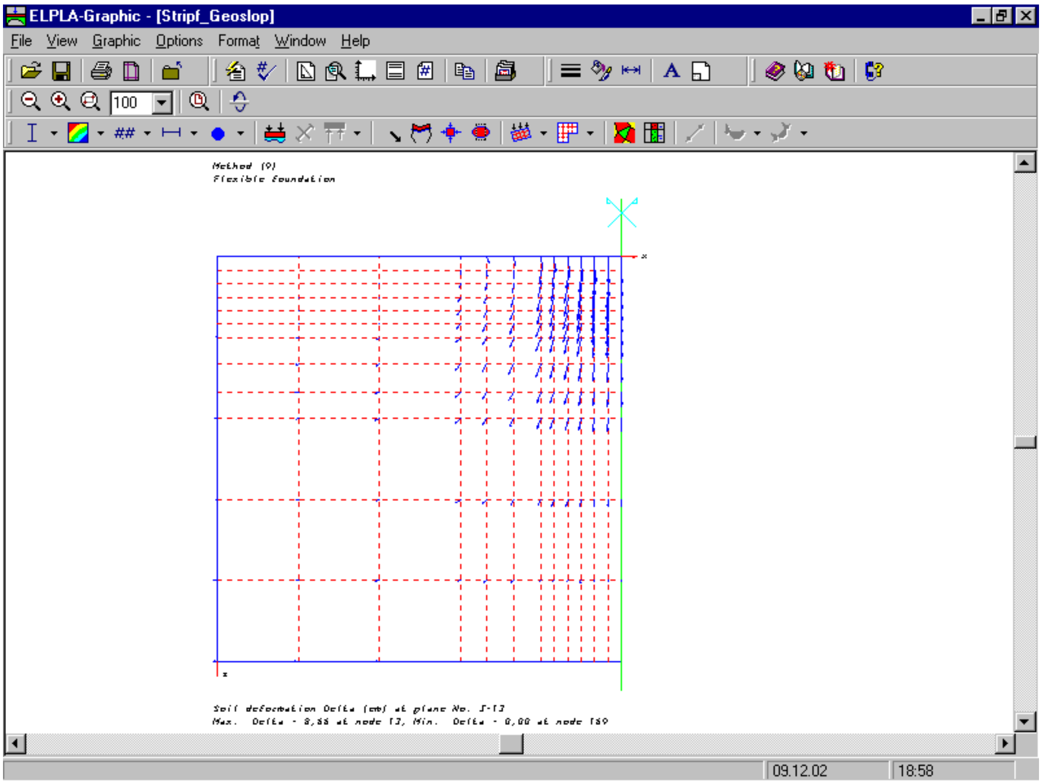


Figure B-40 Soil deformation as vectors

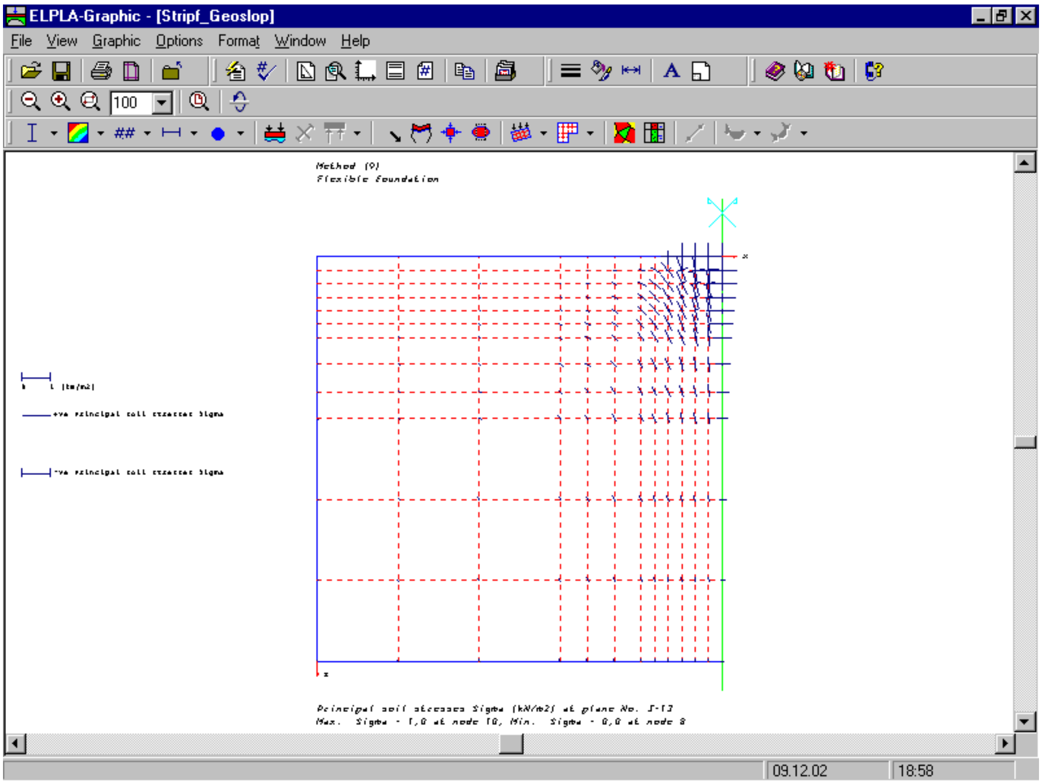


Figure B-41 Principal soil stresses as streaks

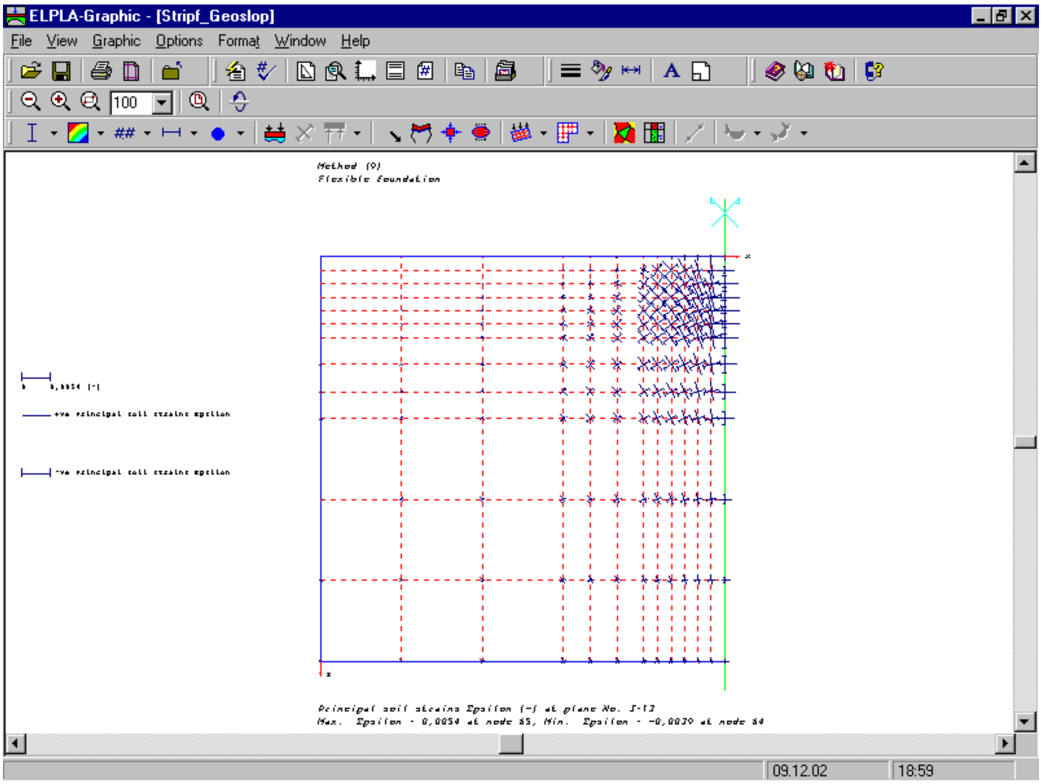


Figure B-42 Principal soil strains as streaks

3.3 Language of the help

You can define the language of the help system used in ELPLA applications. The three languages are English, German and Arabic (Figures B-43 to B-46).

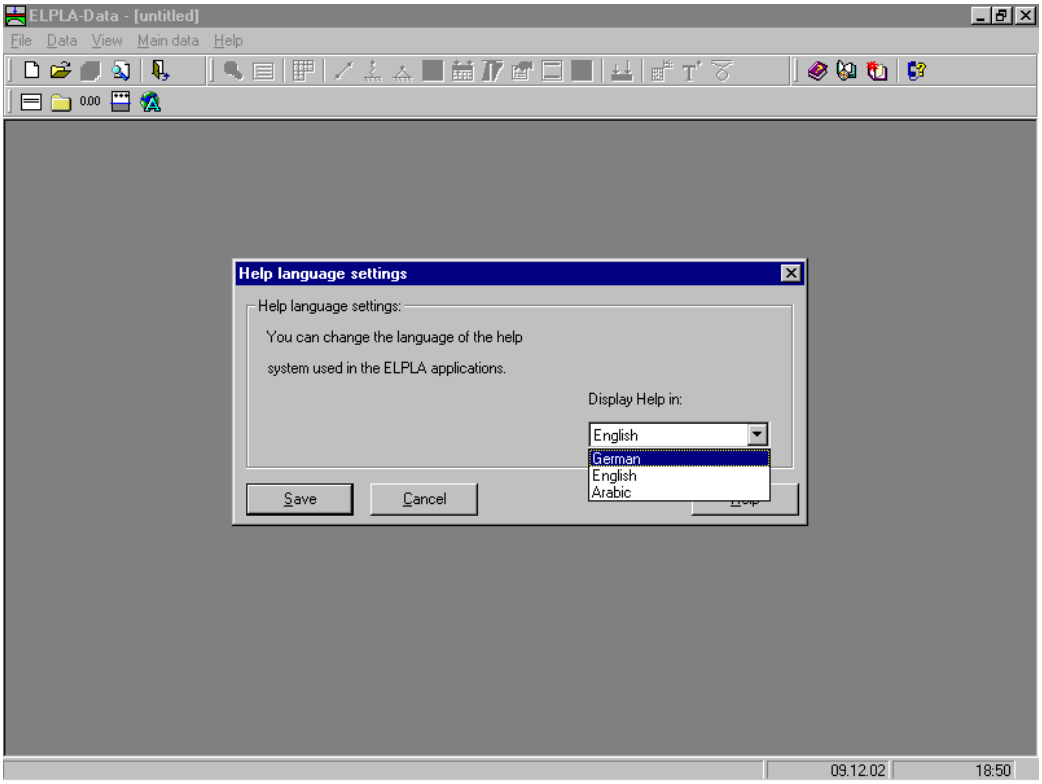


Figure B-43 Menu "Help language settings"

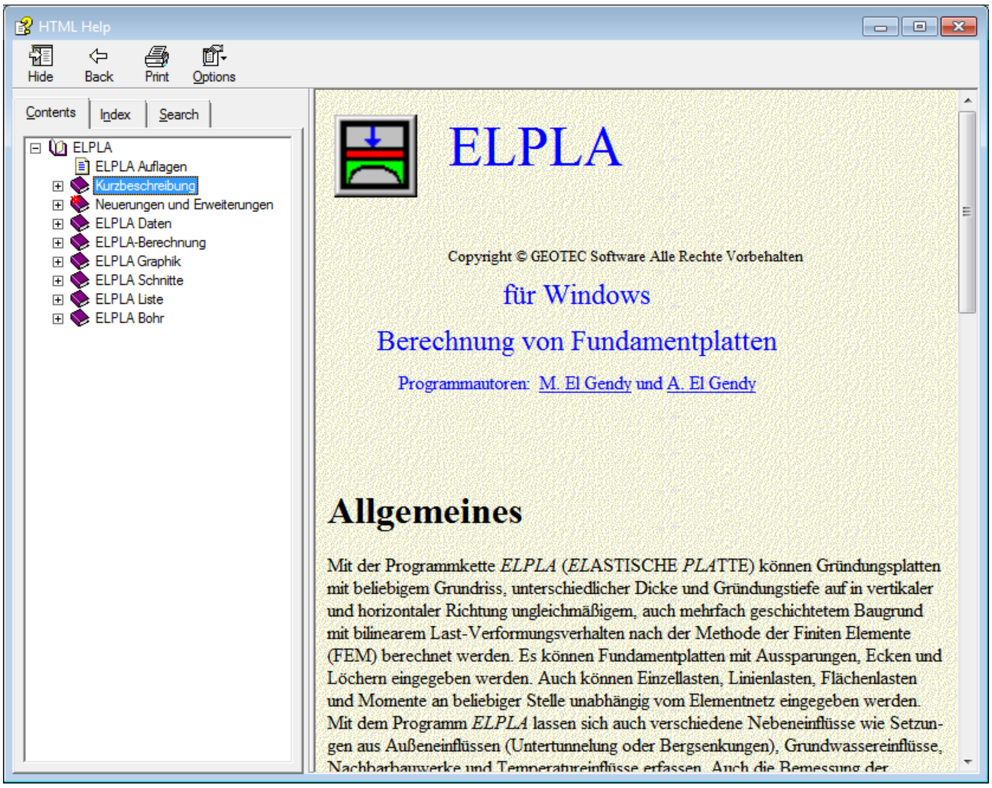


Figure B-44 Help language in German

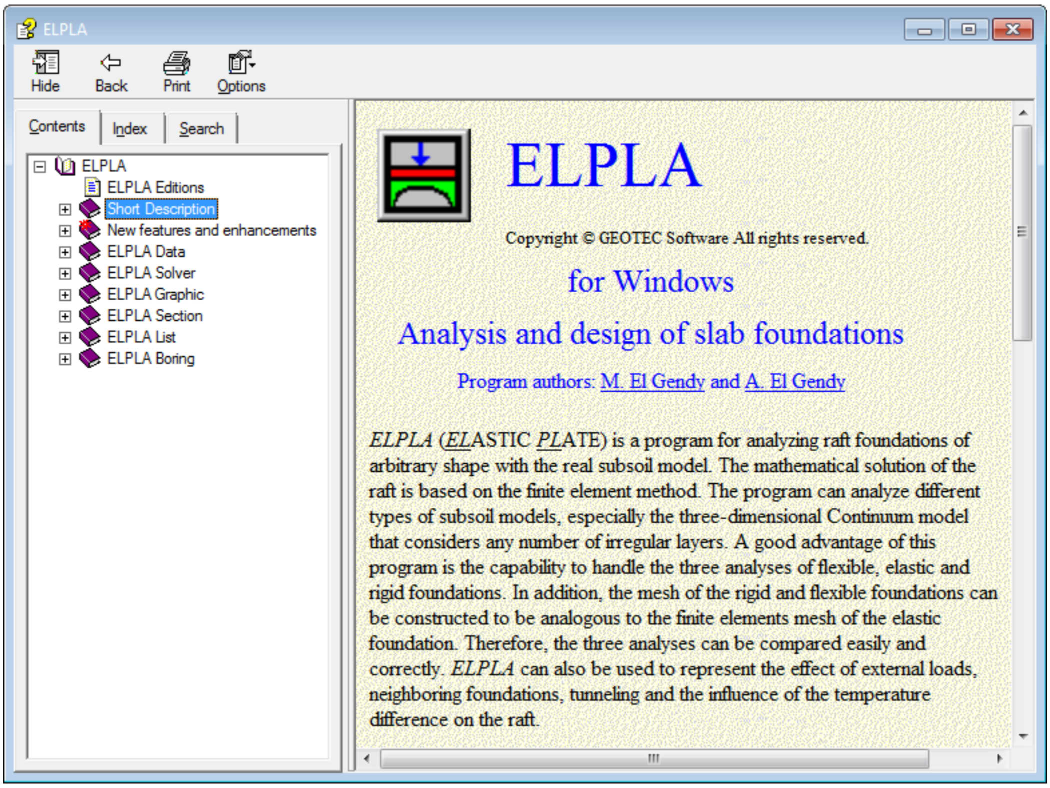


Figure B-45 Help language in English

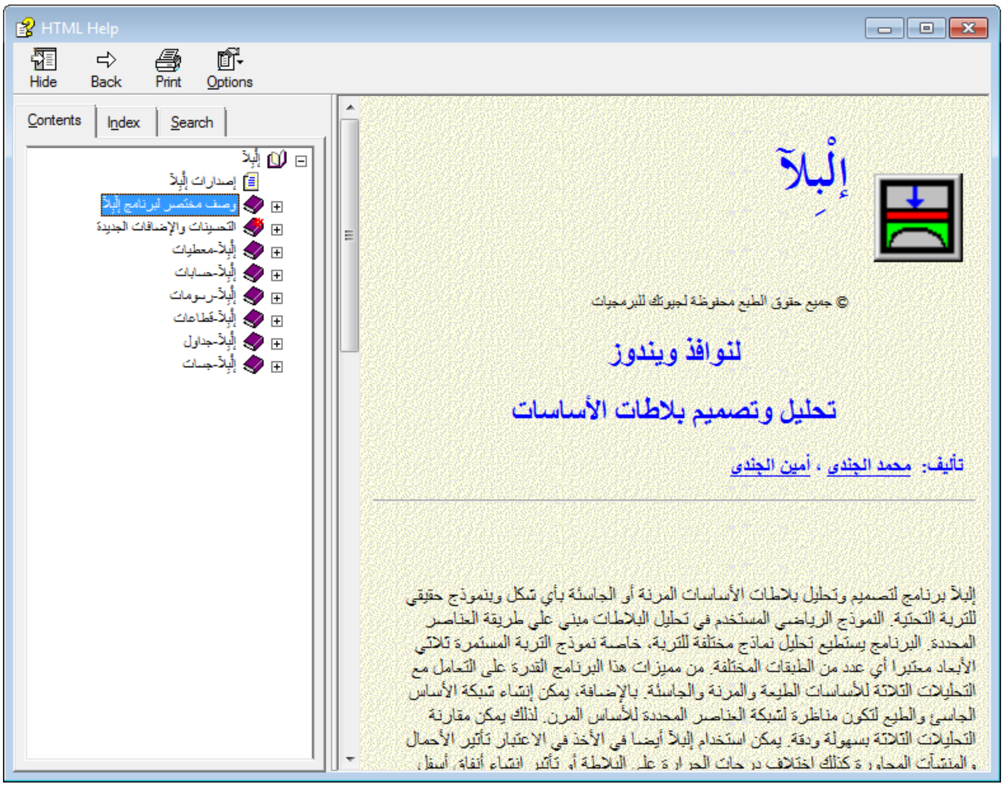


Figure B-46 Help language in Arabic

3.4 Converting Loads

Contact pressures on nodes or reactions may be converted to applied loads on these nodes, Figure B-47.

3.5 Displaying axis arrows

You can display axis arrows in x-, y- and z-directions, Figure B-48.

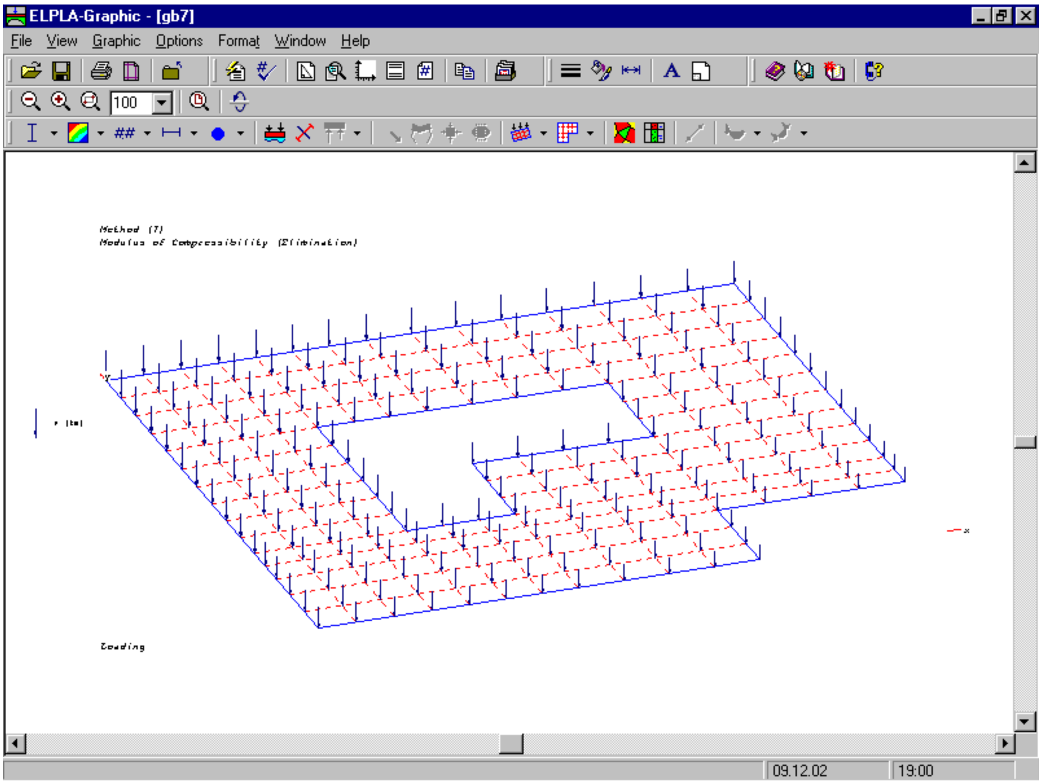


Figure B-47 Contact pressures on nodes as applied loads

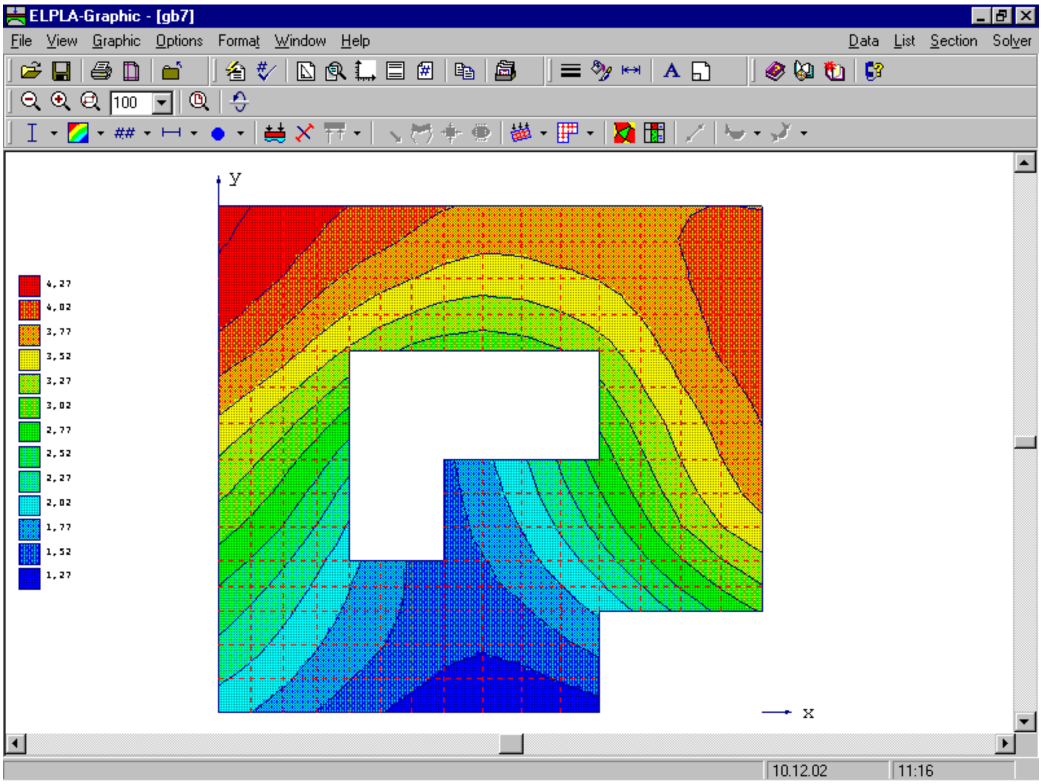


Figure B-48 Raft with x- and y-axis arrows

3.6 Boring logs

The display of soil properties C and Phi is optional. This option allows displaying more borings together, Figure B-49.

Soil colors according to German Standard DIN 4023 are considered automatically, if desired, in the drawing of boring logs, Figure B-50.

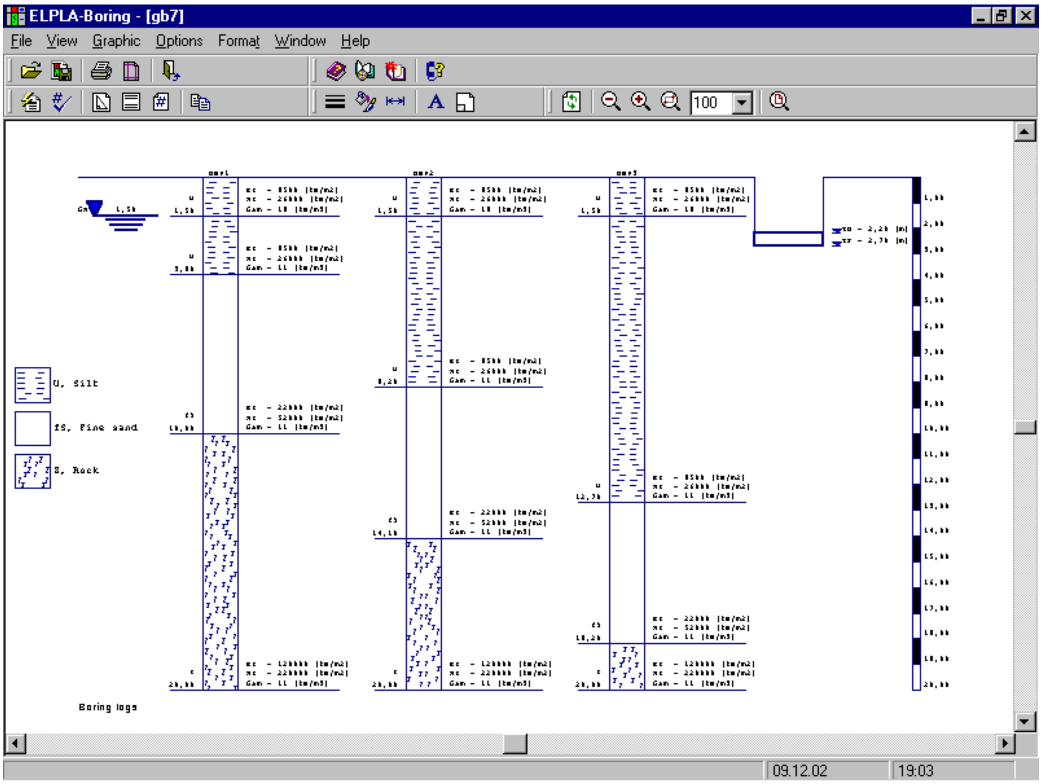


Figure B-49 Many borings together

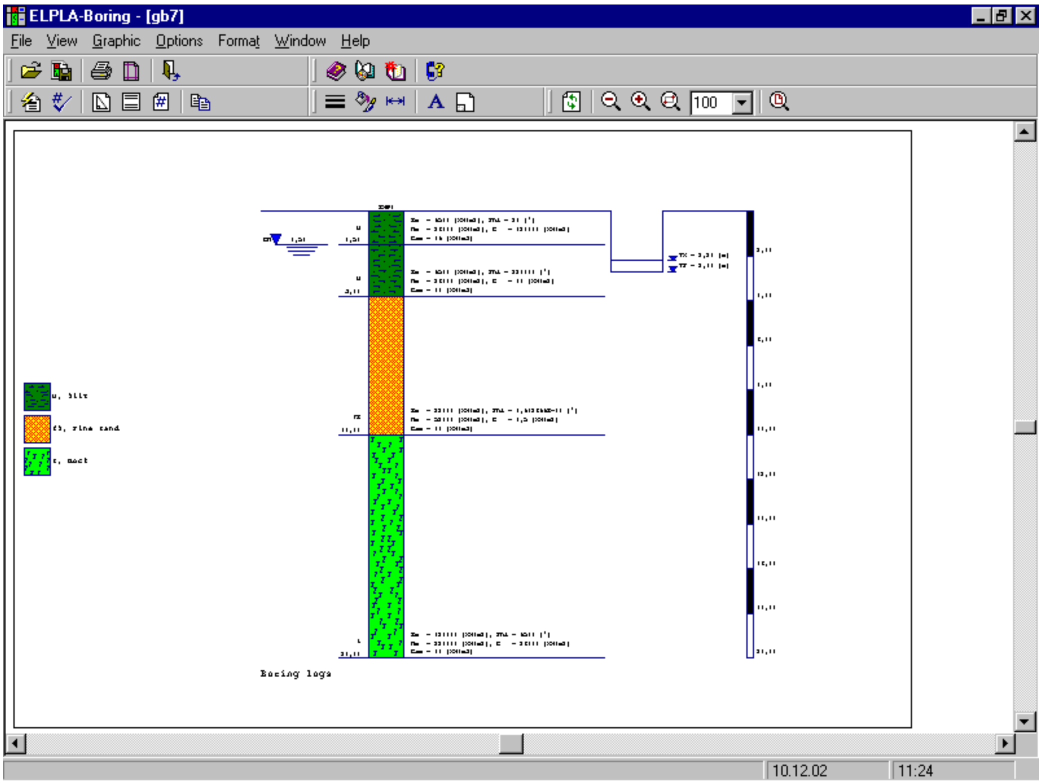


Figure B-50 Boring log with soil colors according to German Standard DIN 4023

4 Enhancements in version ELPLA 8.0

4.1 Calculation methods

- Besides the eight different calculation methods available in ELPLA to analyze foundations, a new method "Flexible foundation (method 9)" is added. This method can be used for settlement calculations of flexible foundations such as embankments, dams or direct loads on the ground, Figure B-51

- You can analyze system of flexible, elastic and rigid foundations together, Figure B-52

- You can determine the influence of flexible neighbor foundations or external loads of different arts on the examined foundation. Until now only the influence of elastic or rigid neighbor foundations could be considered, Figure B-53

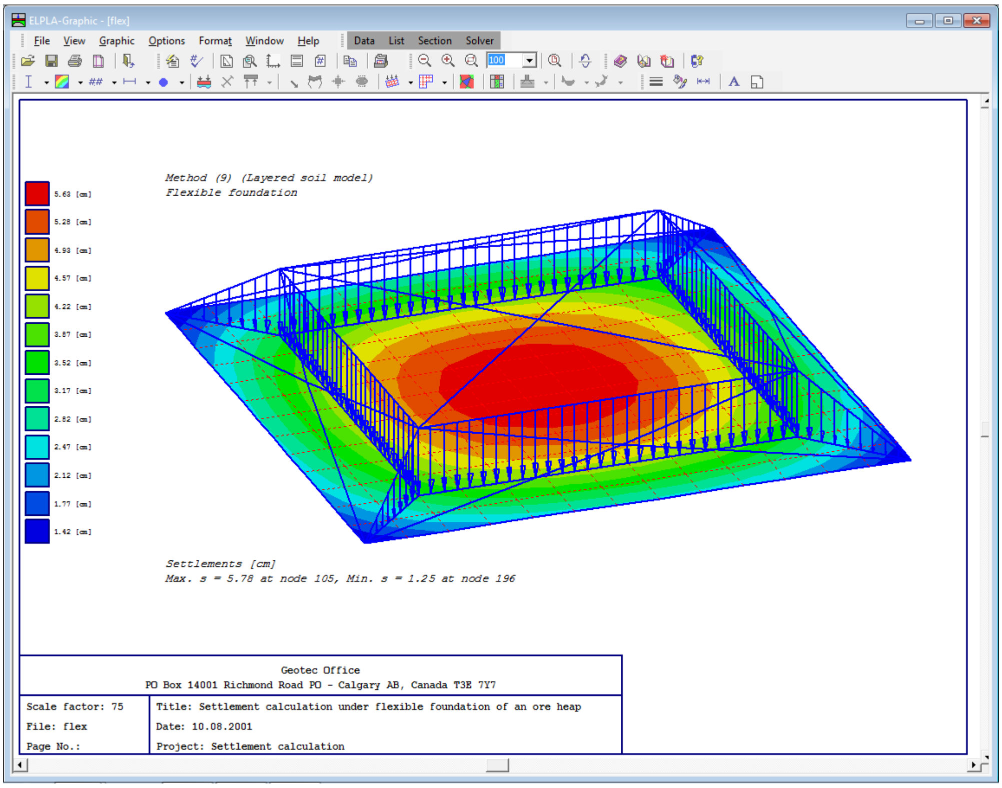


Figure B-51 Flexible foundation

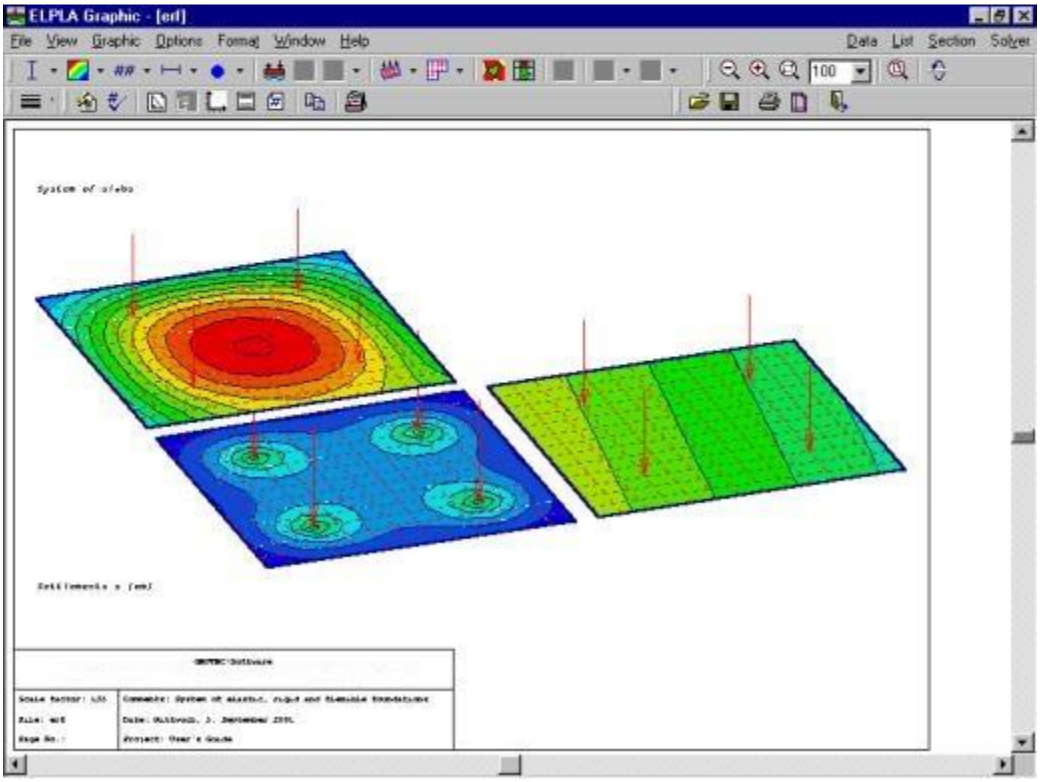


Figure B-52 System of flexible, rigid and elastic foundations

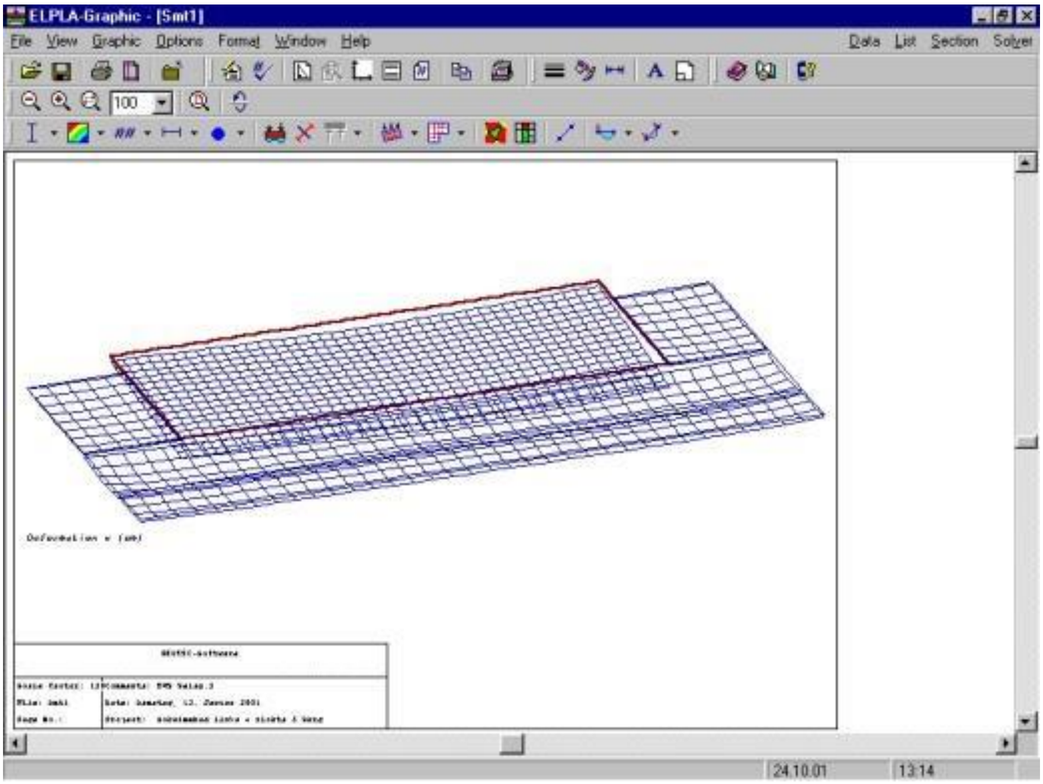


Figure B-53 Influence of flexible neighbor foundations

- You can introduce translational or rotational spring stiffness on nodes for the rigid slab (method 8). Until now, it was possible only for methods 1 to 7. The springs can be used for analysis of rigid pile caps, Figure B-54

- Negative contact pressure can be eliminated for methods 1 to 8, if it appears on the bottom of the foundation. Until now, elimination of negative contact pressures was available for methods 4, 6 and 8, Figure B-55

4.2 Soil

- Not only bilinear but also nonlinear soil deformations can be taken into account, as example for plastic deformations at heavy loading, Figure B-56

- For all nodes the bearing capacity of foundation on a variable soil can be determined through interpolation, Figure B-57

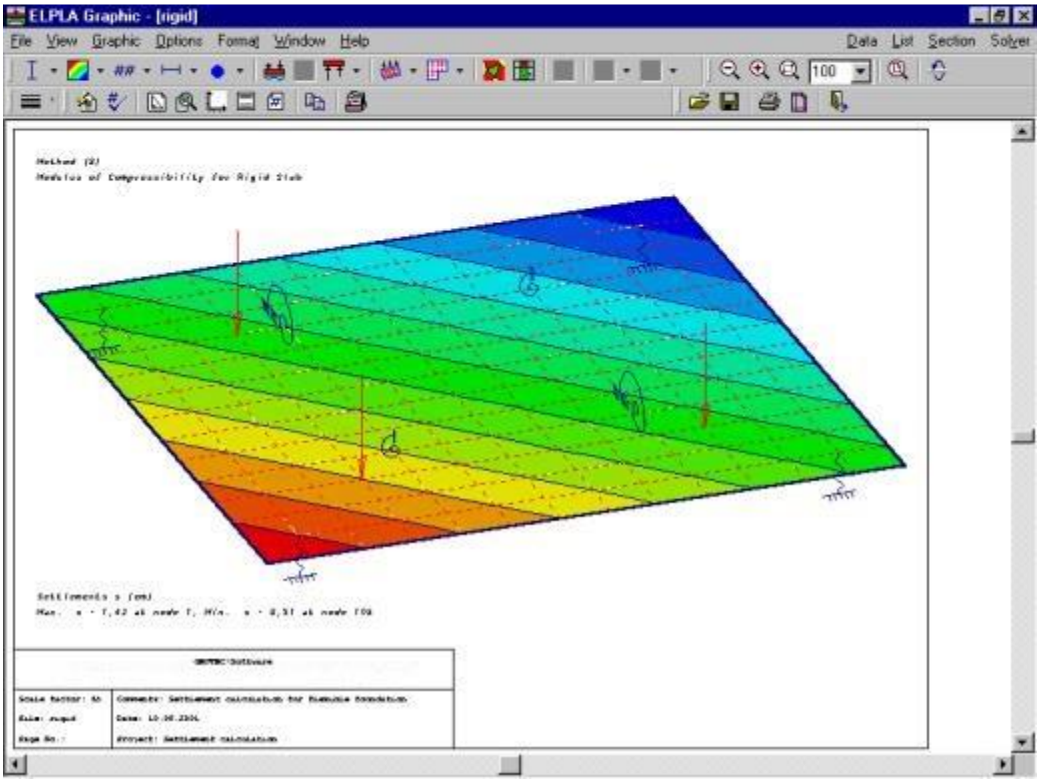


Figure B-54 Rigid raft with translational and rotational spring

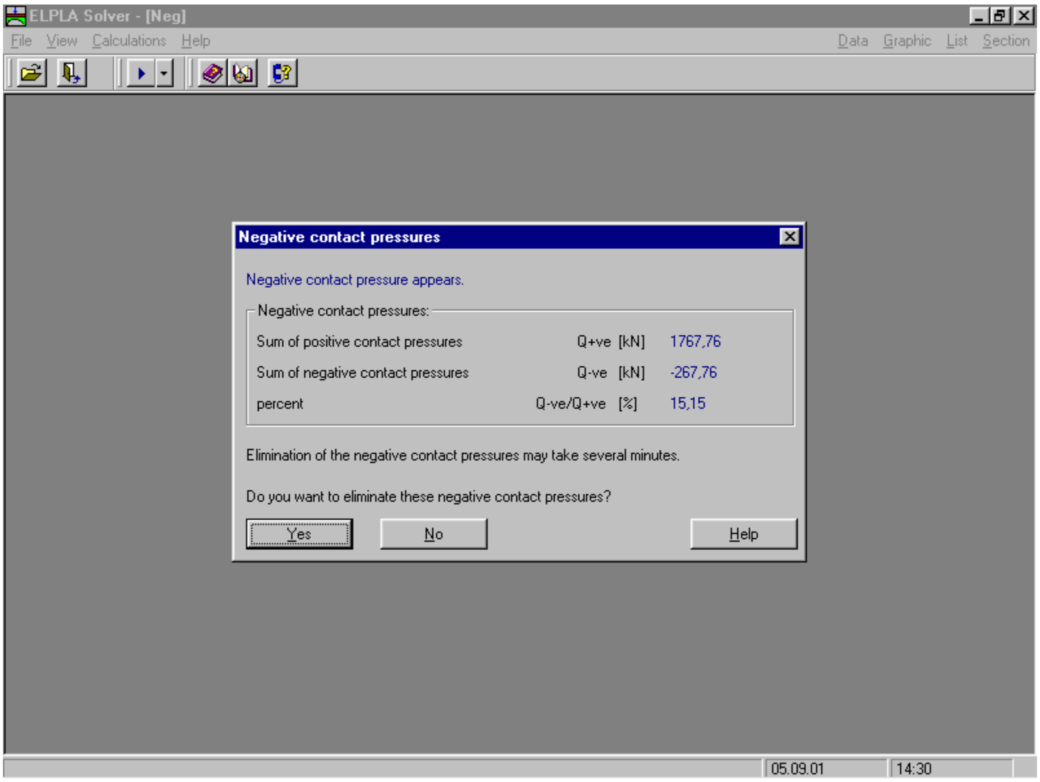


Figure B-55 Elimination of negative contact pressure

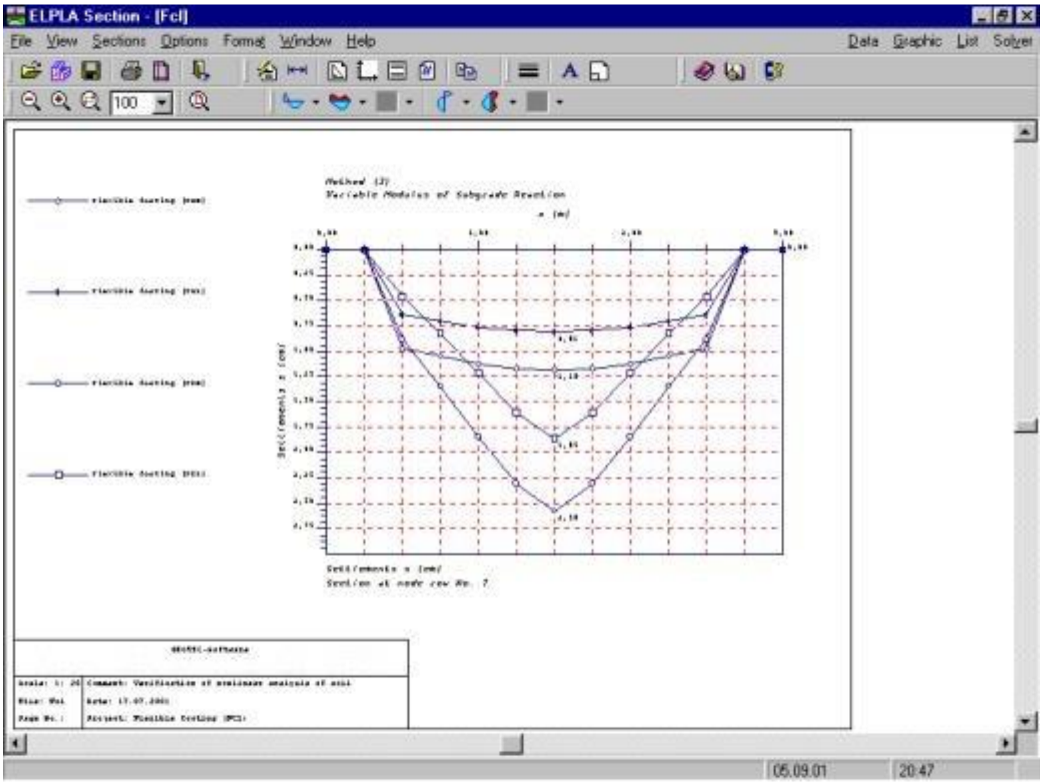


Figure B-56 Nonlinear soil deformation

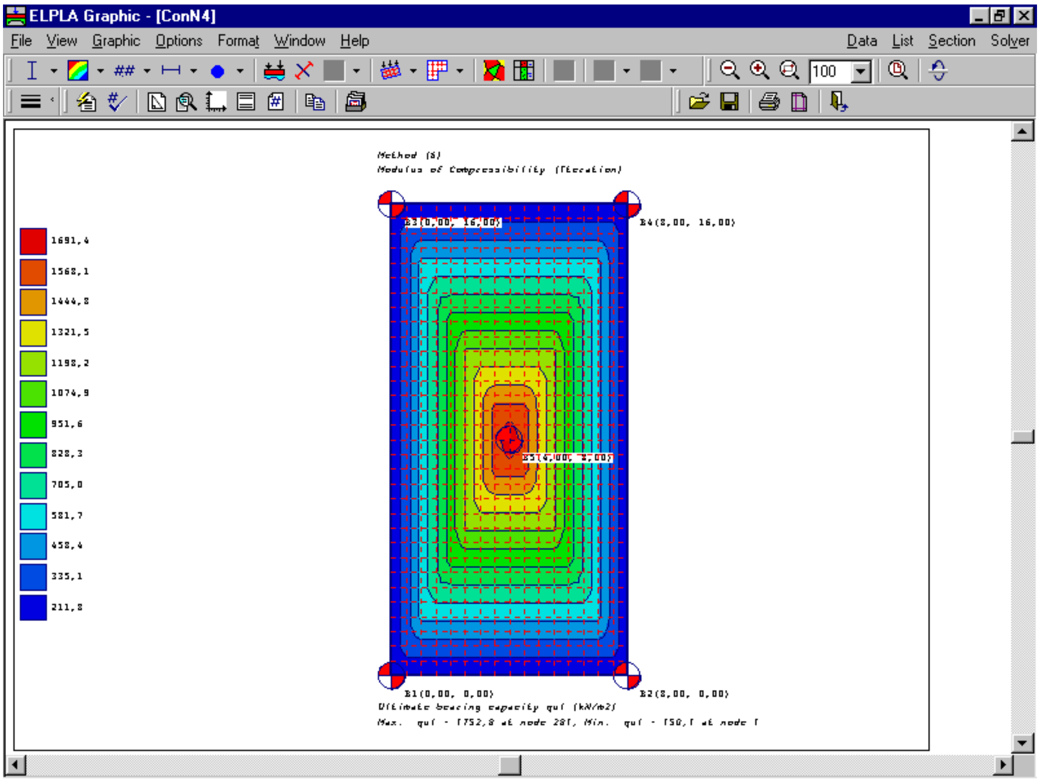


Figure B-57 Bearing capacity for variable soil

4.3 Plot parameters

- Line formats can be defined and saved, Figure B-58
- The fill color can be defined and saved, Figure B-59
- Font format (size, type …) can be defined and saved, Figure B-60
- You can define and save the maximum ordinate, length, side or diameter of symbols for the drawing, Figure B-61

4.4 Input data

- The menus are extended by some explanation figures, Figure B-62
- The values of data, if desired, can be displayed on the FE-Net, Figure B-63
- The additional settlement can be defined graphically, Figure B-64
- Boring fields can be defined graphically, Figure B-65
- Boring logs can be inserted from a file, Figure B-66
- Besides the definition of girder section by moment of inertia and torsion moment of inertia, the section can be also defined by its width and height, Figure B-67

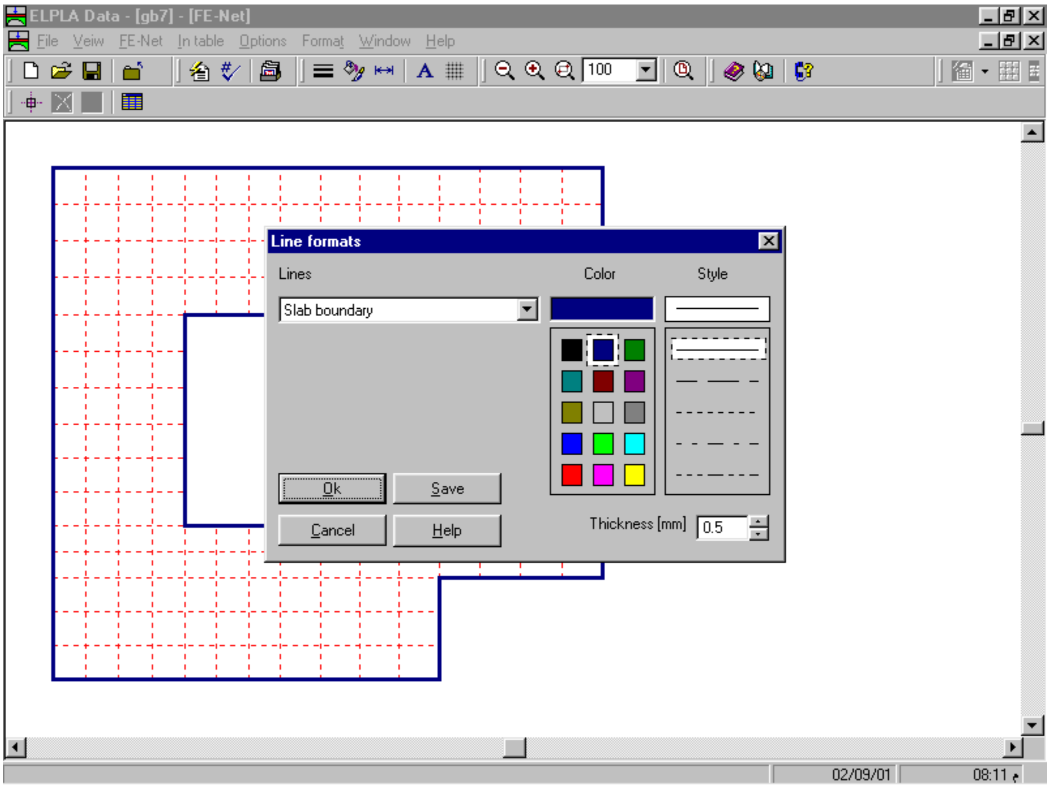


Figure B-58 Line formats

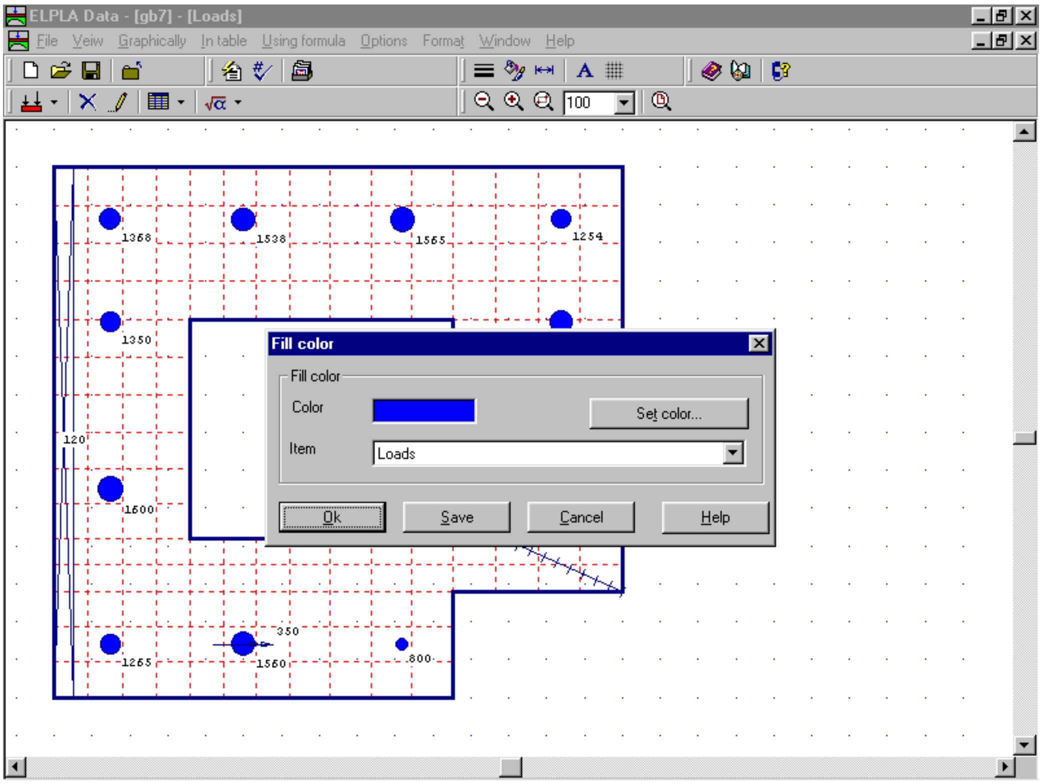


Figure B-59 Fill color

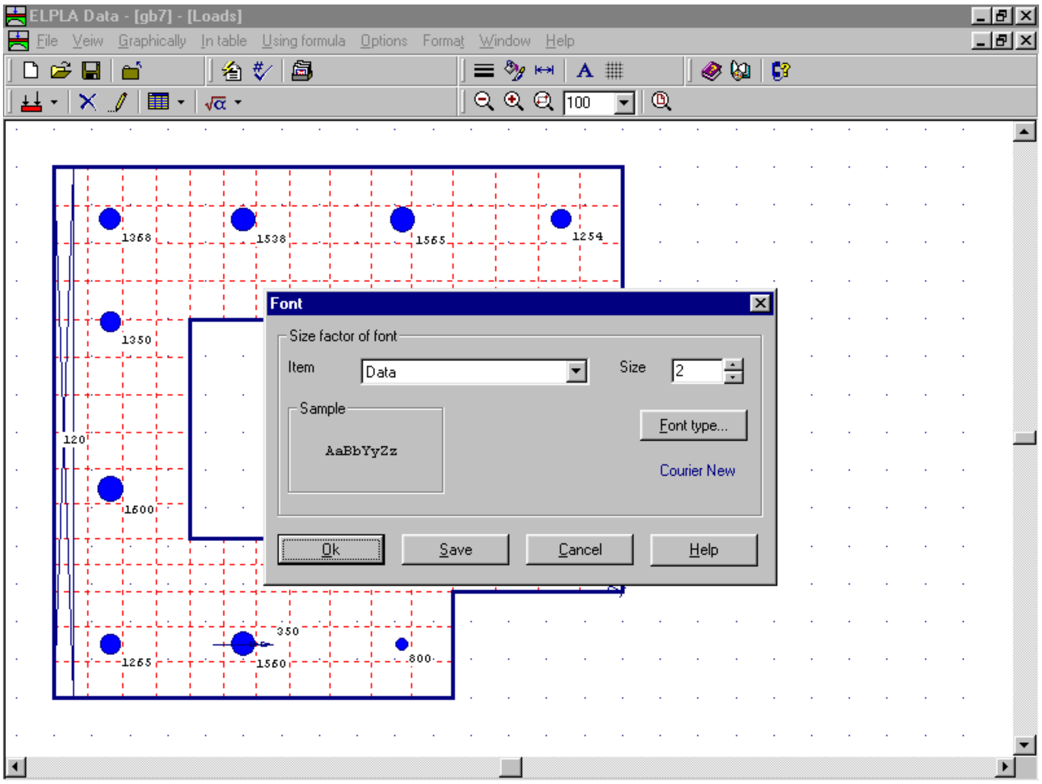


Figure B-60 Font

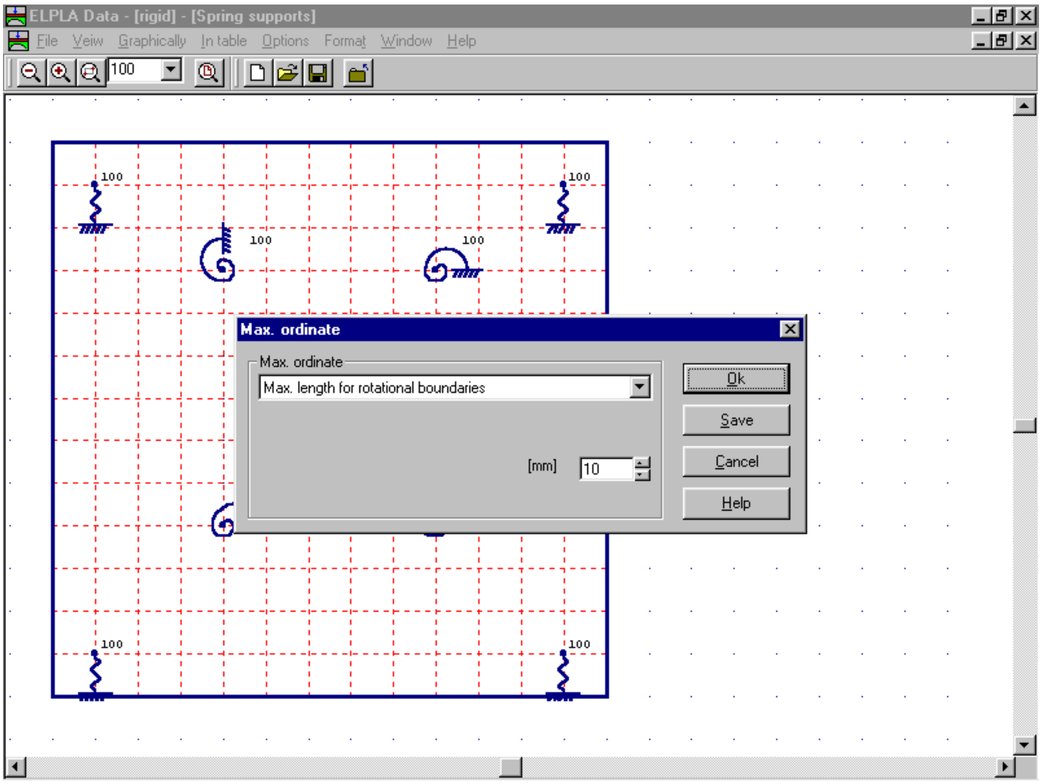


Figure B-61 Max. ordinate

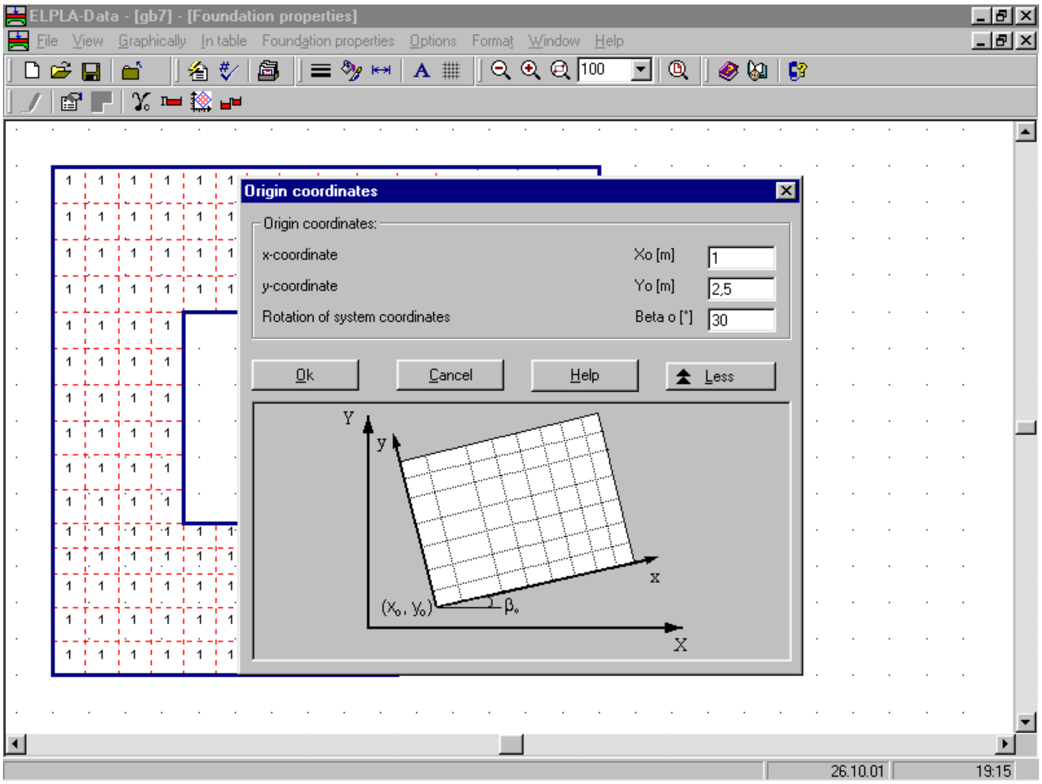


Figure B-62 Menu with explanation figure

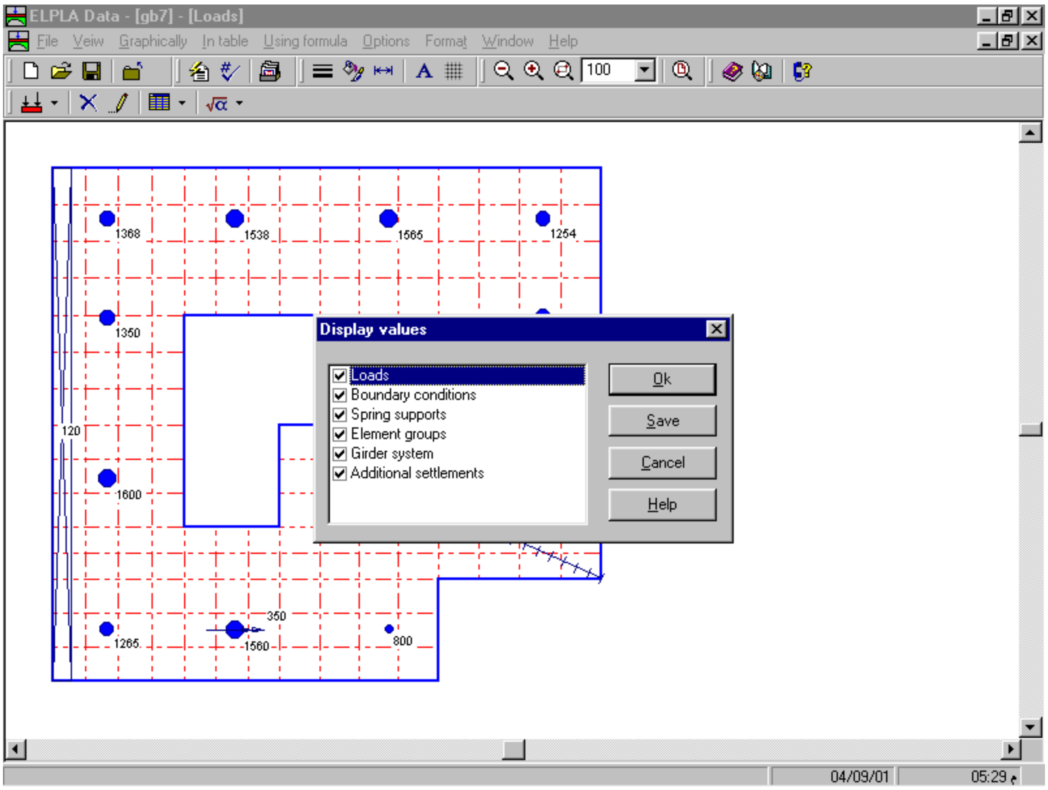


Figure B-63 Displaying values

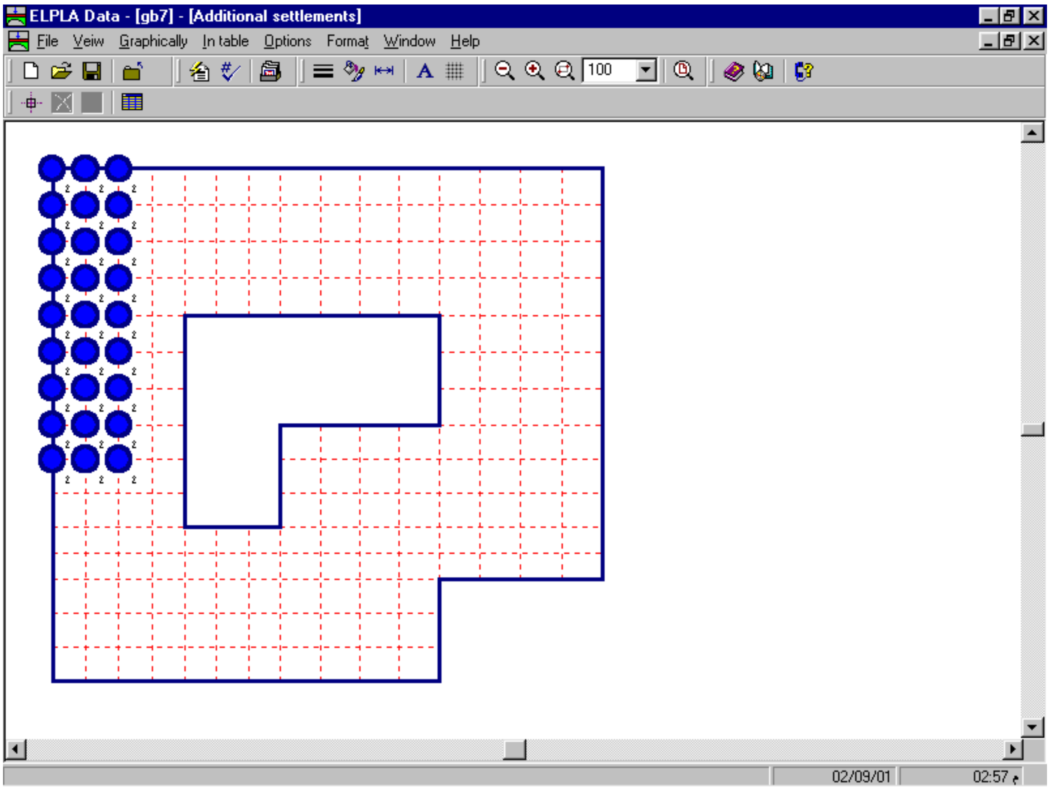


Figure B-64 Defining additional settlements

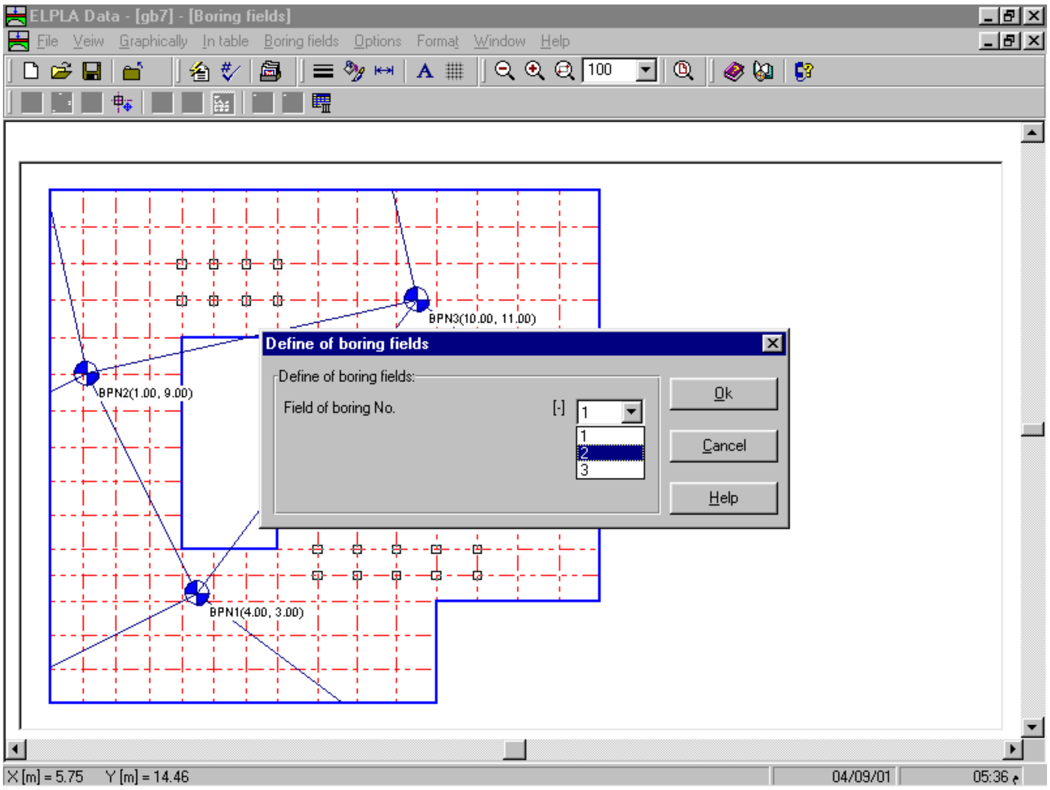


Figure B-65 Defining boring fields

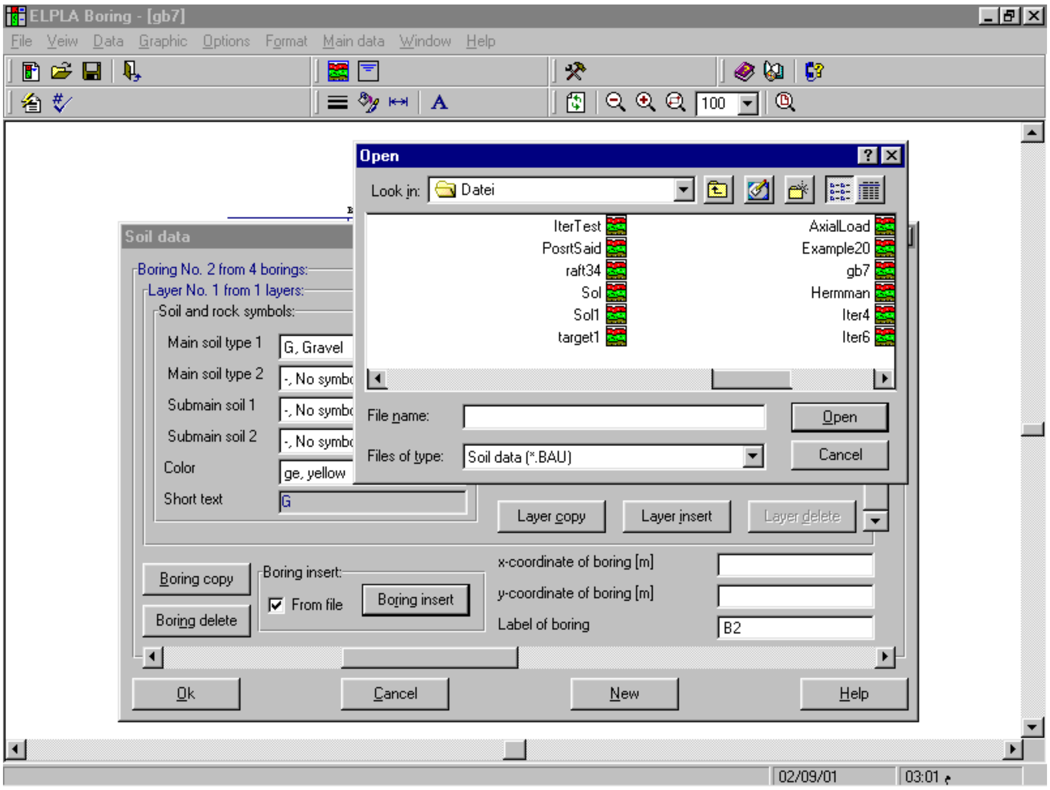


Figure B-66 Inserting the boring log from file

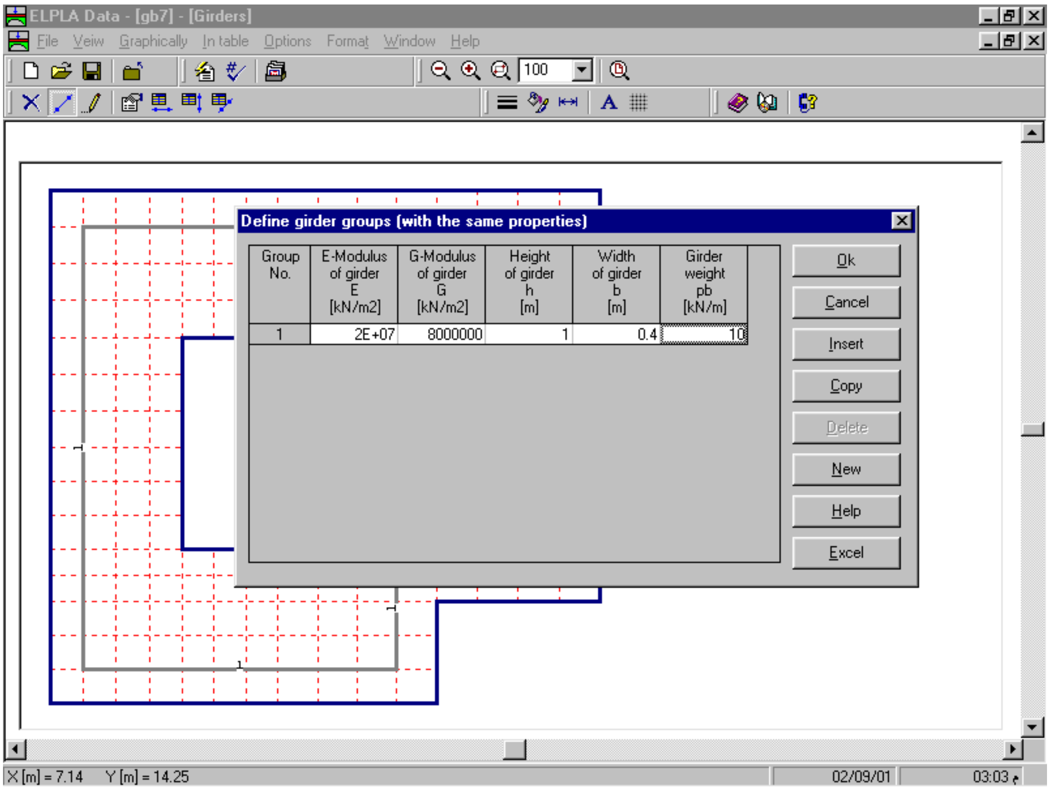


Figure B-67 Defining the girder section by width and height

- The stiffness of girders those have T/ L-section can be simulated on the slab by using additional beam elements. The stiffness of the girder can be obtained through a replacement beam arranged in the center plan of the slab. The dimensions of the replacement beam can be determined as in DIN 1075 or EC 2, Figure B-68

- When defining a set of data (for example FE-Net), another data with this set can be displayed - for example FE-Net with loads and boring locations, Figure B-69

- The date can be defined from the computer calendar, Figure B-70

- When all data sets are defined and saved, changes are carried out to one set, the data that are affected by these changes can be automatically fixed (for example the data those are set outside the FE-Net), Figure B-71

- It is possible to import or export data to MS Excel, Figure B-72

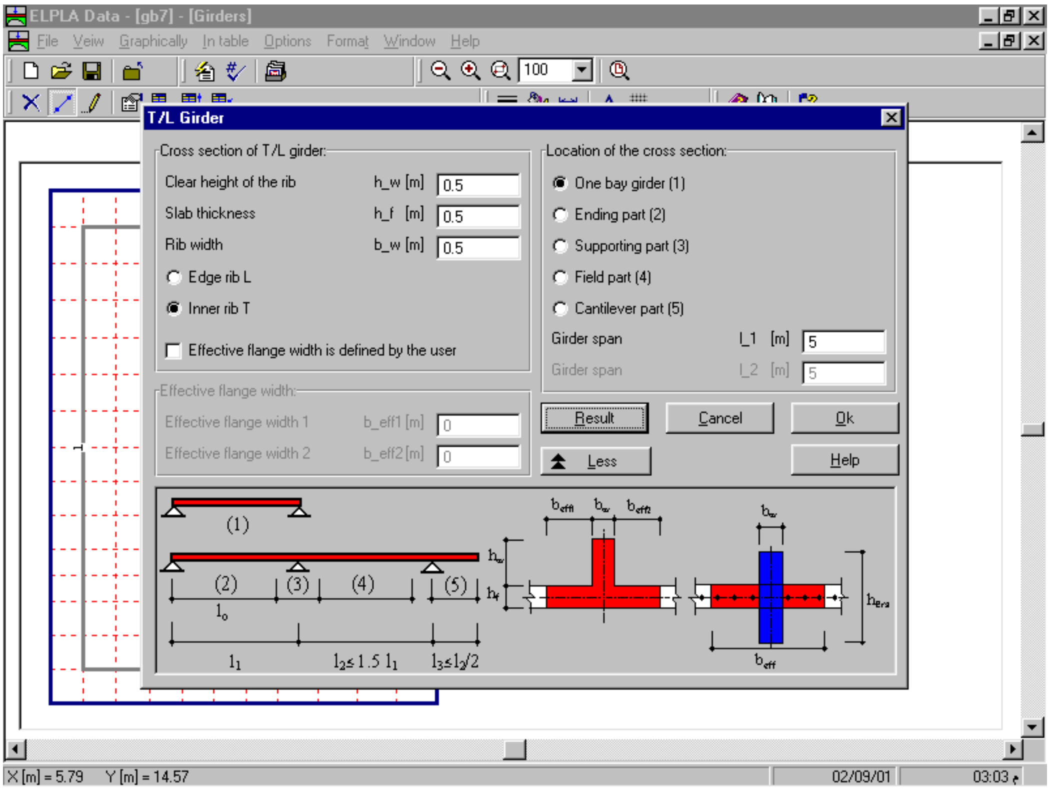


Figure B-68 Defining the girder of T/L-section

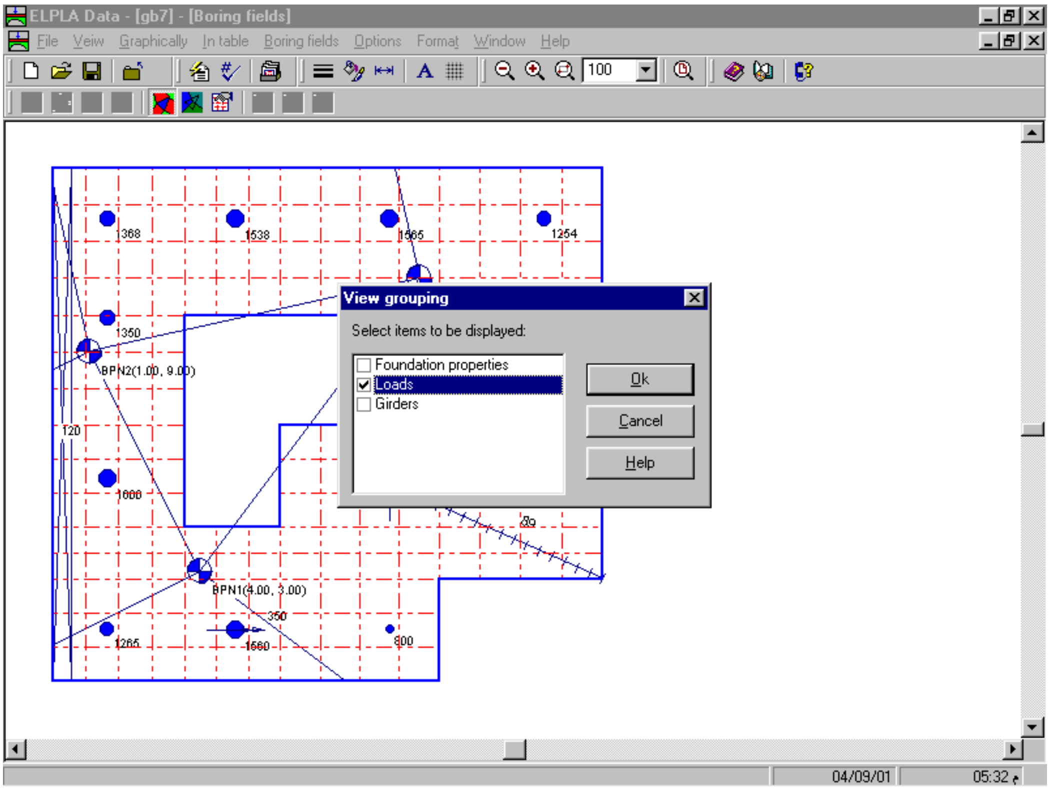


Figure B-69 View grouping data

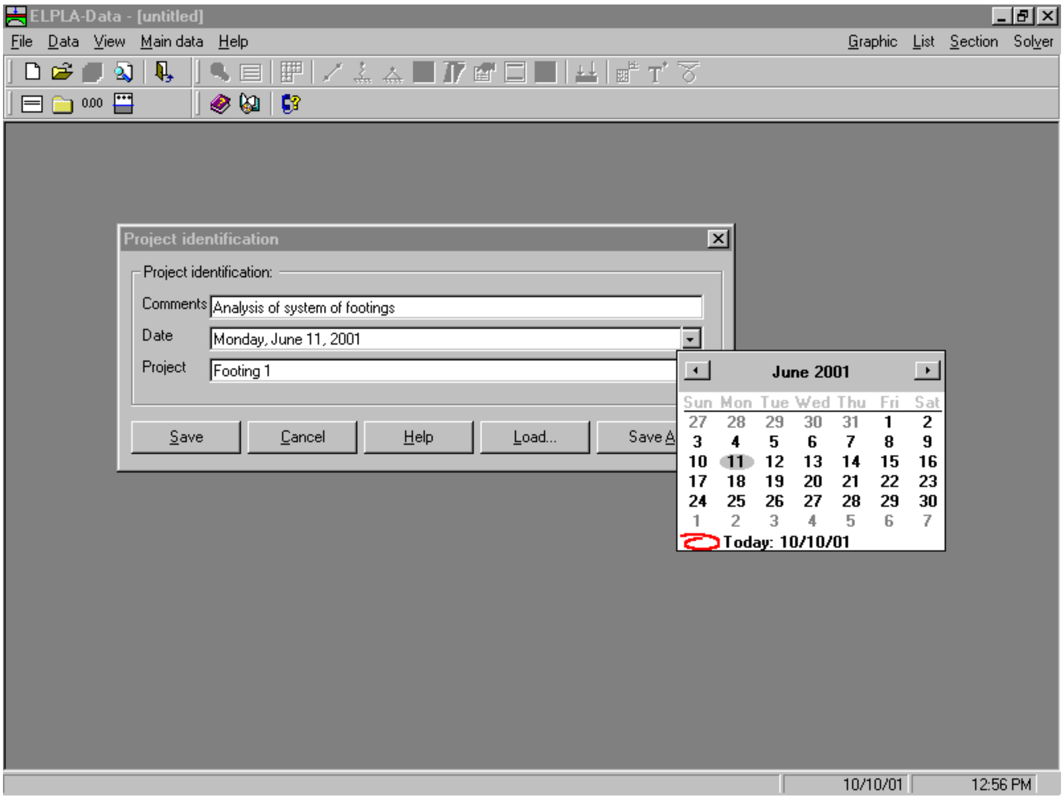


Figure B-70 Defining the date from computer calendar

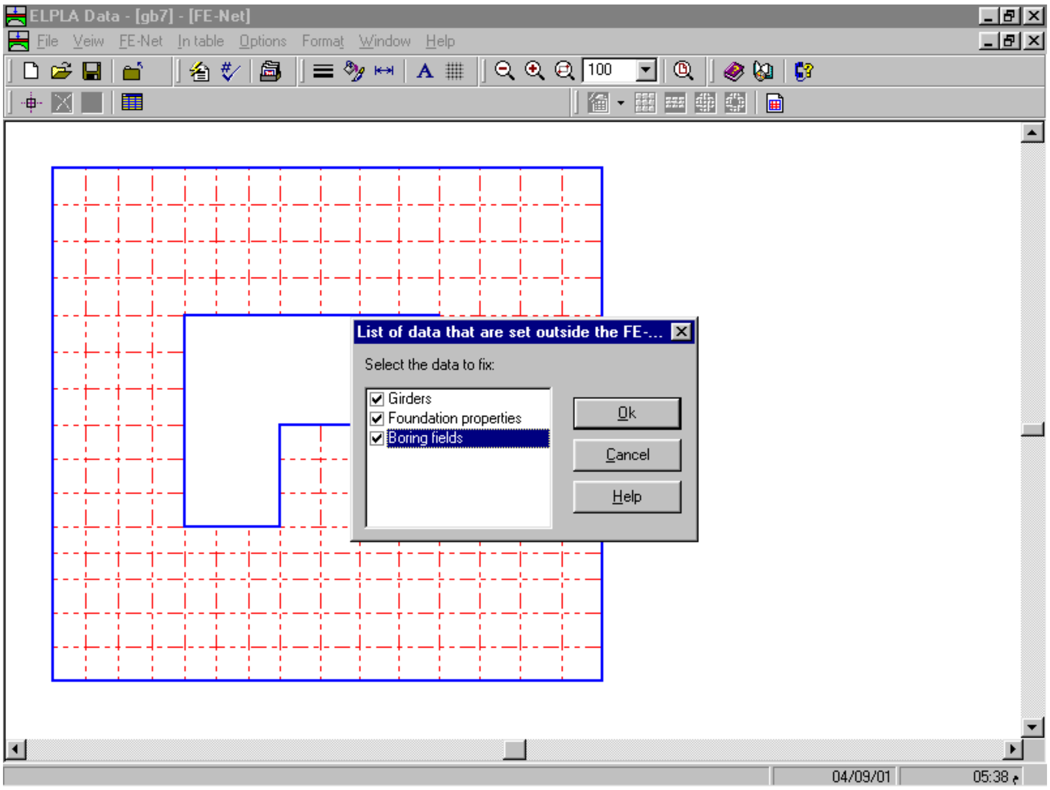


Figure B-71 Fixing data outside FE-Net

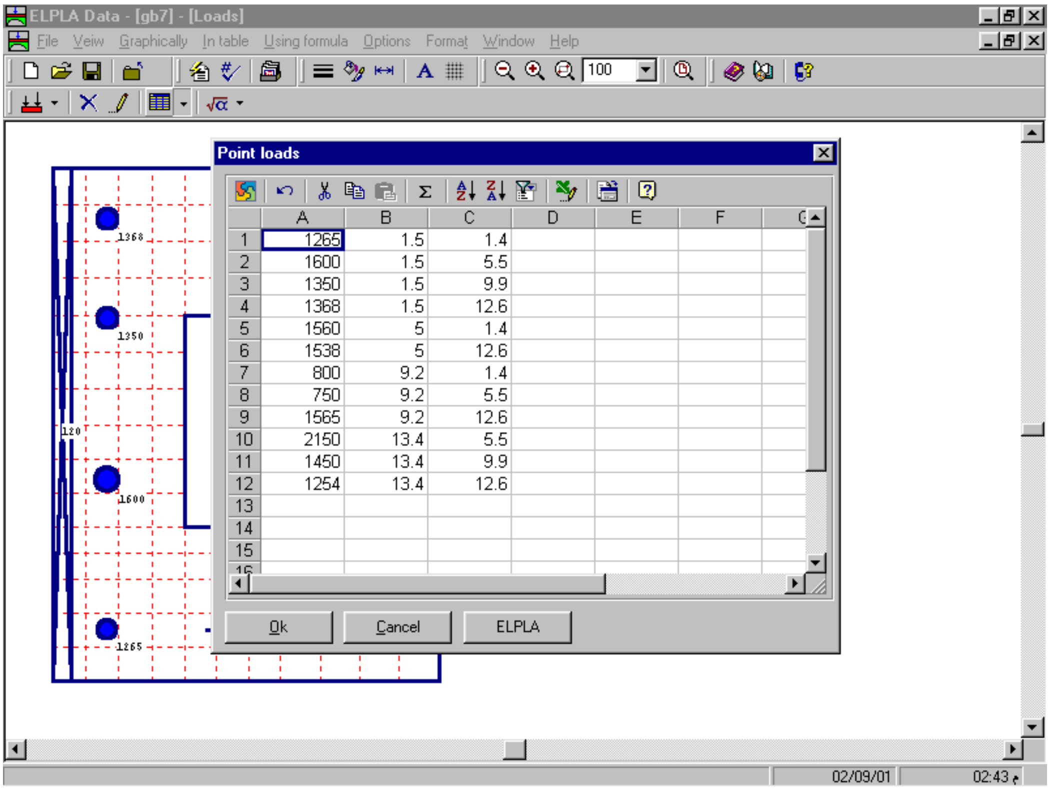


Figure B-72 Exporting and importing data to MS Excel

4.5 File list

- Not only all project data can be deleted but also intermediate results or final results can be separately deleted, Figure B-73

- Deleted files by ELPLA are sent to the recycle pin, Figure B-74

- It is possible to sort ELPLA-files according to project identification data (file name, description, date and project), Figure B-75

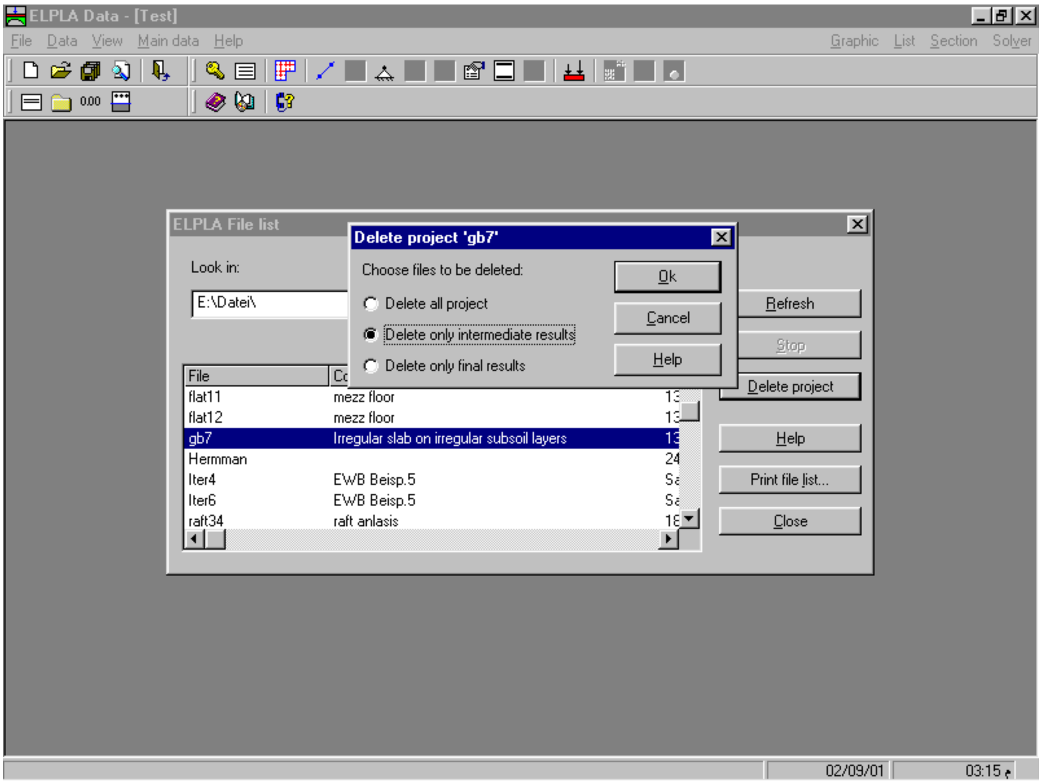


Figure B-73 Deleting intermediate and finial results

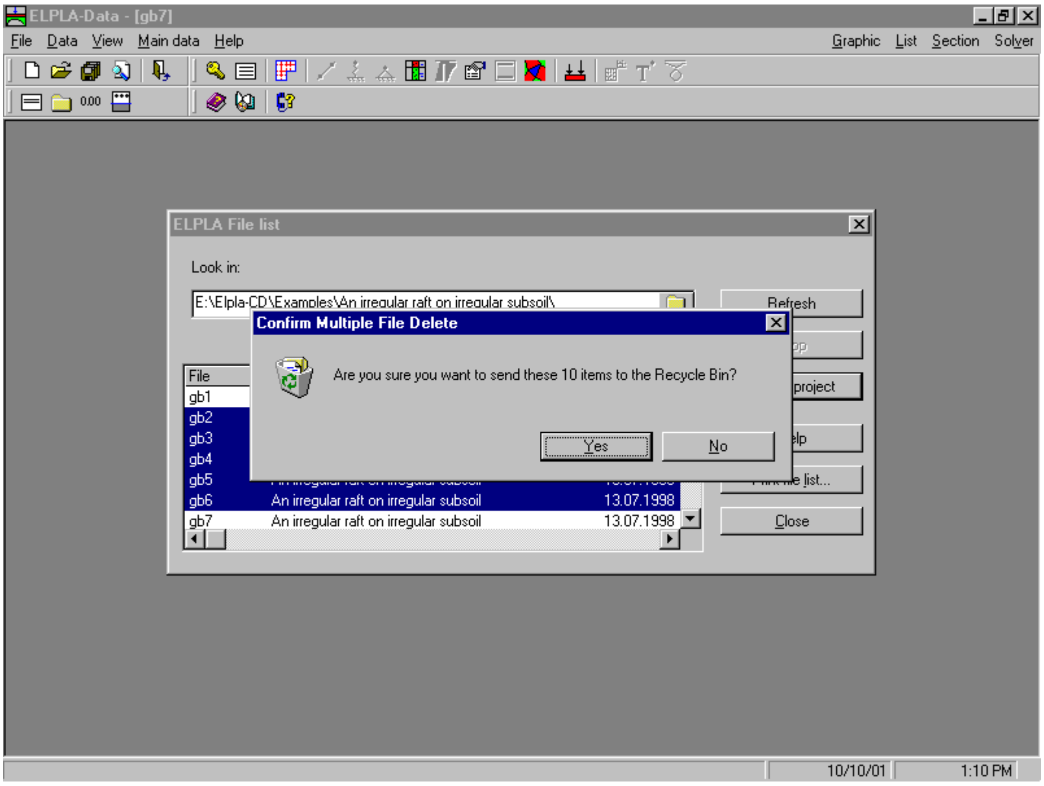


Figure B-74 Sending deleted files to recycle pin

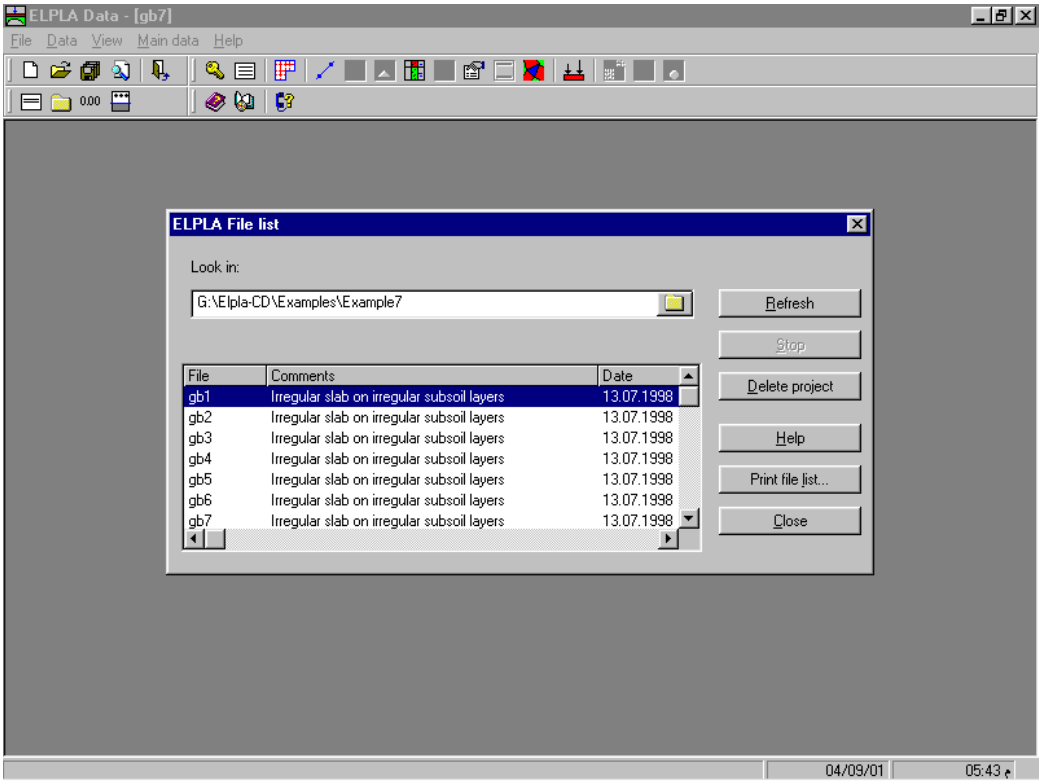


Figure B-75 Sorting ELPLA-files

4.6 Templates

- Templates for different types of FE-Net are available, Figure B-76

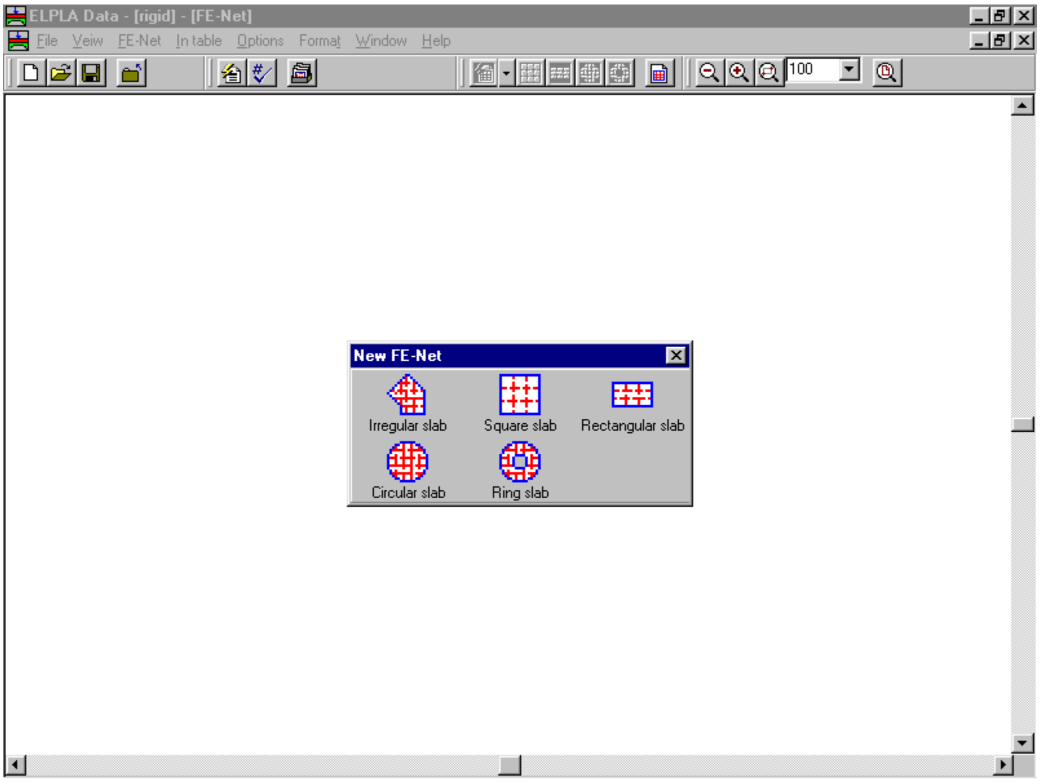


Figure B-76 Templates for different types of FE-Net

4.7 Graphic

- The limit depth can be drawn, if desired, beside its corresponding boring log, Figure B-77

- When drawing a group of boring logs or limit depths in one presentation, the user has the possibility to choose the sequence of boring logs or limit depths (for example boring 6 at the left, boring 1 at the right next to it, boring 3 at the right next to boring 1 etc.). So the user can draw subsoil sections, Figure B-78

- The intensity of numbers on the contour lines can be decided through a writing factor, Figure B-79

- Legend for the maximum ordinate is inserted for results in isometric view, distribution of results in the plan, results as circular diagrams, deformation and principal moments as streaks, Figure B-80

- When studying the effect of neighbor foundations, all foundations can be drawn with data or results in one representation, Figure B-81

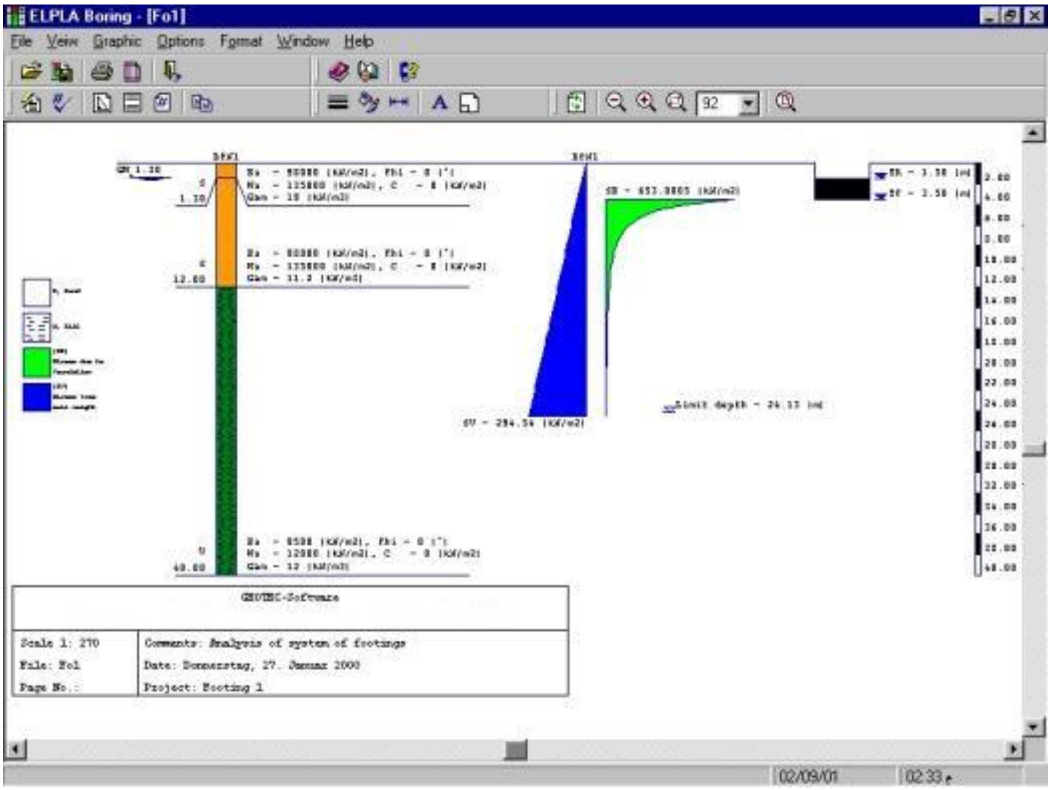


Figure B-77 Limit depth beside boring log

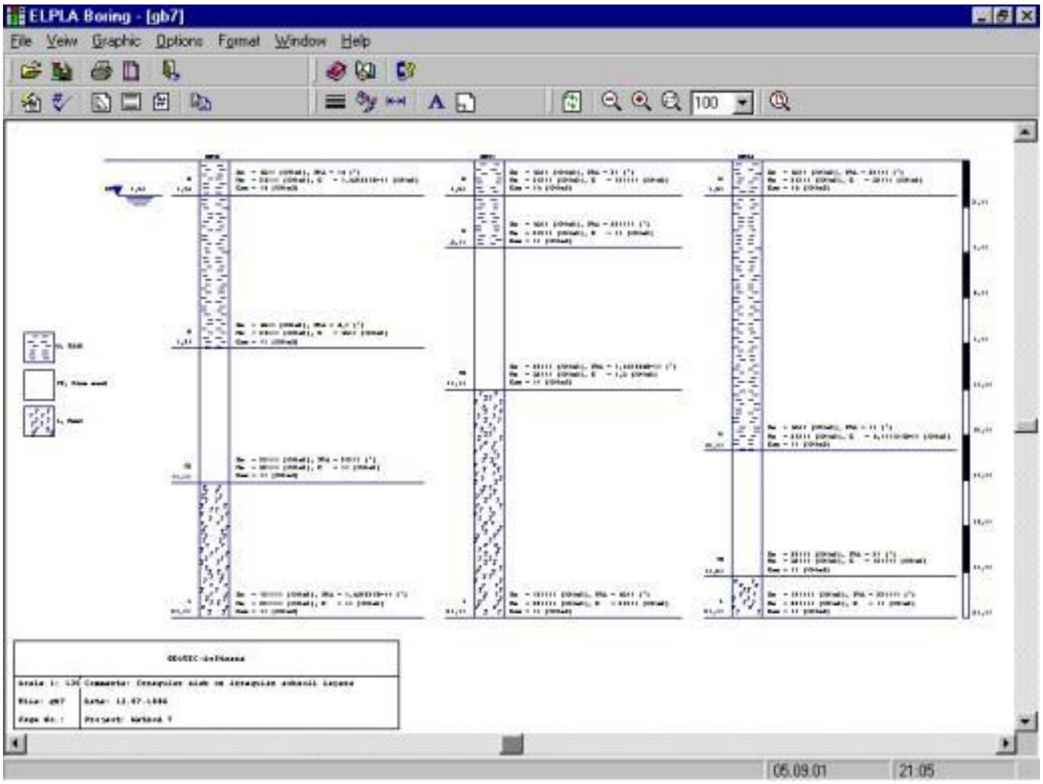


Figure B-78 Choosing the sequence of boring logs in the drawing

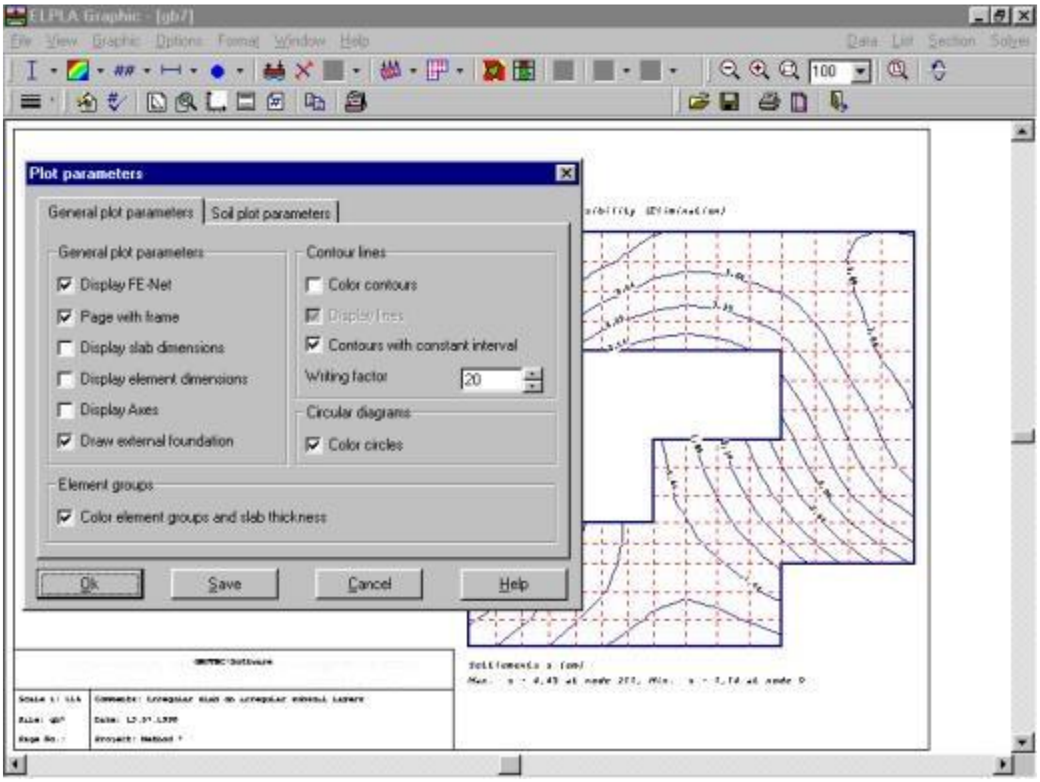


Figure B-79 Writing factor for contour lines

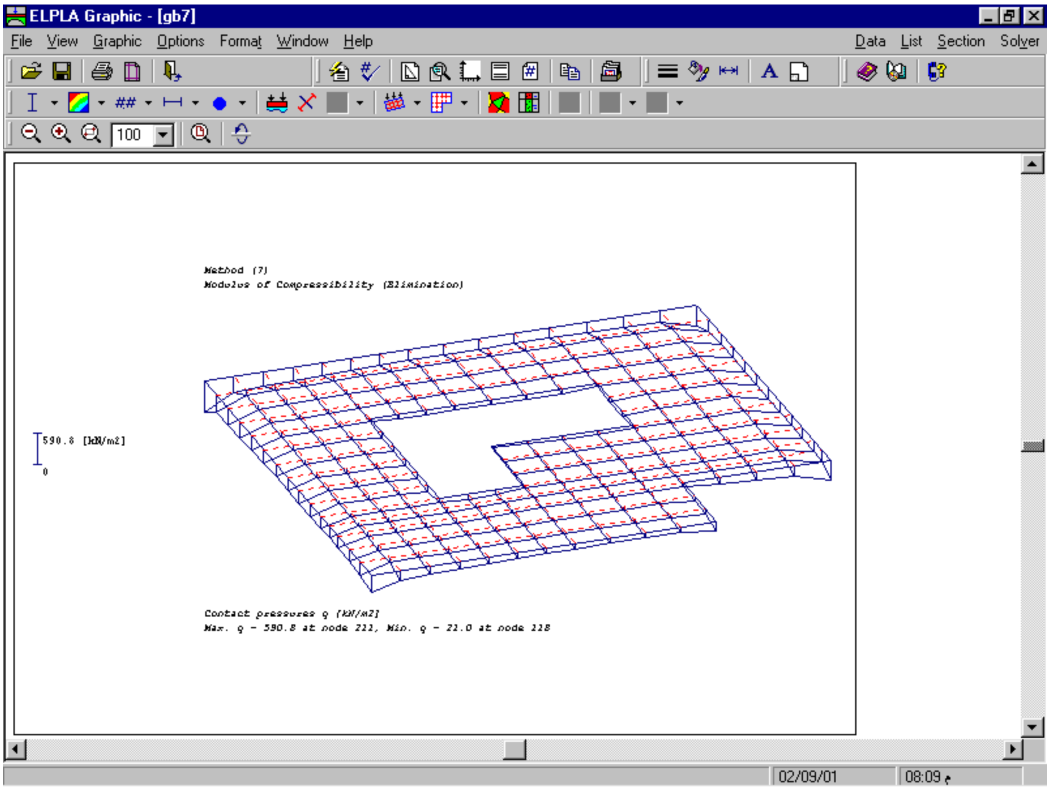


Figure B-80 Legend for maximum ordinate

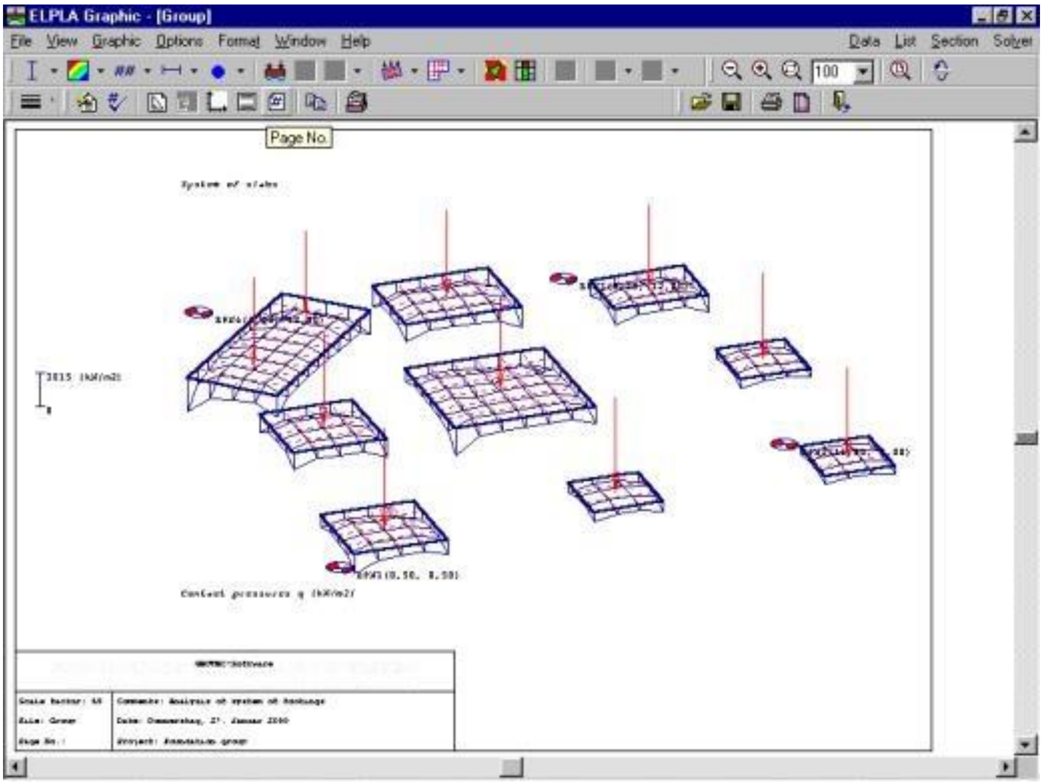


Figure B-81 Drawing a group of foundations with data

- You can draw a group of data with results or a group of data together in one presentation (for example contour lines of settlements with loads and slab thickness, or loads with boring locations), Figure B-82

- When defining view angles about x-, y- and z-axis for a drawing in three-dimensional, the drawing is previewed on a small menu before displaying it on the screen, Figure B-83

- New expressed symbols for boundary conditions and supports are presented, Figure B-84

- If the system is symmetrical, the symbols of symmetry will be automatically drawn, Figure B-85

- The graphical drawing can be copied in Metafile-Format to Clipboard and then inserted directly to other Windows-programs, Figure B-86

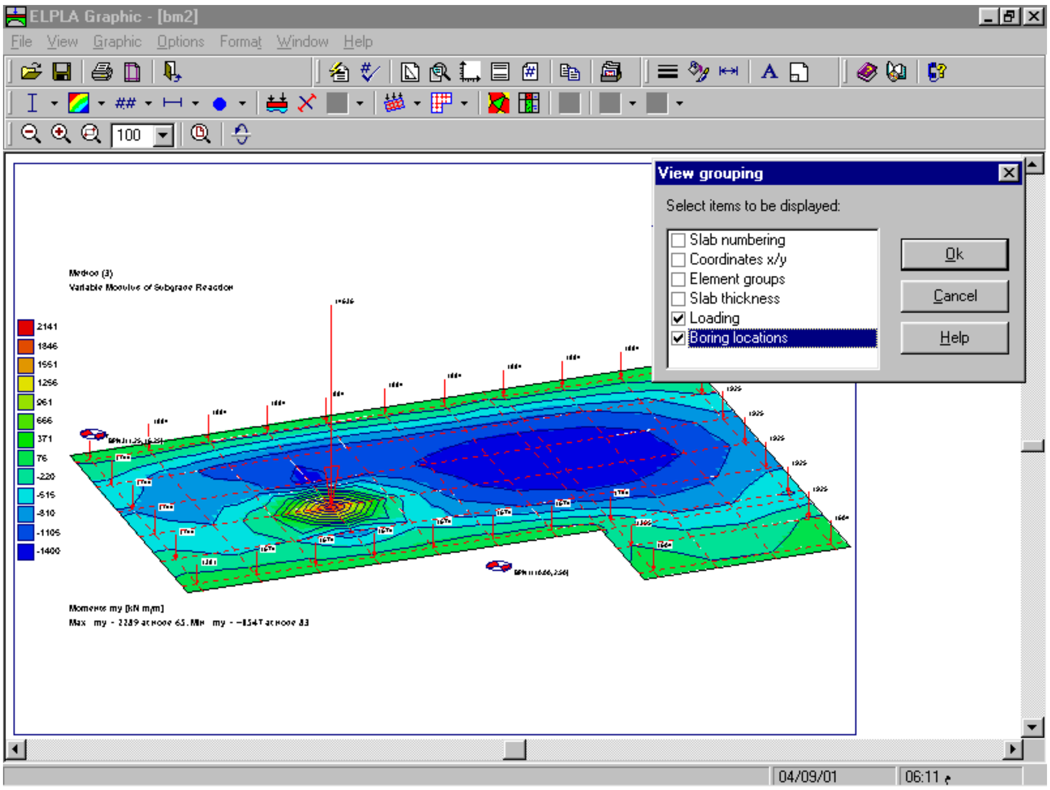


Figure B-82 Drawing data and results in one presentation

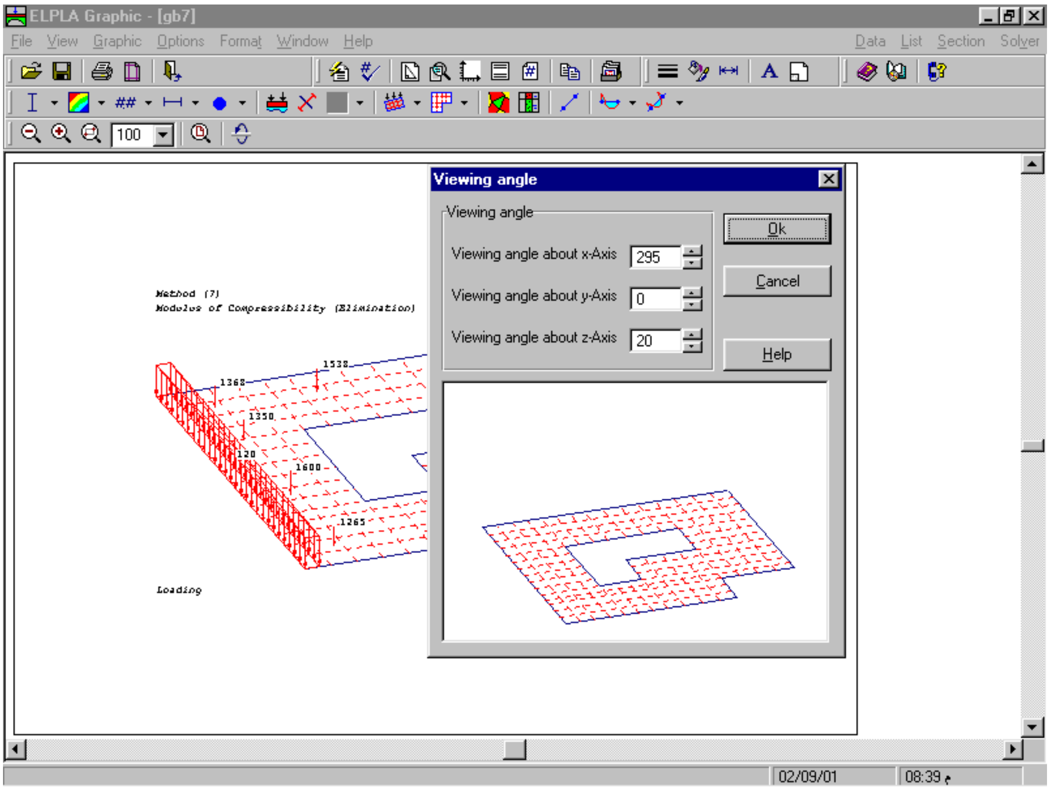


Figure B-83 Viewing angle

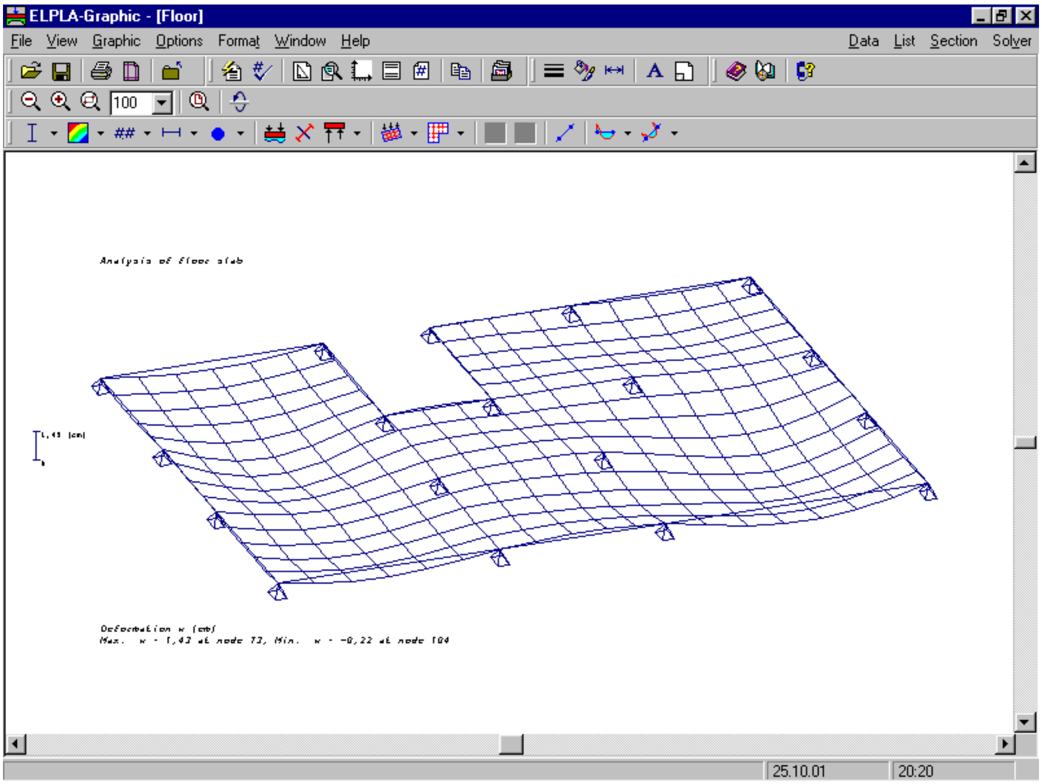


Figure B-84 New expressed symbols for boundary conditions

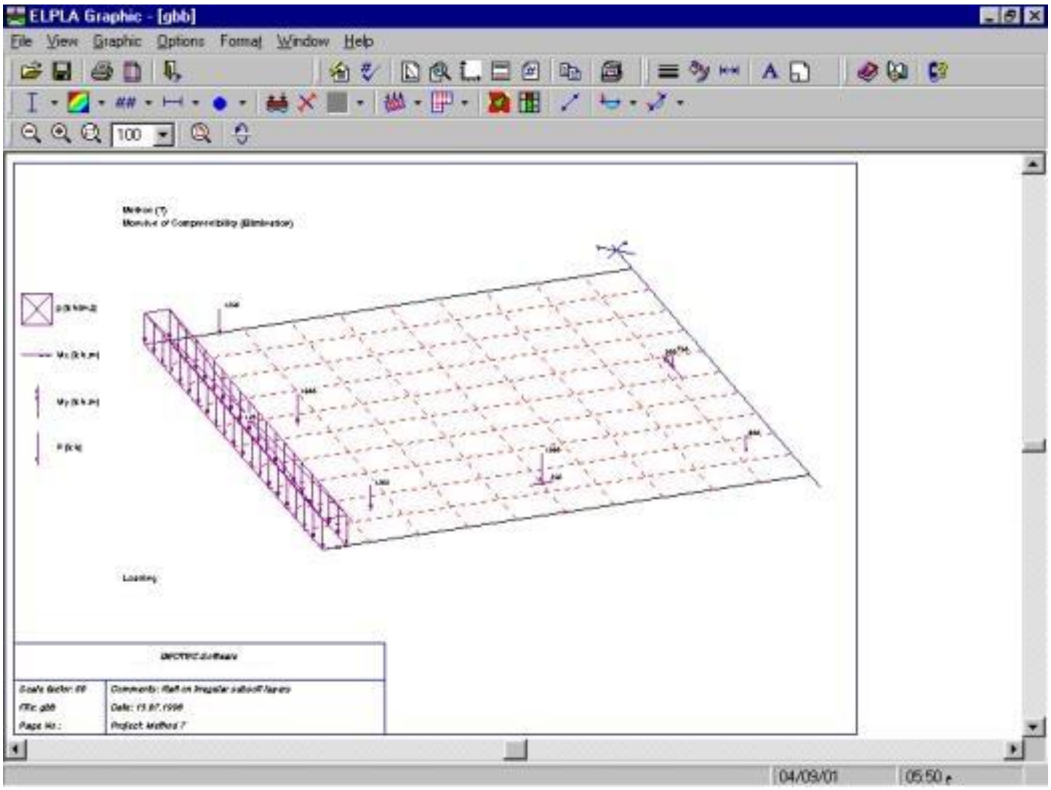


Figure B-85 Drawing symbols of symmetry

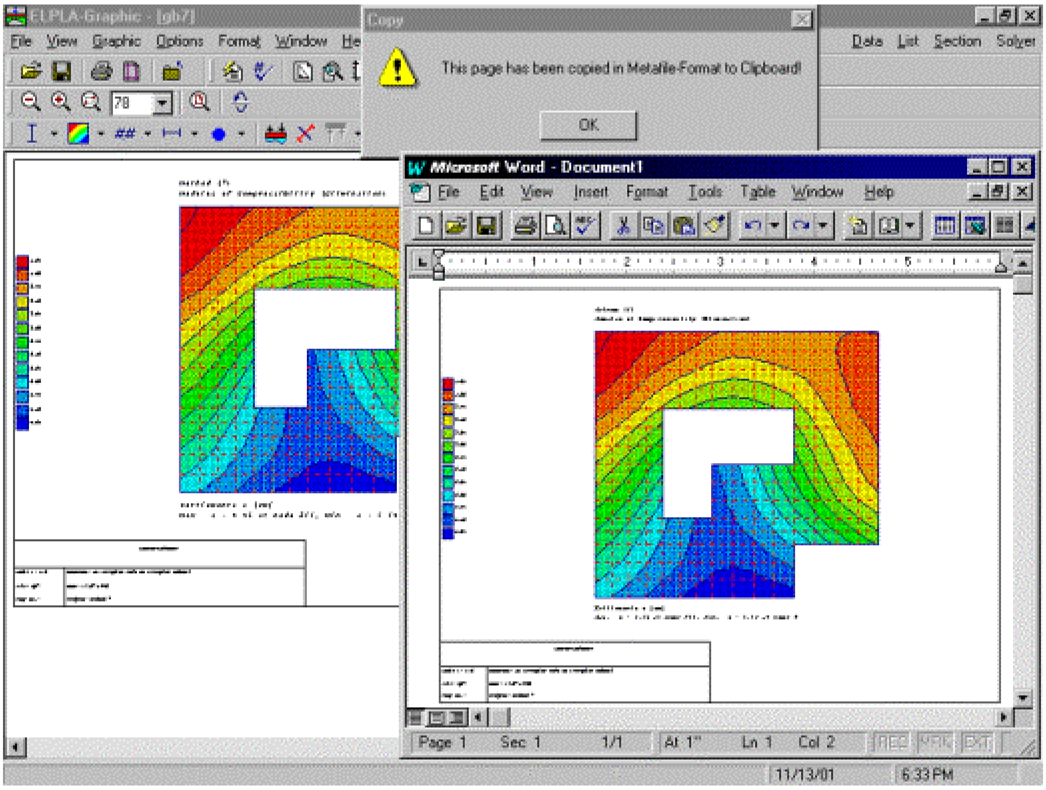


Figure B-86 Copying the drawing to clipboard

4.8 List

- It is possible to export results to MS Excel, Figure B-87

- It is possible to export results to MS Word, Figure B-88

- A new user interface for the program ELPLA-List is developed. The user can deal with different results in different windows at the same time, Figure B-89

- With project Explorer it is possible to swap between data and results easily, Figure B-90

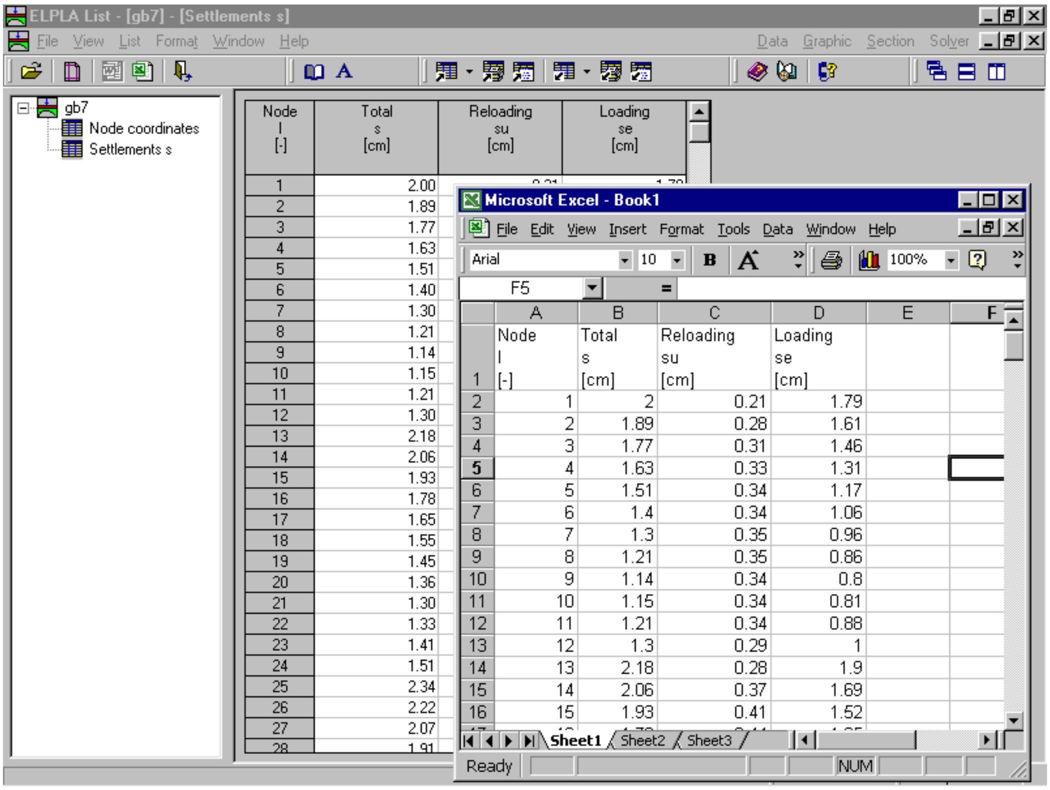


Figure B-87 Export results to MS Excel

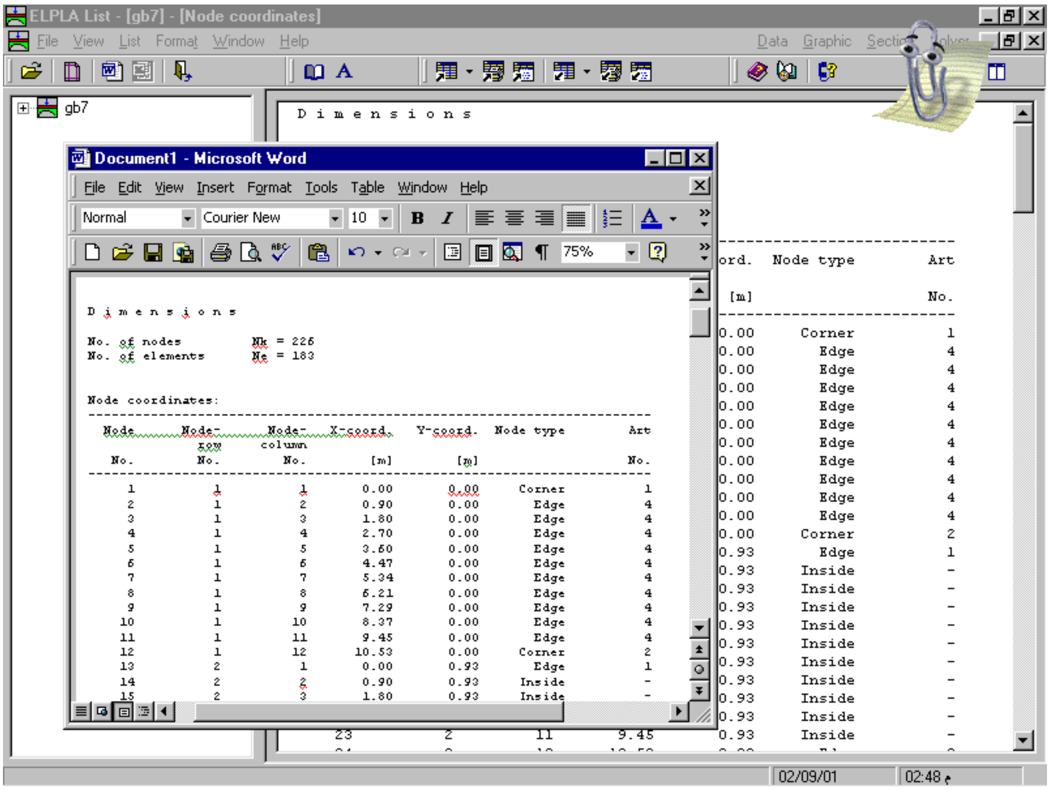


Figure B-88 Export results to MS Word

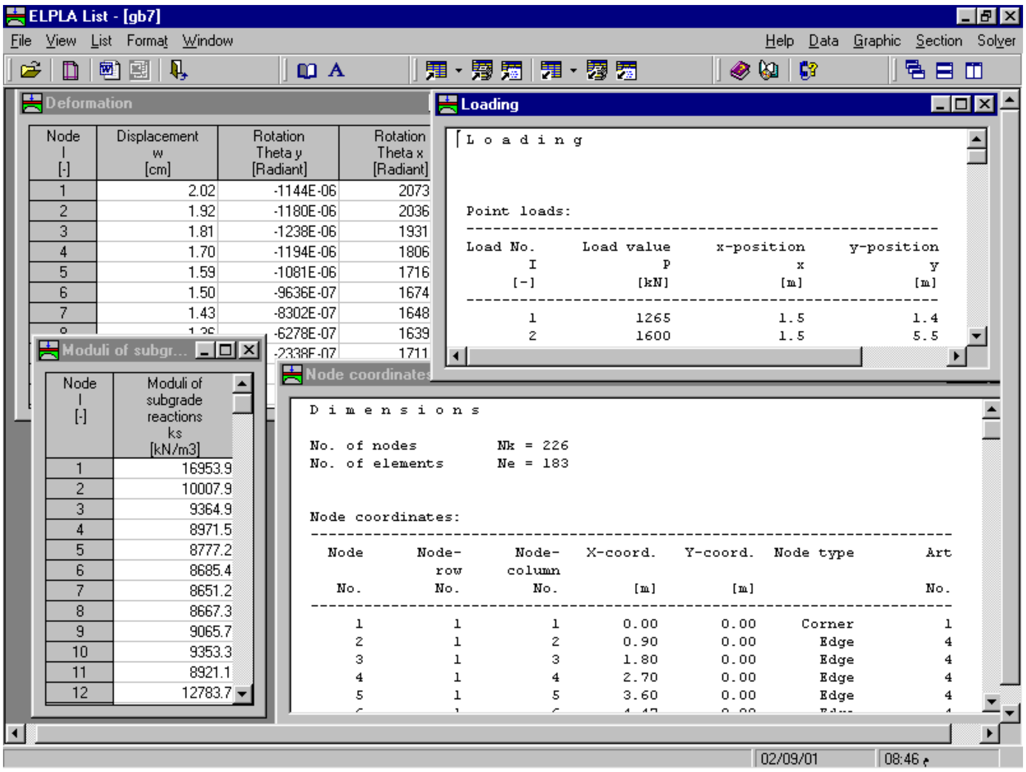
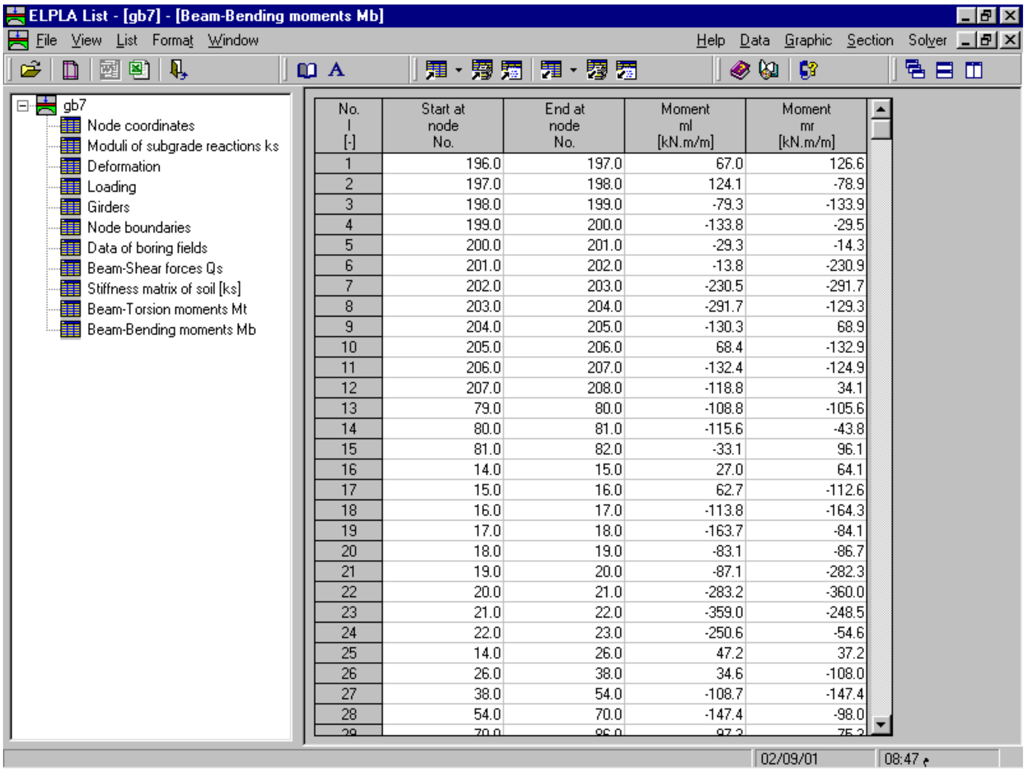


Figure B-89 Many windows of data and results in ELPLA-List



No. I [-]	Start at node No.	End at node No.	Moment ml [kN.m/m]	Moment mr [kN.m/m]
1	196.0	197.0	67.0	126.6
2	197.0	198.0	124.1	-78.9
3	198.0	199.0	-79.3	-133.9
4	199.0	200.0	-133.8	-29.5
5	200.0	201.0	-29.3	-14.3
6	201.0	202.0	-13.8	-230.9
7	202.0	203.0	-230.5	-291.7
8	203.0	204.0	-291.7	-129.3
9	204.0	205.0	-130.3	68.9
10	205.0	206.0	68.4	-132.9
11	206.0	207.0	-132.4	-124.9
12	207.0	208.0	-118.8	34.1
13	79.0	80.0	-108.8	-105.6
14	80.0	81.0	-115.6	-43.8
15	81.0	82.0	-33.1	96.1
16	14.0	15.0	27.0	64.1
17	15.0	16.0	62.7	-112.6
18	16.0	17.0	-113.8	-164.3
19	17.0	18.0	-163.7	-84.1
20	18.0	19.0	-83.1	-86.7
21	19.0	20.0	-87.1	-282.3
22	20.0	21.0	-283.2	-360.0
23	21.0	22.0	-359.0	-248.5
24	22.0	23.0	-250.6	-54.6
25	14.0	26.0	47.2	37.2
26	26.0	38.0	34.6	-108.0
27	38.0	54.0	-108.7	-147.4
28	54.0	70.0	-147.4	-98.0

Figure B-90 Swapping between data and results

4.9 Languages

ELPLA is available in three languages: English, German and Arabic. An exchange of data for the three versions is possible with full compatibility .

4.10 Boring log

The new separate program ELPLA-Boring is developed. By the program the user can define boring logs graphically and draw limit depths, Figure B-94.

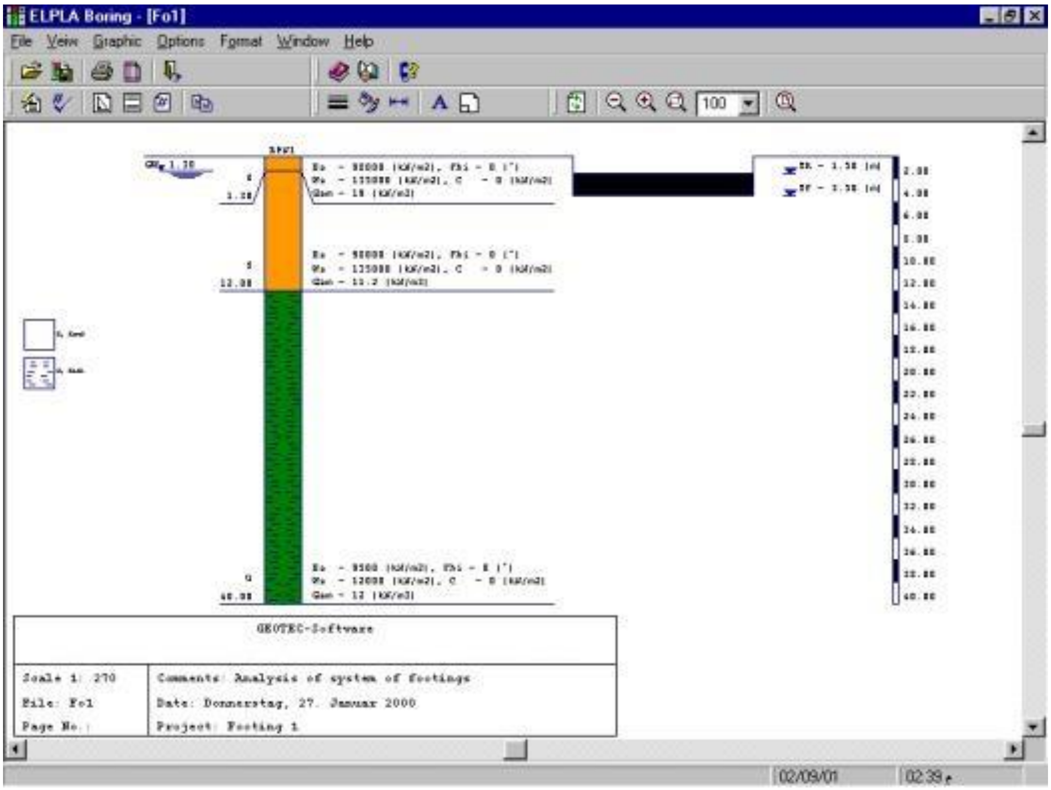


Figure B-94 ELPLA-Boring

4.11 ELPLA-Editor

A new separate program ELPLA-Editor is developed. The program is a simple editor program and can deal with text files that are created by ELPLA, Figure B-95.

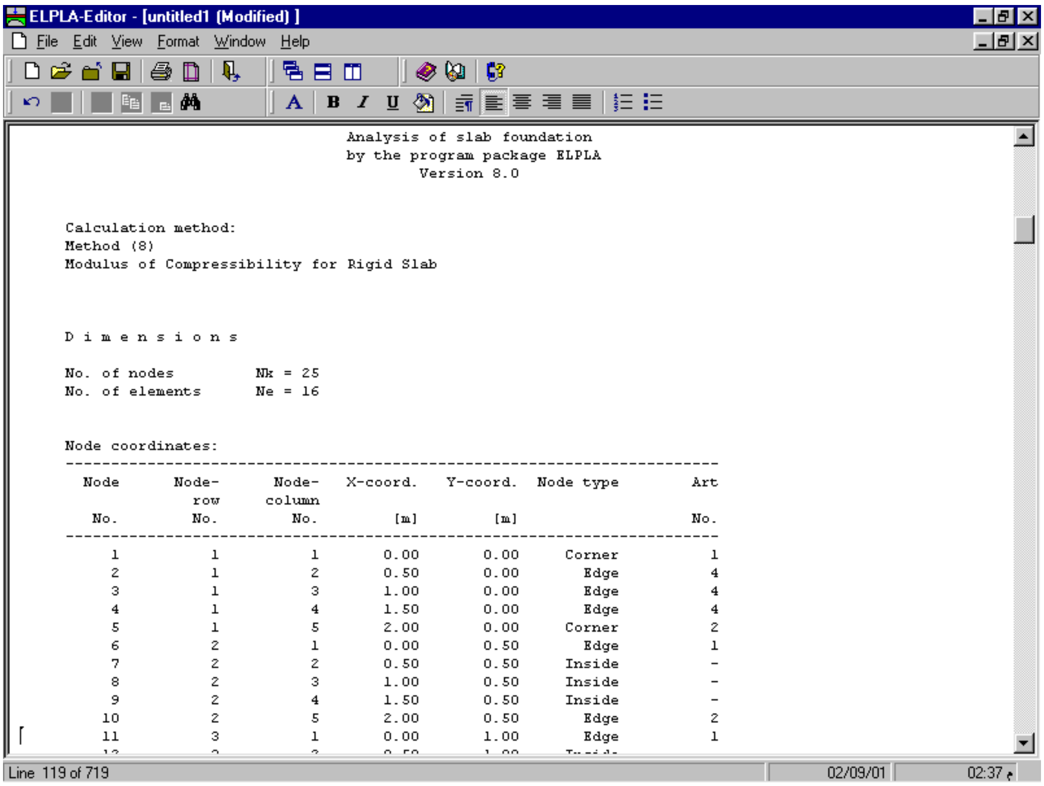


Figure B-95 ELPLA-Editor

4.12 Group functions

- The 7 separate programs ELPLA-Data, ELPLA-Solver, ELPLA-Graphic, ELPLA-Section, ELPLA-List, ELPLA-Boring and ELPLA-Editor can run independently of each other or as a group, Figure B-96

- All program menus are represented also by icons (Toolbar), Figure B-97

- Page setup (portrait format, landscape format, printer, ...) is saved separately for each of the programs ELPLA-Graphic, ELPLA-List, ELPLA-Section, ELPLA-Boring and ELPLA-Editor, Figure B-98

- The user can connect directly through ELPLA with ELPLA-Website in Internet, Figure B-99

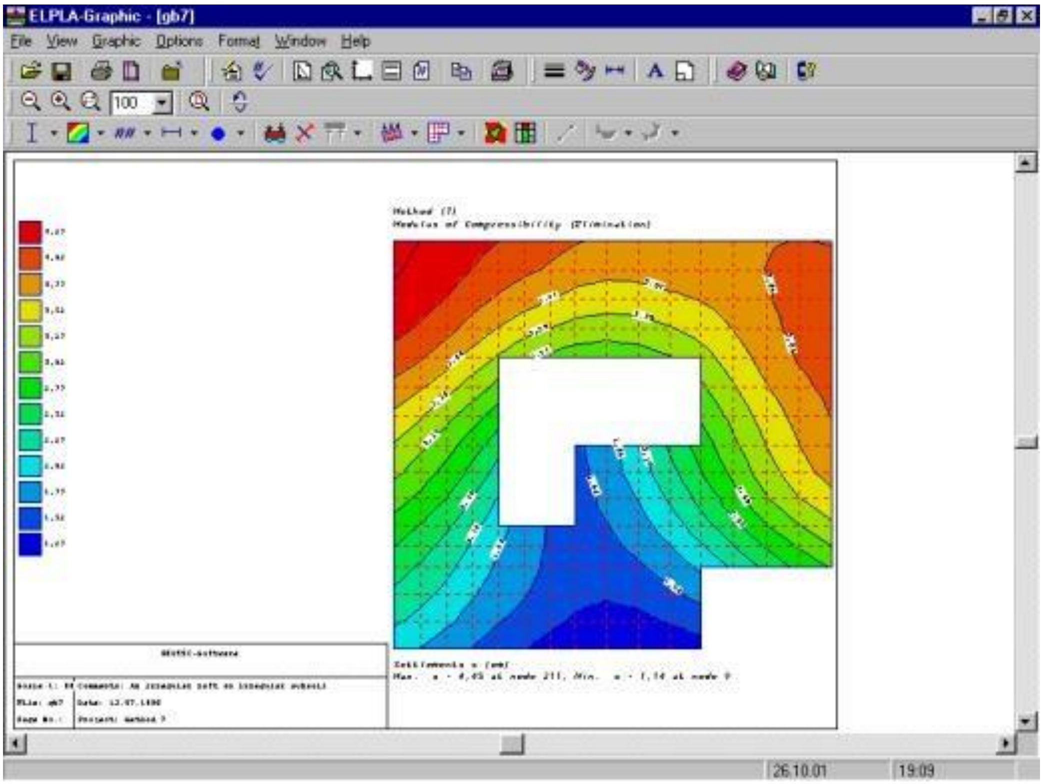


Figure B-96 ELPLA-Graphic alone

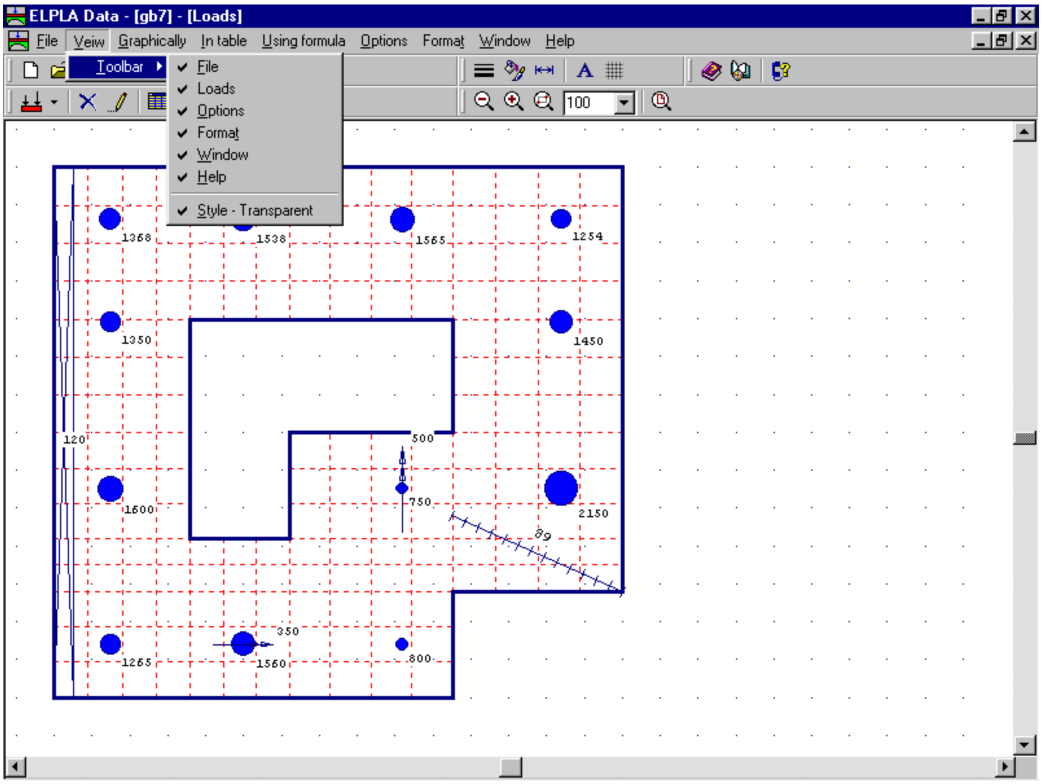


Figure B-97 Menu with Toolbars and icons

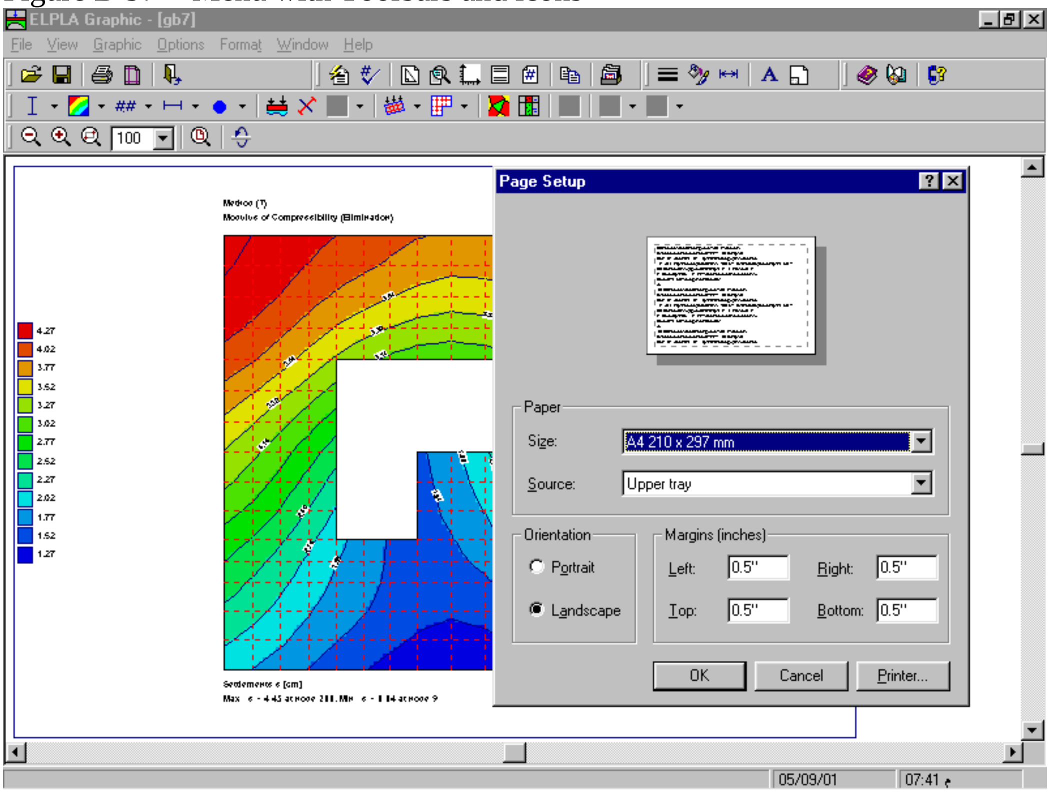


Figure B-98 Page setup for each program

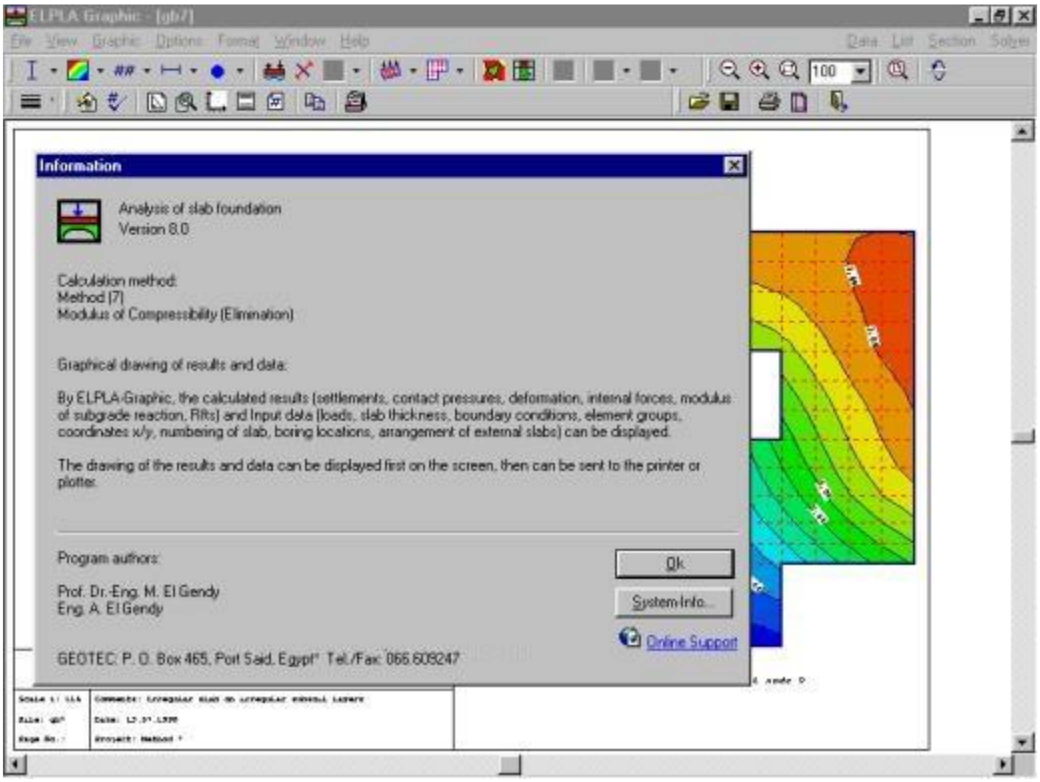


Figure B-99 The user can connect to internet through ELPLA

- The computer knows the files of the ELPLA-project automatically. Running the program can be carried out by clicking on the icons of the files *.PO1, *.PO2 or *.BAU, Figure B-100

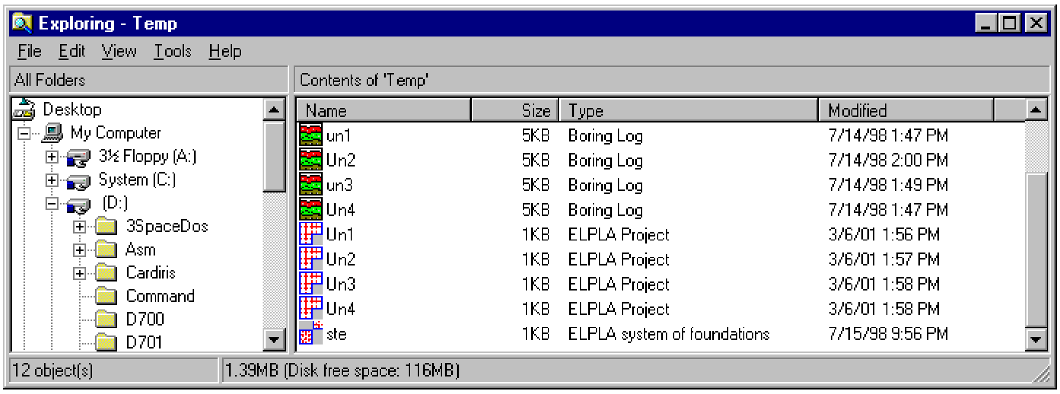


Figure B-100 Running ELPLA by clicking on project icon

4.13 Help file

- The help file in HTML-Format contains the text of the user's guide, Figure B-101

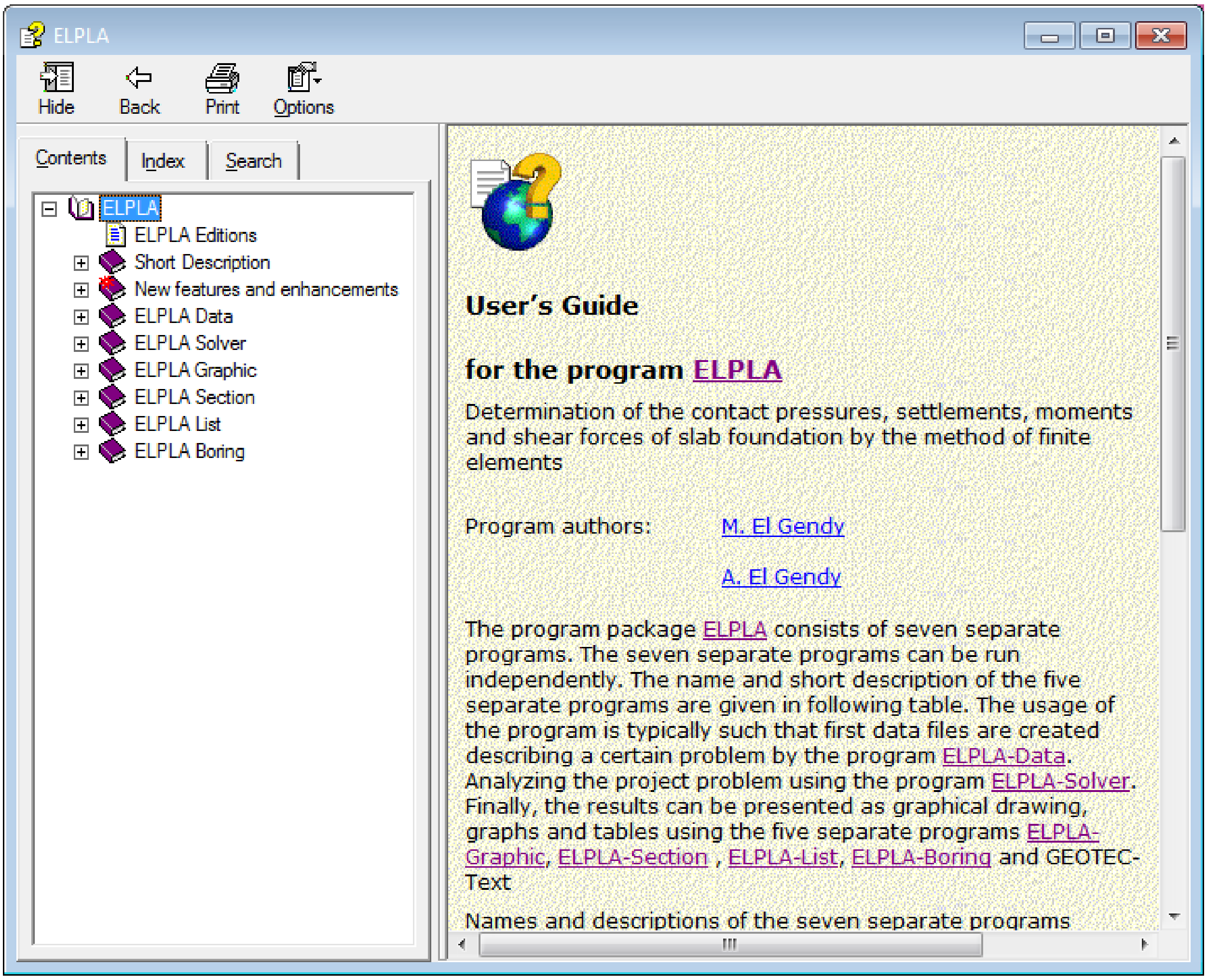


Figure B-101 Help file in HTML-Format

ELPLA Tools

Self-Adaptive Mesh Wizard

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

Web site: http://www.elpla.com
e-mail: geotec@elpla.com

Table of Contents

1 Self-Adaptive Mesh Wizard

The Self-Adaptive Mesh Wizard is a tool that helps the user for generating finite element mesh with better element/ node distributions. The wizard is used to regenerate the finite element mesh for slab floor and raft problems. The new mesh will be generated based on the strain energy distribution. The self-adaptive technique used in this wizard reduces general system error besides enhances stress distributions. Adaptive mesh is generated either by using Delaunay generation or by using Grid-based generation (Figure 1).

Although the Self-Adaptive Mesh Wizard works with ELPLA Limit Edition, it is recommended to be used with ELPLA professional edition because the adapted mesh resulted from this wizard always has more than 300 nodes.

Note:
This wizard does not analyze any project. The project must be analyzed before running this wizard. Also the adapted project must be analyzed again after running the wizard.

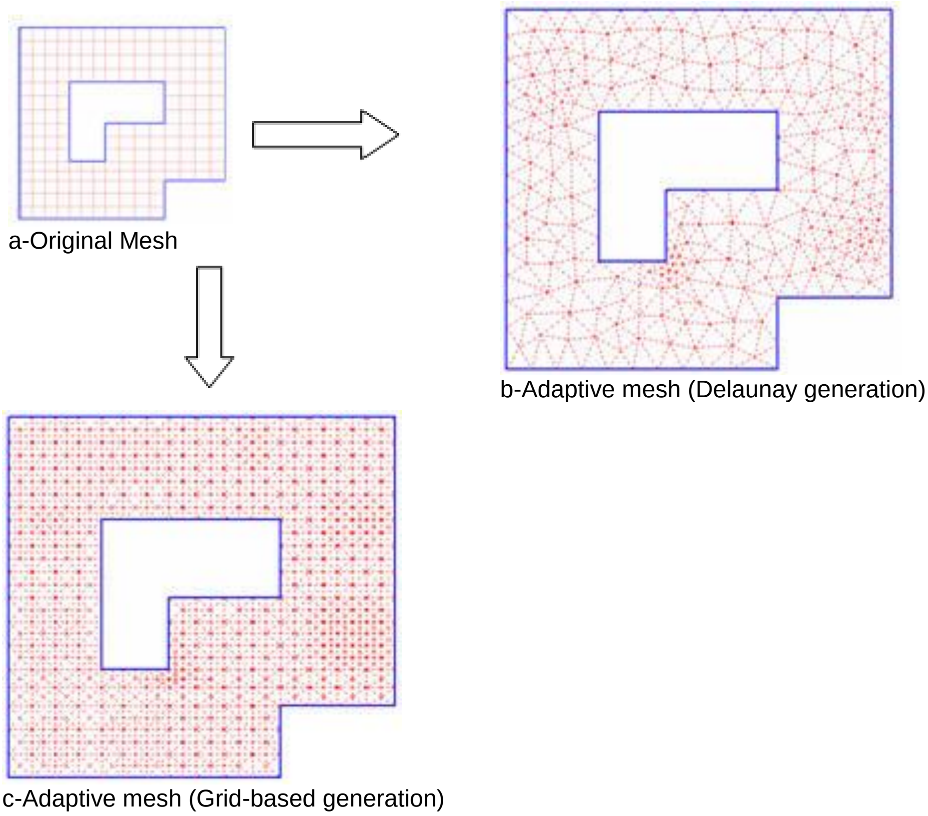


Figure 1 Two types of adaptive mesh generation: Delaunay generation and Grid-based generation

Self-Adaptive Mesh Wizard

Figure 2 shows the first form in the wizard which contains the following options:

Open Project

Displays the name and path of the selected project. You may type in the name and path of a project or choose a project using the "Browse" button.

Browse

Displays a dialog box you can use to browse to a project.

Recalculate Mesh Boundary

The Self-Adaptive Mesh Wizard can not detect any change made to the mesh boundary after generating mesh. For example, removing node from mesh will produce opening that can not be detected from the original mesh boundary. In this case, the "Recalculate Mesh Boundary" check box must be selected to ignore the original mesh boundary and create a new mesh boundary according to the last mesh modification.

Help

Displays the help topic for this step. Also pressing F1, gives the same action.

Cancel

Cancels previous actions and closes the Application Wizard.

Back

Moves to the previous step.

Next

Moves to the next step.

Finish

Accepts any selections that have been entered and the defaults for the remaining steps, then creates the adaptive mesh.

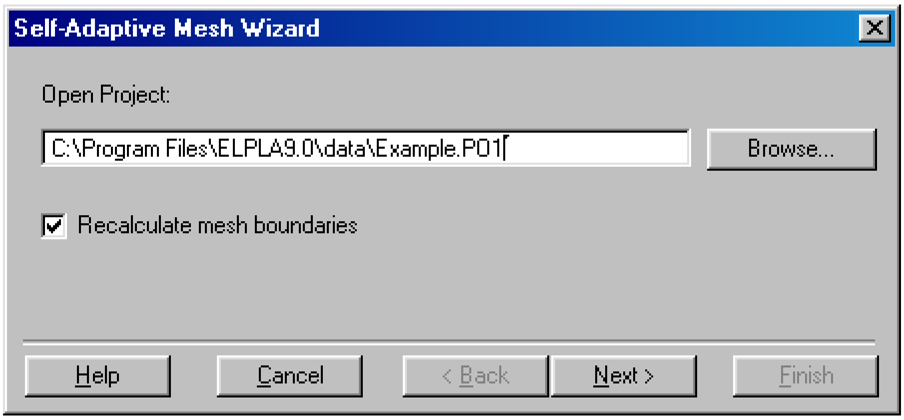


Figure 2 "Project" Form

The next form of the wizard is the "Generation Type" form. There are two types of mesh generation: Delaunay and Grid-based. Mesh optimization options are only available for Delaunay generation.

Smoothing mesh

By the "Smoothing mesh" option, the dimension of FE-Net can be optimized by making all element sides around each node having nearly the same length.

Directing border elements

By the "Directing border elements" option, all elements on the slab borders can be directed and arranged (see ELPLA-Data User's Guide for more details). This command is useful to present contact pressures at raft edges in good manner when analyzing the raft by Continuum model in which the contact pressure at raft edges are higher than that at the middle.

Element dimension parameters defined in the next wizard form depends on the generation type as follow:

Delaunay Generation

In the Delaunay generation (Figure 4), the minimum angle, element circumradius and minimum element circumradius are required to define.

Grid-Based Generation

In the Grid-based generation (Figure 5), the element size (length/width) and minimum element size (length/width) are required to define.

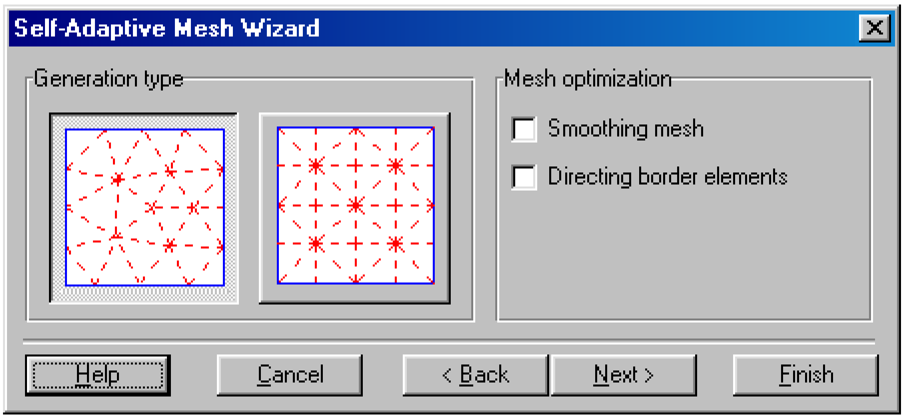


Figure 3 "Generation Type" Form

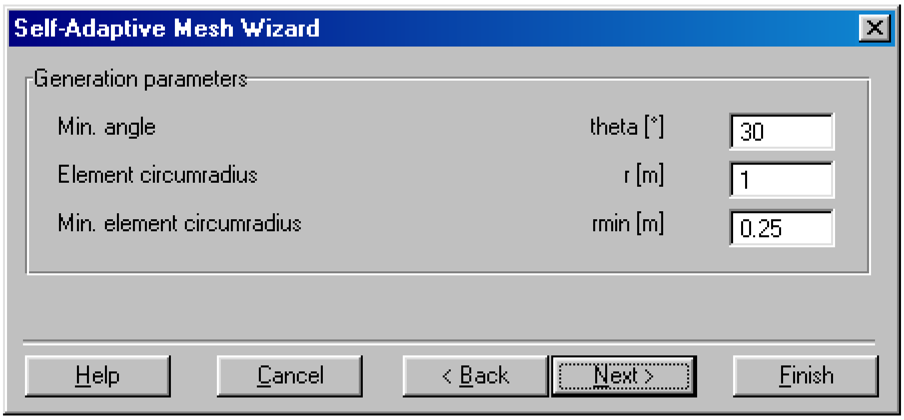


Figure 4 Delaunay generation parameters

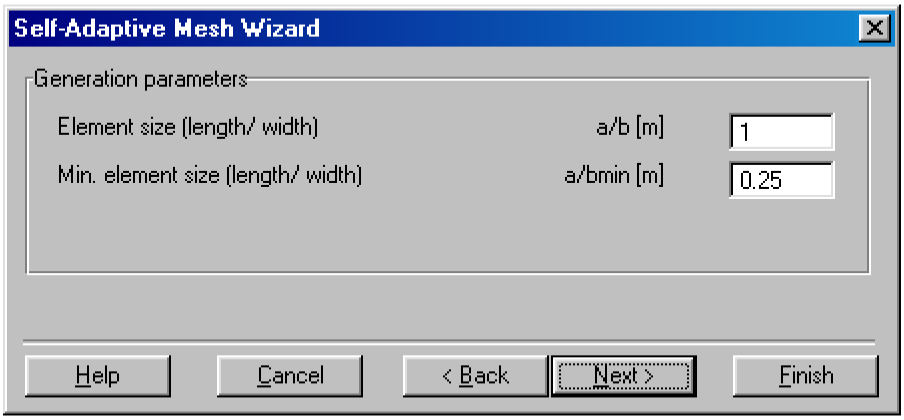


Figure 5 Grid based generation parameters

Adaptive Project

The last wizard form shown in Figure 6 displays the name and path of the output project. For selected project name, e.g. *ProjectName*, the wizard automatically displays an output name, e.g. *ProjectName_Adaptive*. You may type in the file name and path of the output project. You can also use the "Browse" button to navigate to the location where you want to save the project.

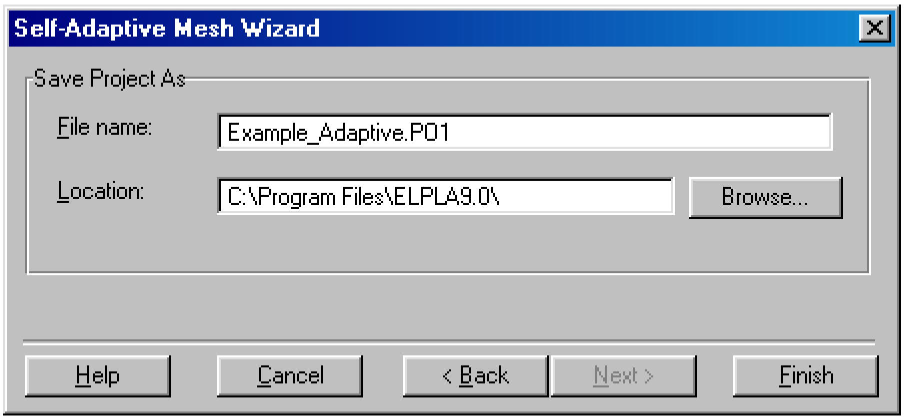


Figure 6 Grid based generation parameters

2 Samples for adaptive mesh

For the mesh of Example1 (Page 1), page 2 shows the adaptive mesh using Delaunay generation while page 3 shows the adaptive mesh using Grid-based generation.

Part C

User's Guide
for the program ELPLA-Data

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

Version 9.2

Program authors: M. El Gendy
A. El Gendy

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

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

Contents Page

1 An overview of ELPLA-Data

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

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

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

2 Description of ELPLA-Data

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

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

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

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

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

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

Table C-2 Names of file groups

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

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

Table C-3 Names and contents of files

A Main data files

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

B Data files

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

C Project data files

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

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

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

3 Starting ELPLA-Data

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

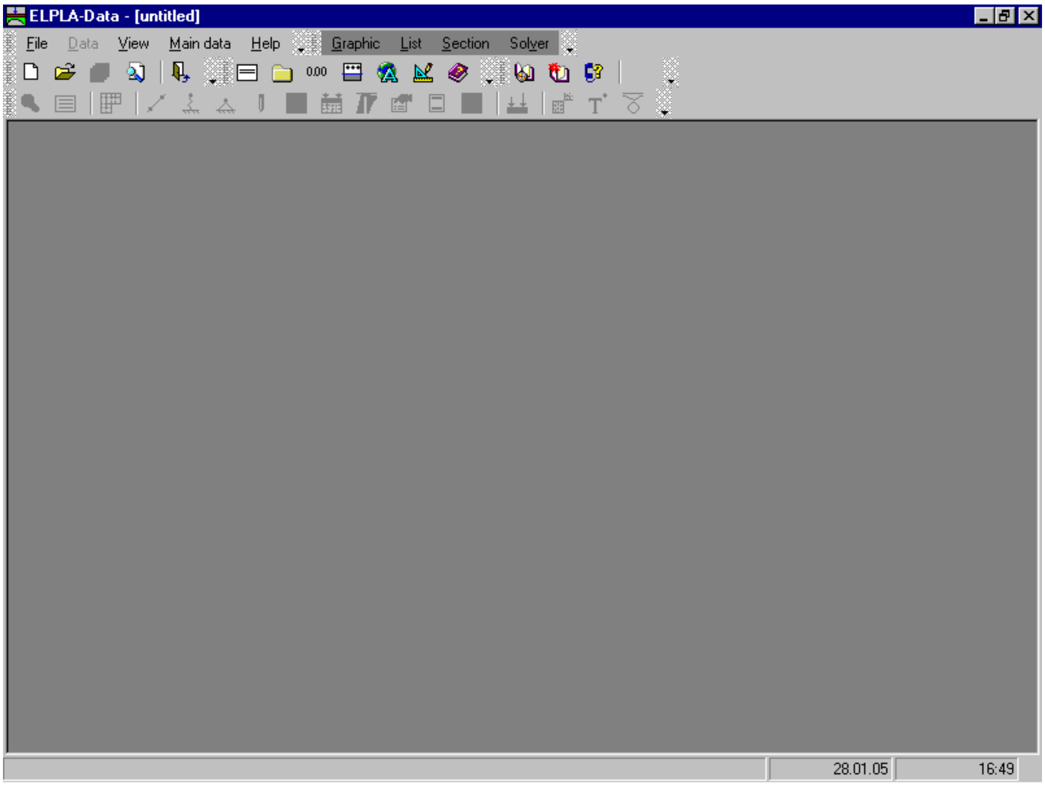


Figure C-1 Introduction screen of the program ELPLA-Data

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

- File
- Data
- View
- Main data
- Help

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

4 File Menu

The File Menu commands are:

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

4.1 File Menu –"New project" command

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

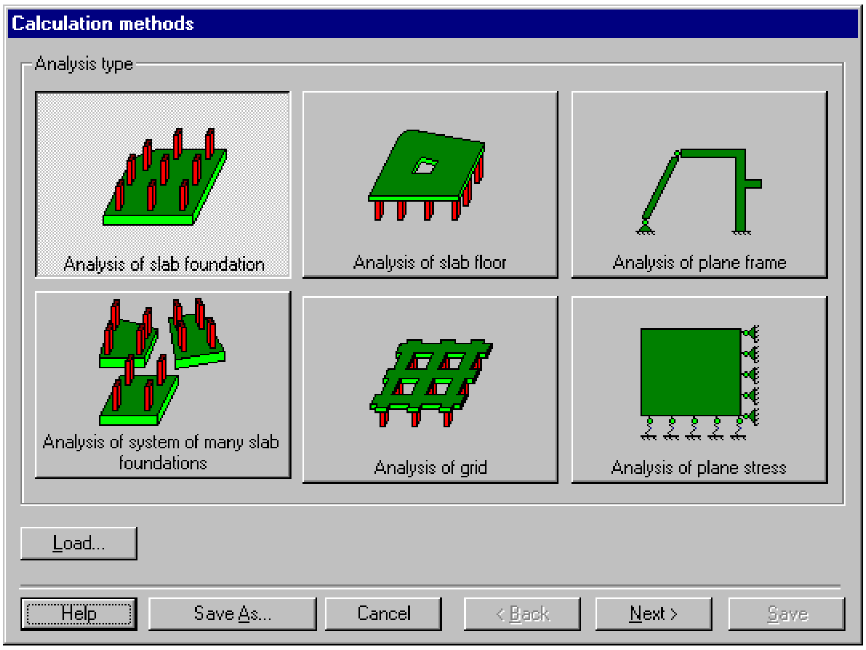


Figure C-2 "Analysis type" Form

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

4.2 File Menu –"Open project" command

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

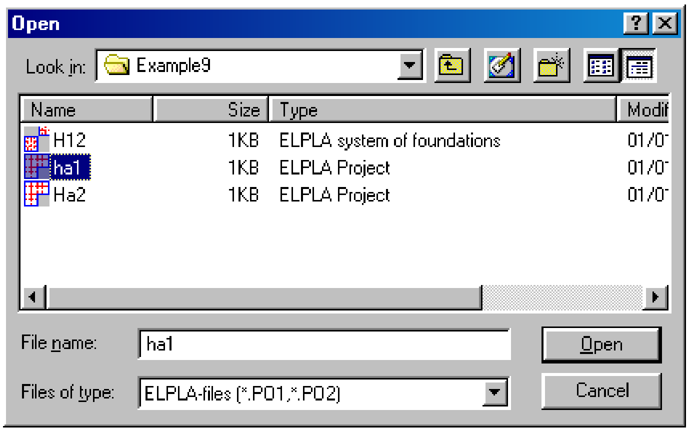


Figure C-3 "Open project" Dialog box

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

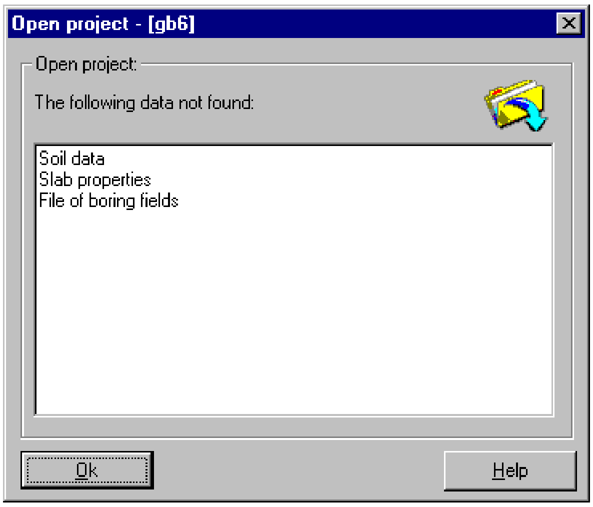


Figure C-4 "Open project" Message box

4.3 File Menu –"Save project as" command

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

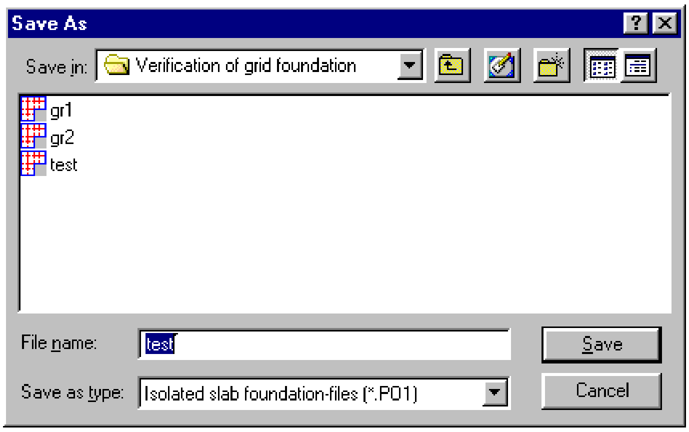


Figure C-5 "Save as" Dialog box

4.4 File Menu –"File list" command

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

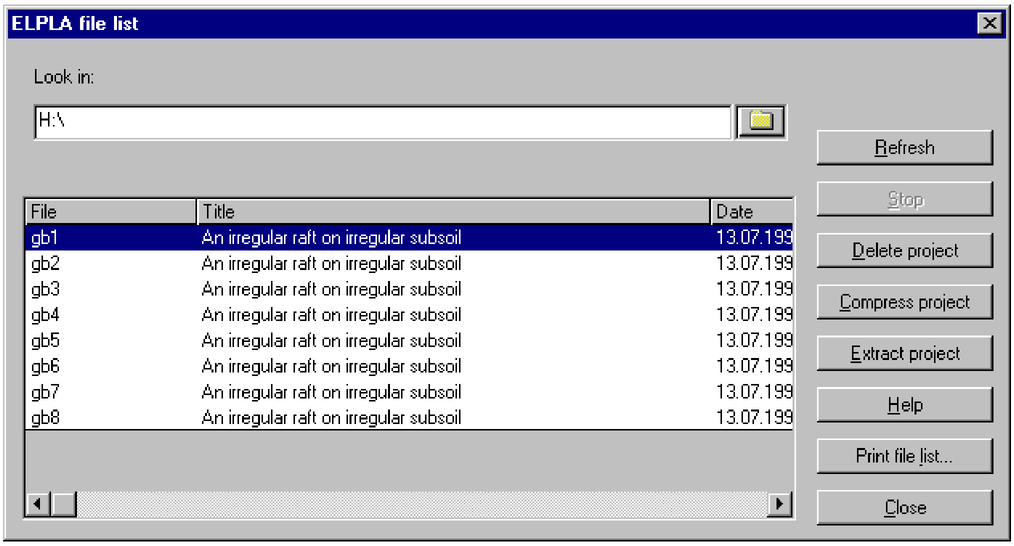


Figure C-6 "ELPLA File list" Dialog box

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

Note

Deleted files by ELPLA go to the recycled pin.

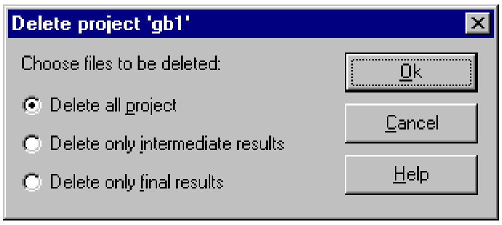


Figure C-7 "Delete project" Message box

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

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

4.6 File Menu –"Exit" command

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

Figure C-8 "Exit" Message box

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

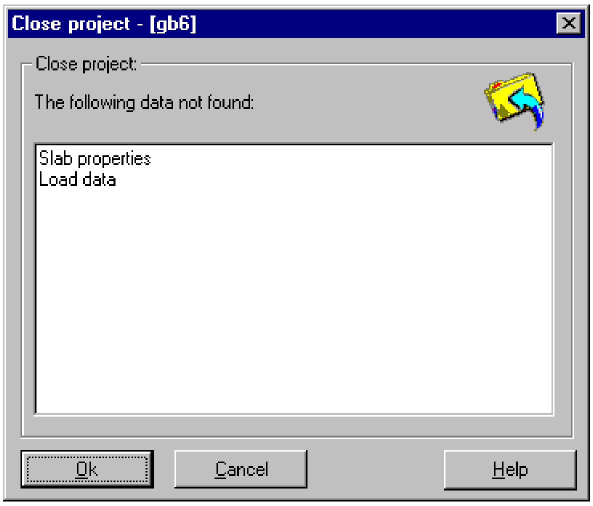


Figure C-9 "Close project" Message box

5 Data Menu

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

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

5.1 Data Menu–"Calculation methods" command

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

Calculation methods for an isolated slab foundation

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

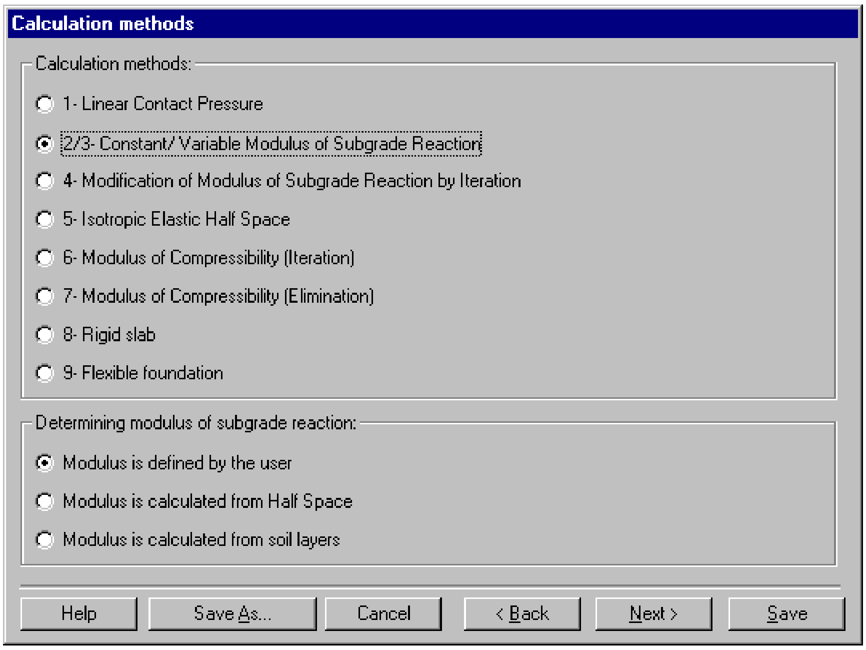


Figure C-10 "Calculation methods" Dialog box

System symmetry

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

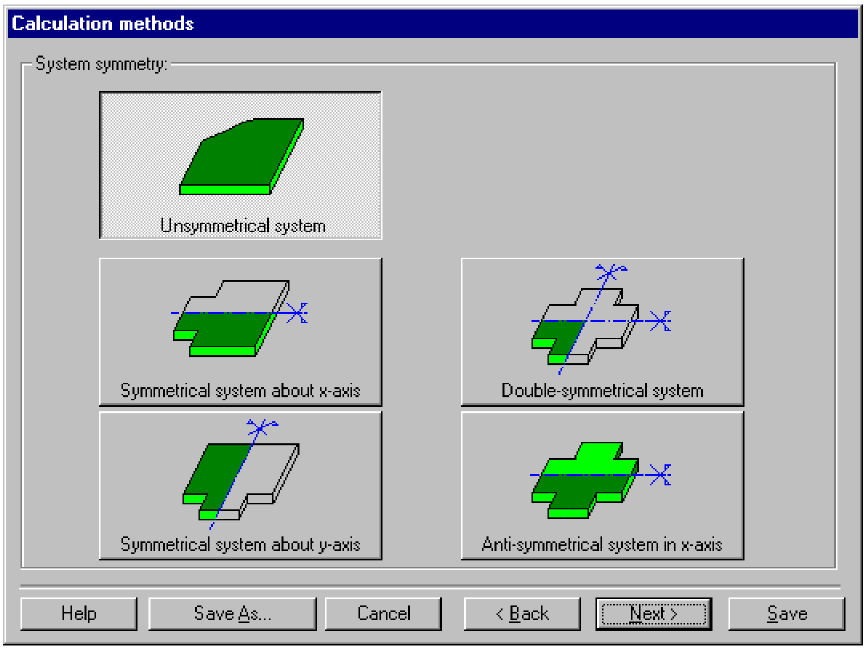


Figure C-11 "System symmetry" Form

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

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

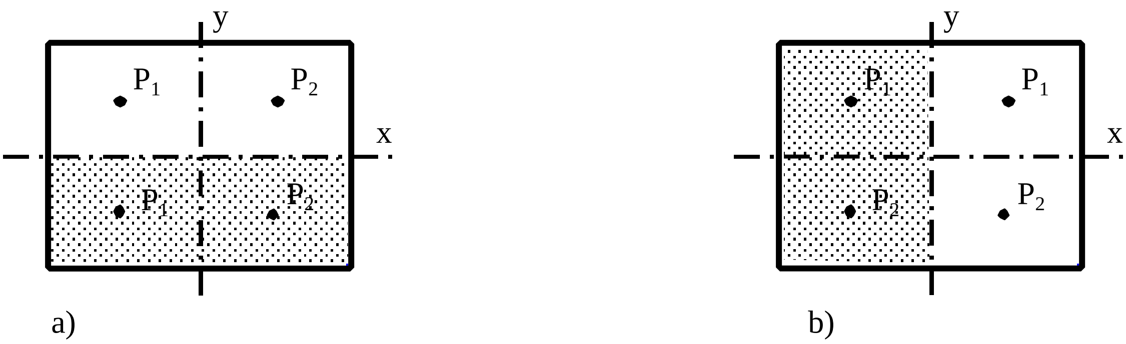


Figure C-12 Simple symmetrical slab system

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

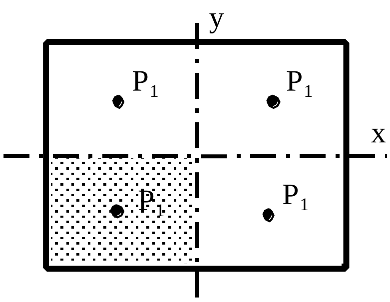


Figure C-13 Double symmetrical loaded slab

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

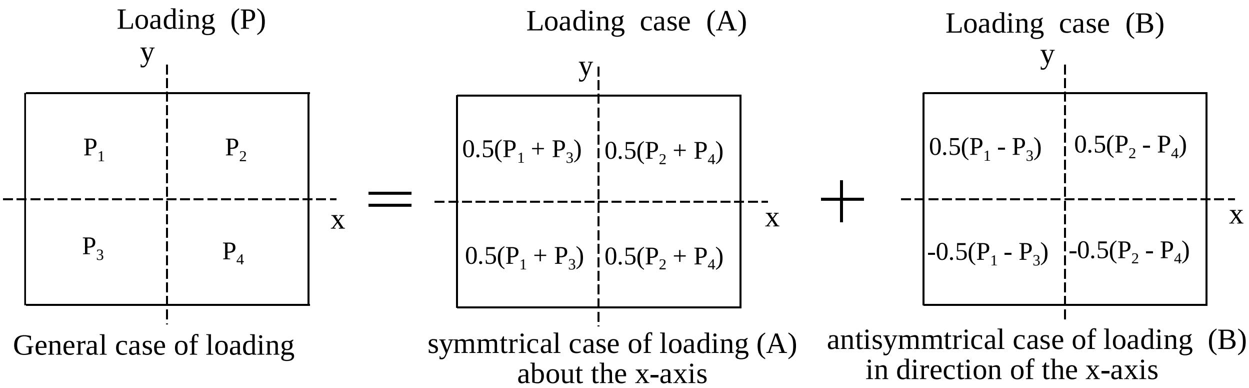


Figure C-14 General case of loading by symmetrical and antisymmetrical loading

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

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

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

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

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

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

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

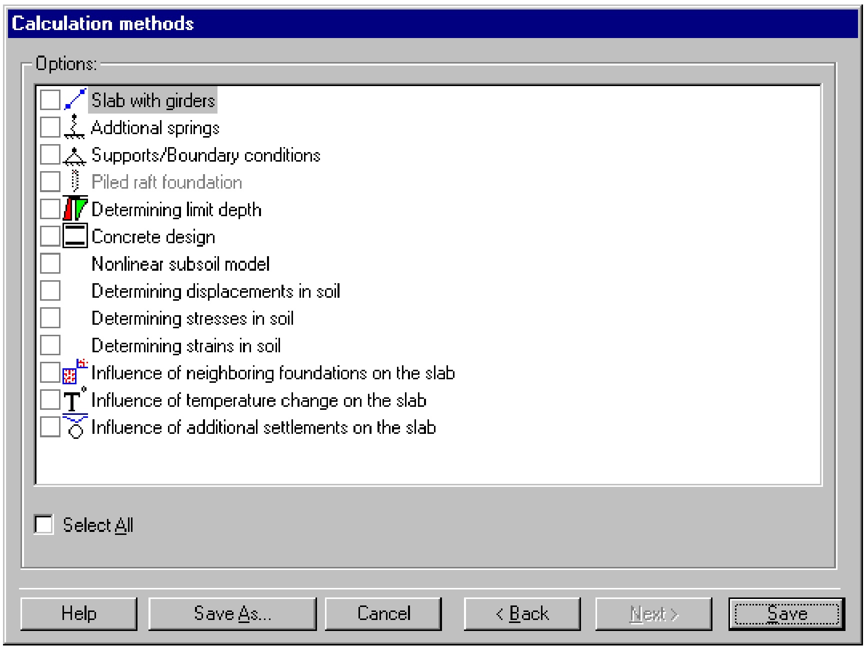


Figure C-15 "Options" Check box

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

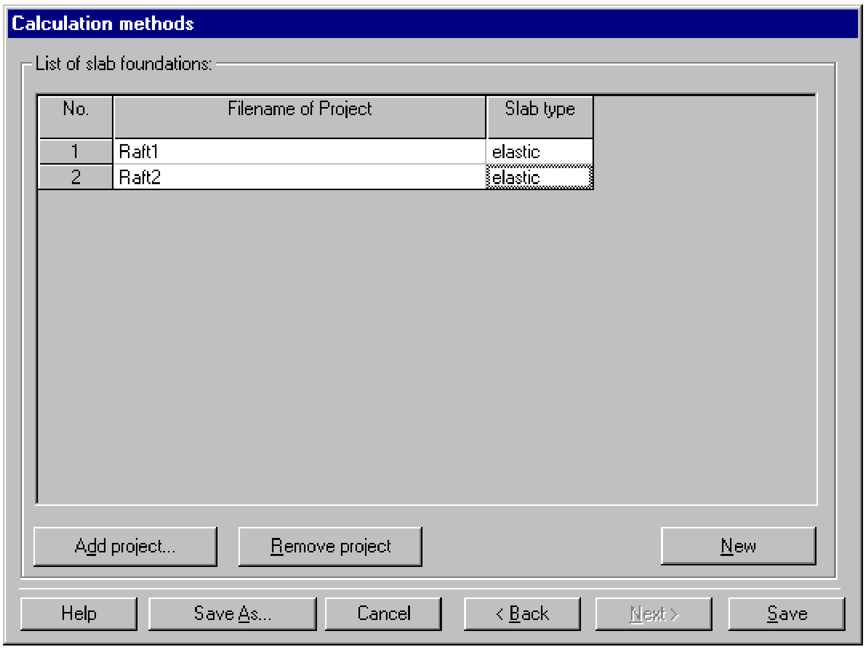


Figure C-16 "Analysis of system of many slab foundations" Dialog box

Analysis of a slab floor, plane frame or plane stress

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

5.2 Data Menu–"Project identification" command

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

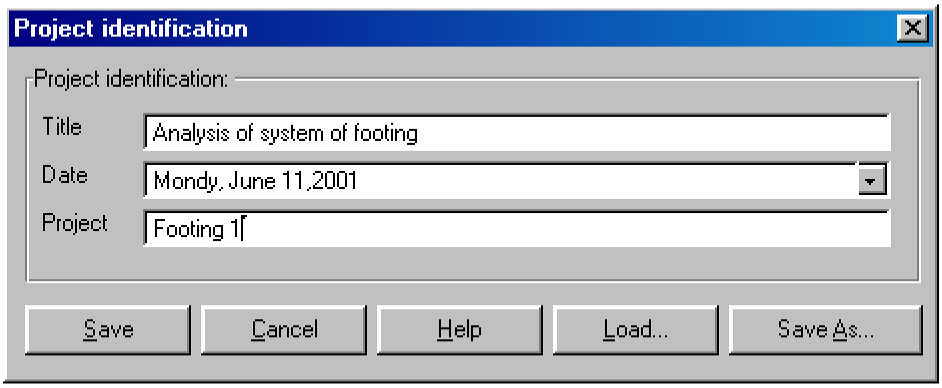


Figure C-17 "Project identification" Dialog box

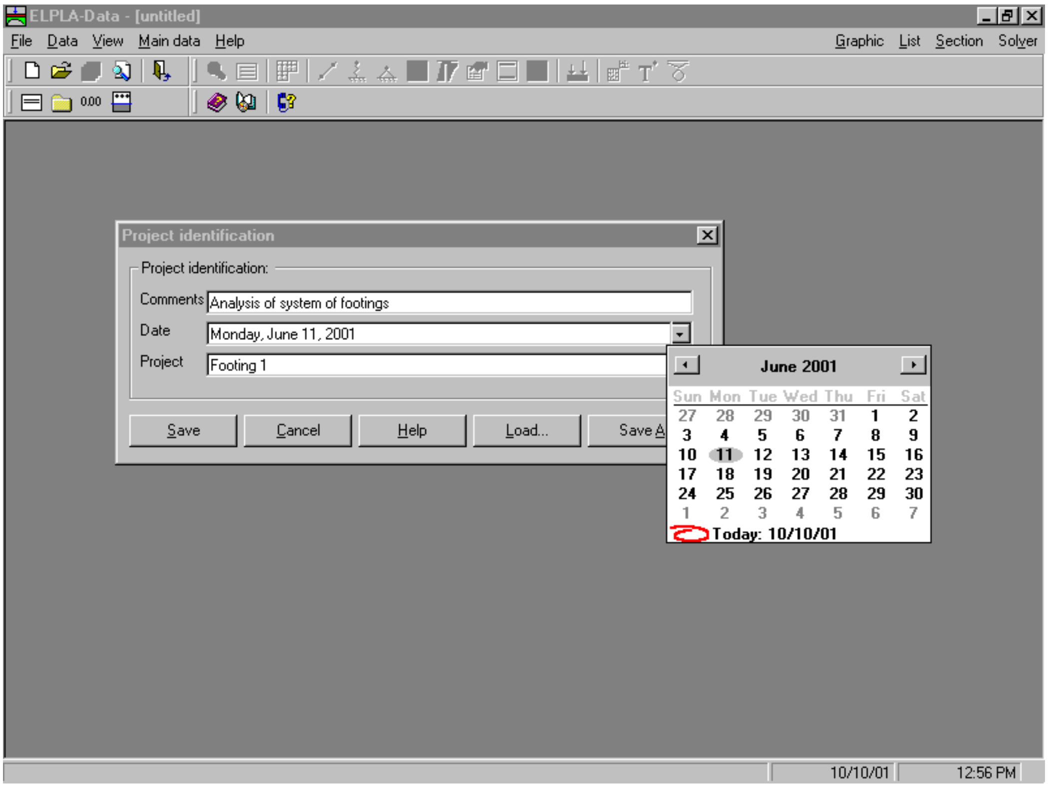


Figure C-18 Input Date information

5.3 Data Menu–"FE-Net data" command

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

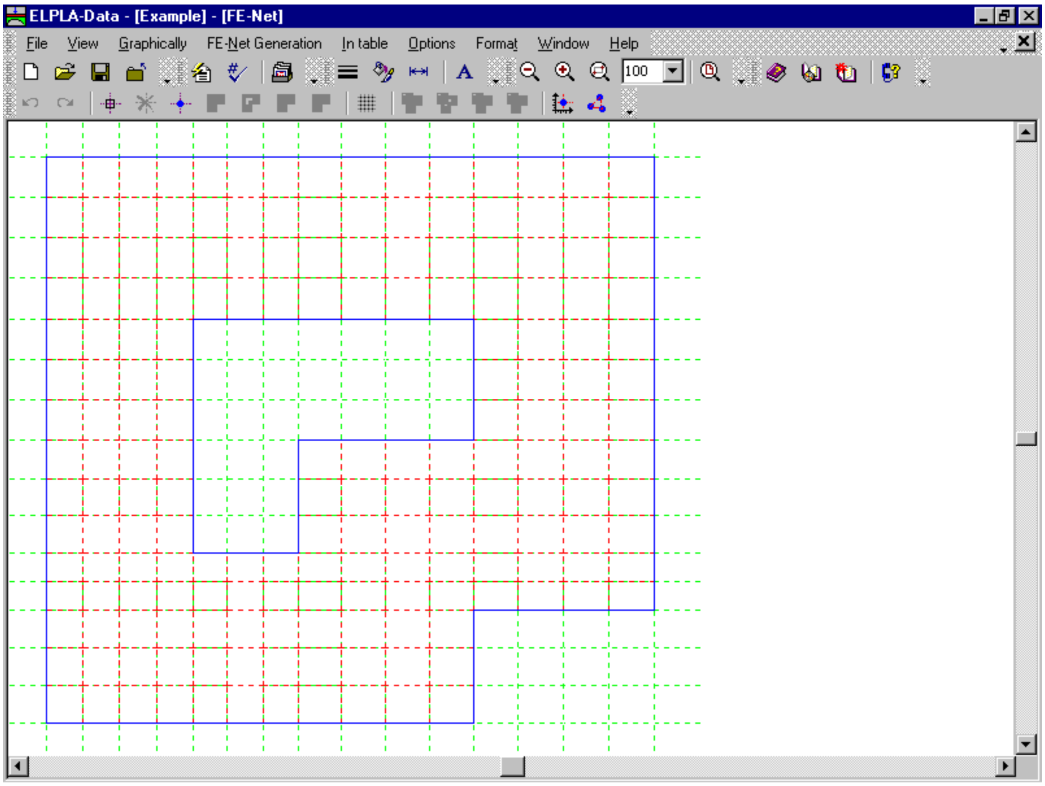


Figure C-19 "FE-Net" embedded program

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

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

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

5.3.1 File Menu

This menu contains the following commands:

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

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

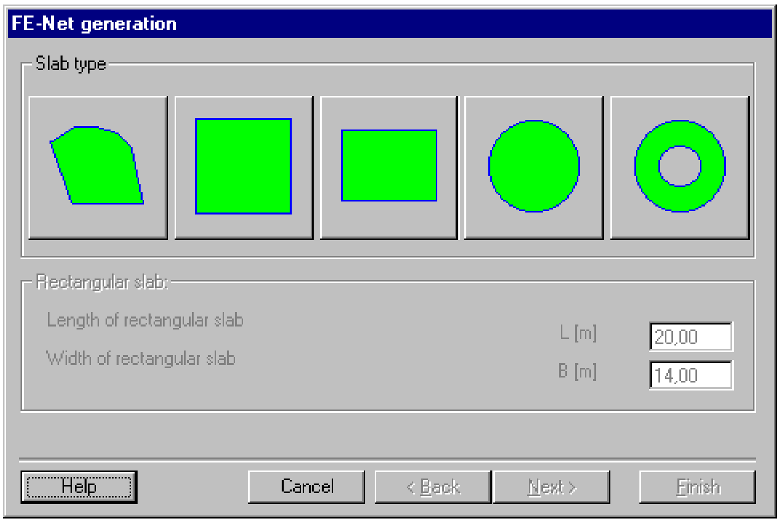


Figure C-20 Templates for different types of FE-Net

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

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

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

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

5.3.2 View Menu

The View Menu command is Tool bars.

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

5.3.3 Graphically Menu

This menu contains the following commands:

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

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

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

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

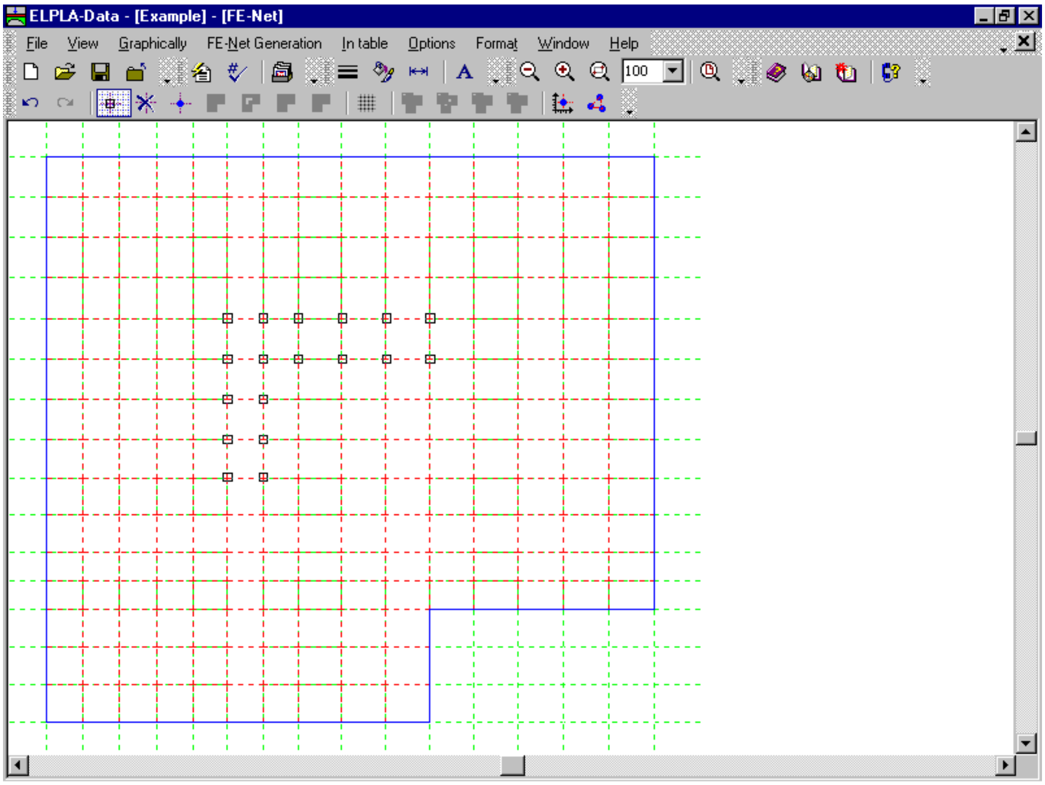


Figure C-21   Select nodes to be removed

Graphically Menu–"Remove nodes" command

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

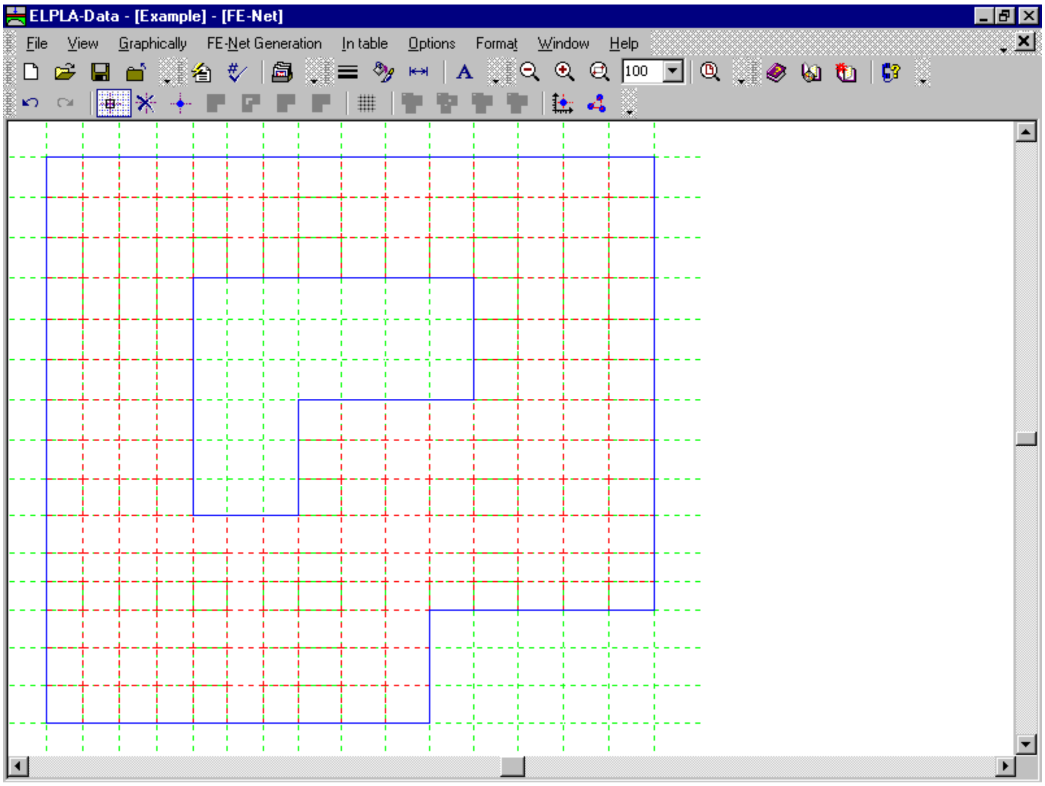


Figure C-22   Selected nodes were removed

Graphically Menu–"Add nodes" command

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

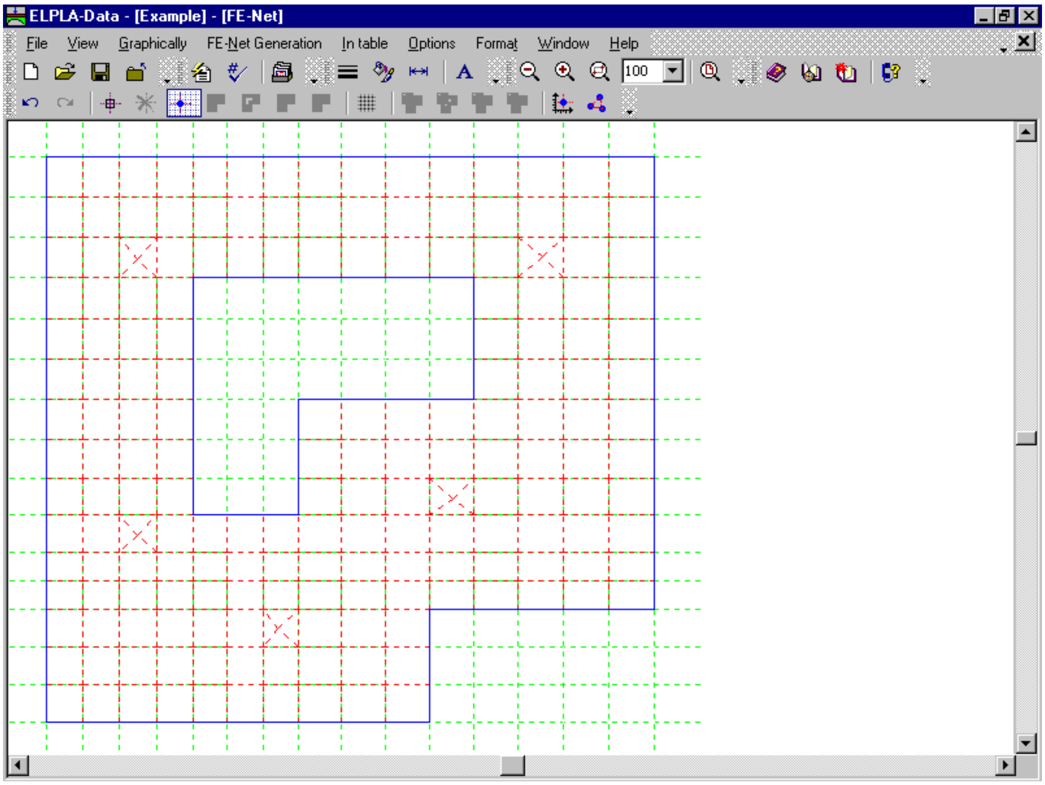


Figure C-23 Selected nodes were added

Graphically Menu–"Slab corners by Mouse" command

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

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

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

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

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

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

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

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

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

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

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

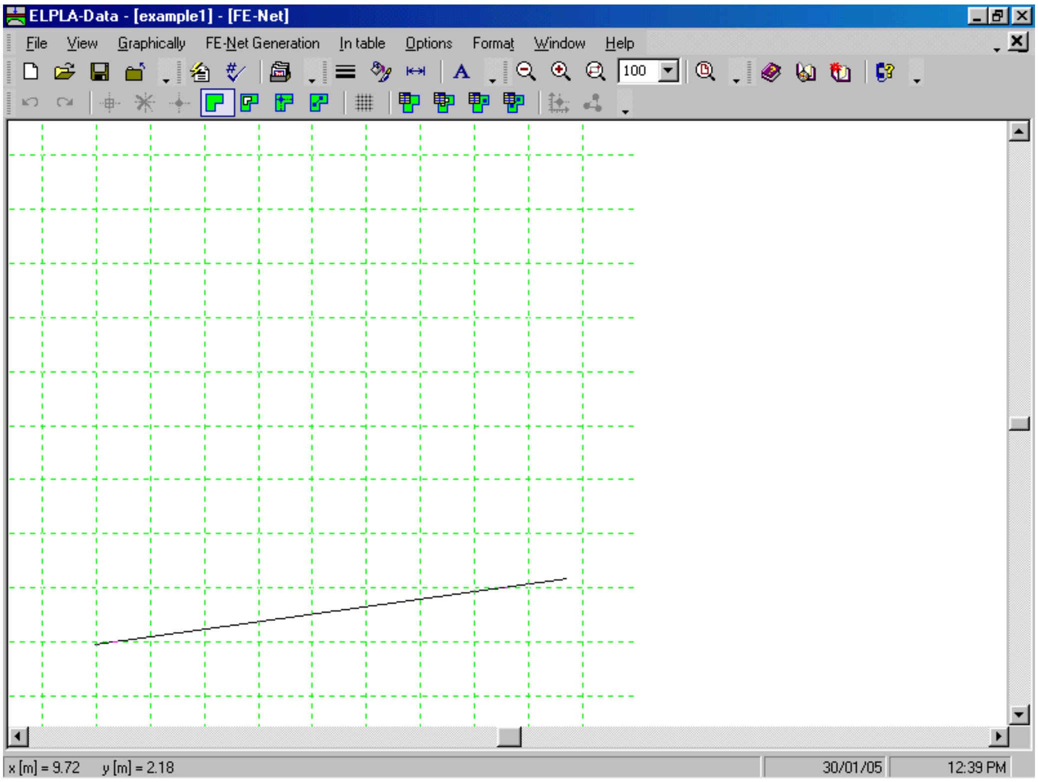


Figure C-24 Drawing the first line segment

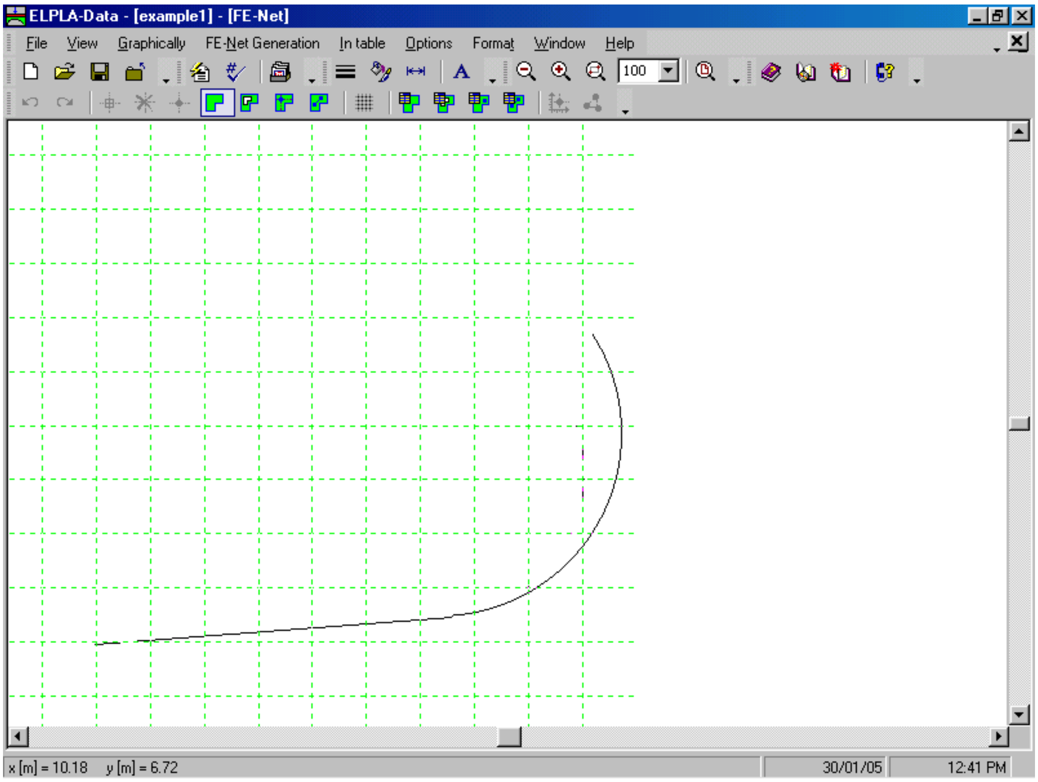


Figure C-25 Drawing the arc segment

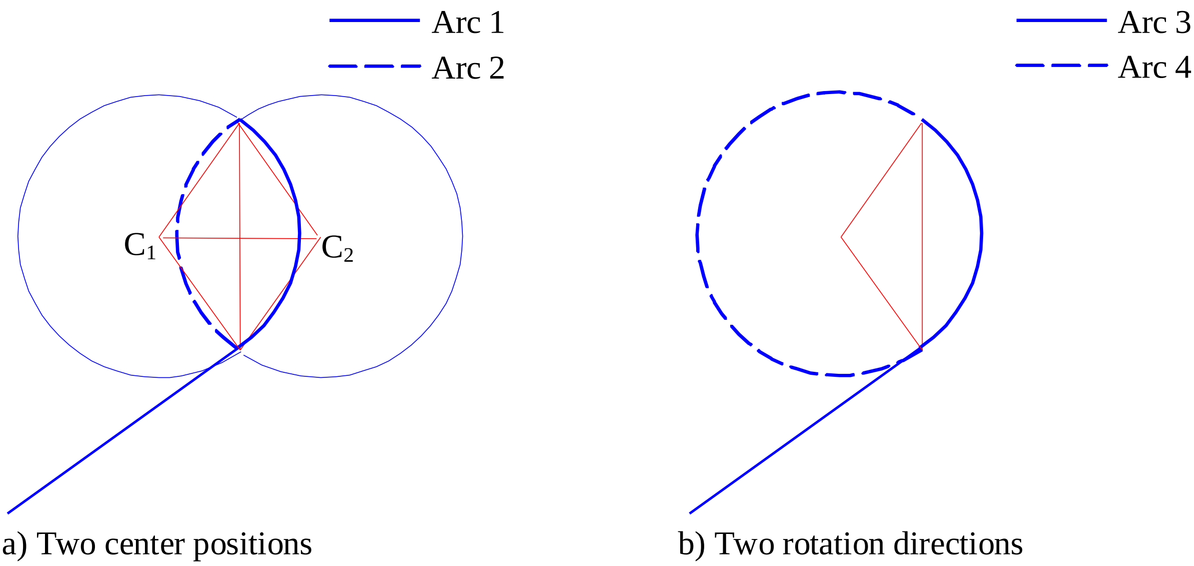


Figure C-26 Four cases for drawing arc segments

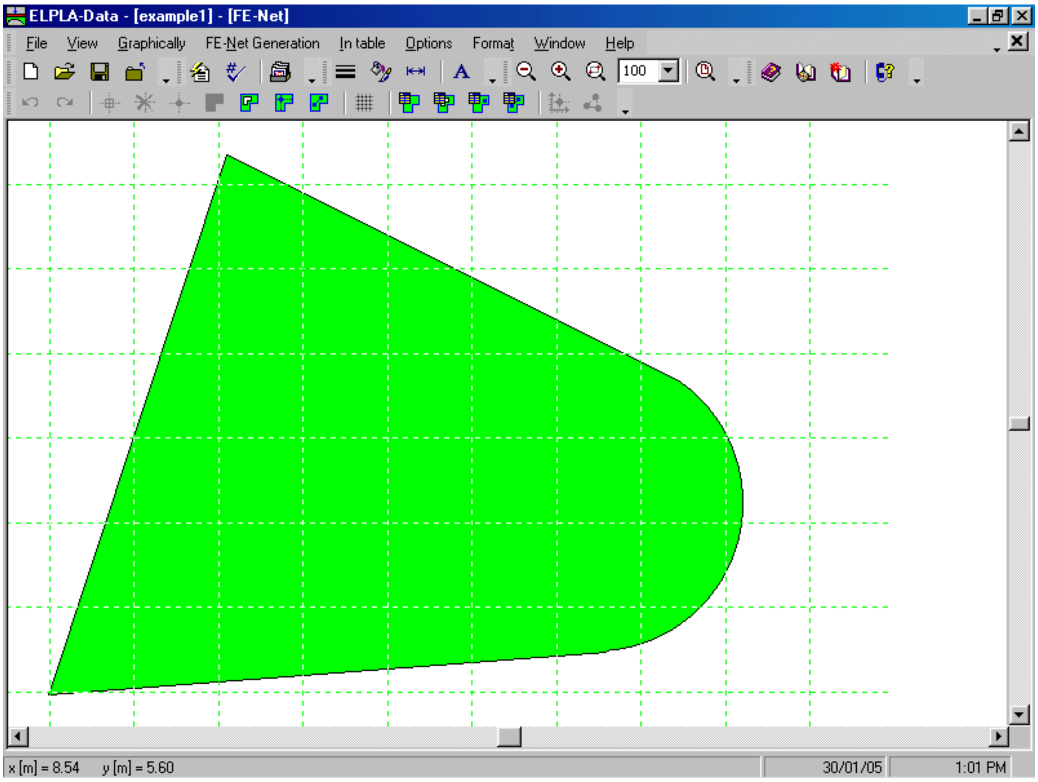


Figure C-27 Slab after defining corner points

Graphically Menu–"Add opening" command

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

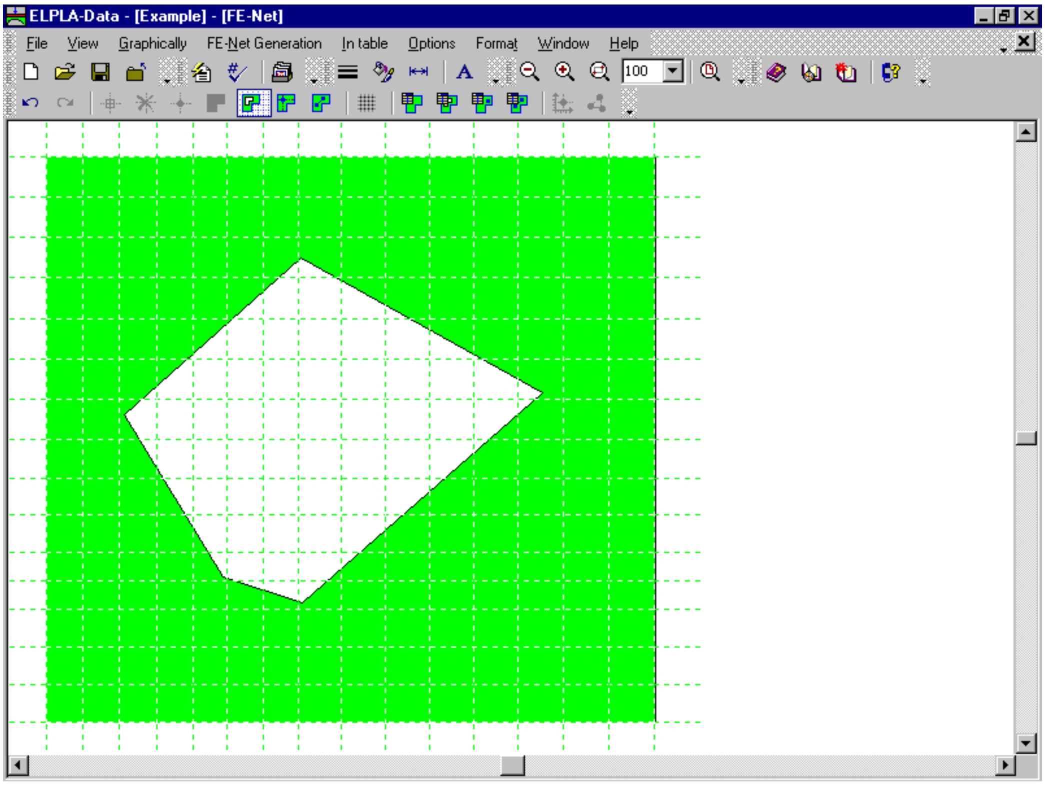


Figure C-28 Defining opening corners in the slab by Mouse

Graphically Menu–"Add reference points" command

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

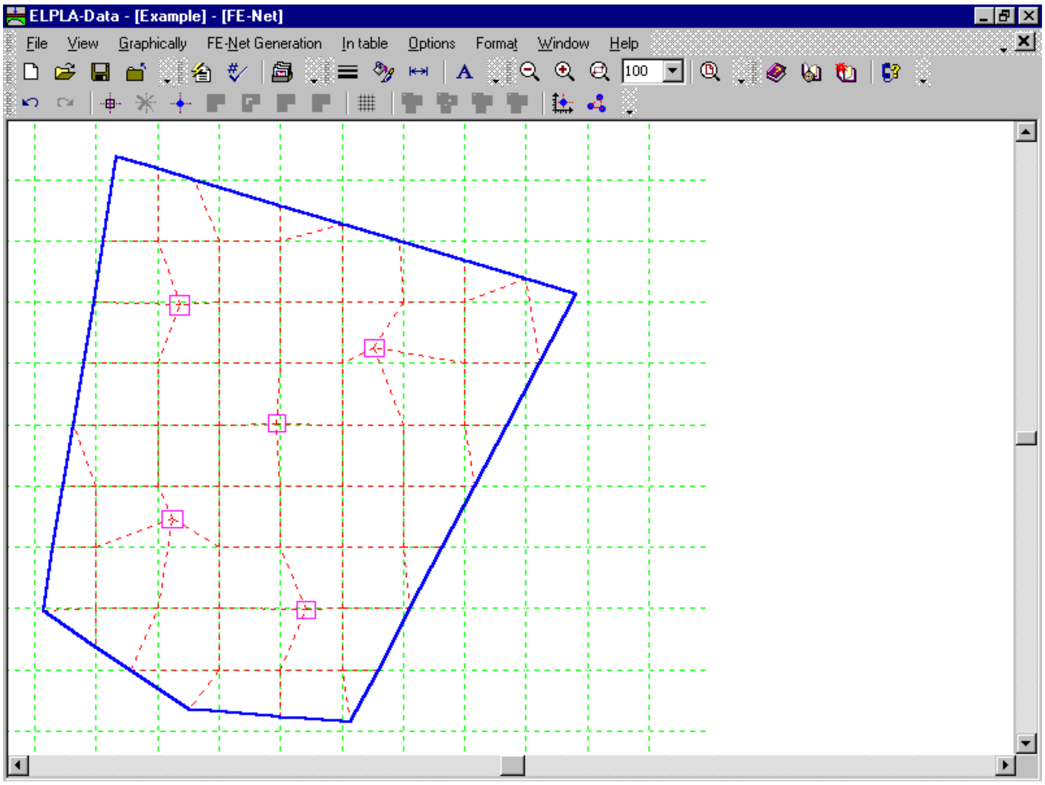


Figure C-29 Reference points on the FE-Net

Graphically Menu–"Add reference lines" command

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

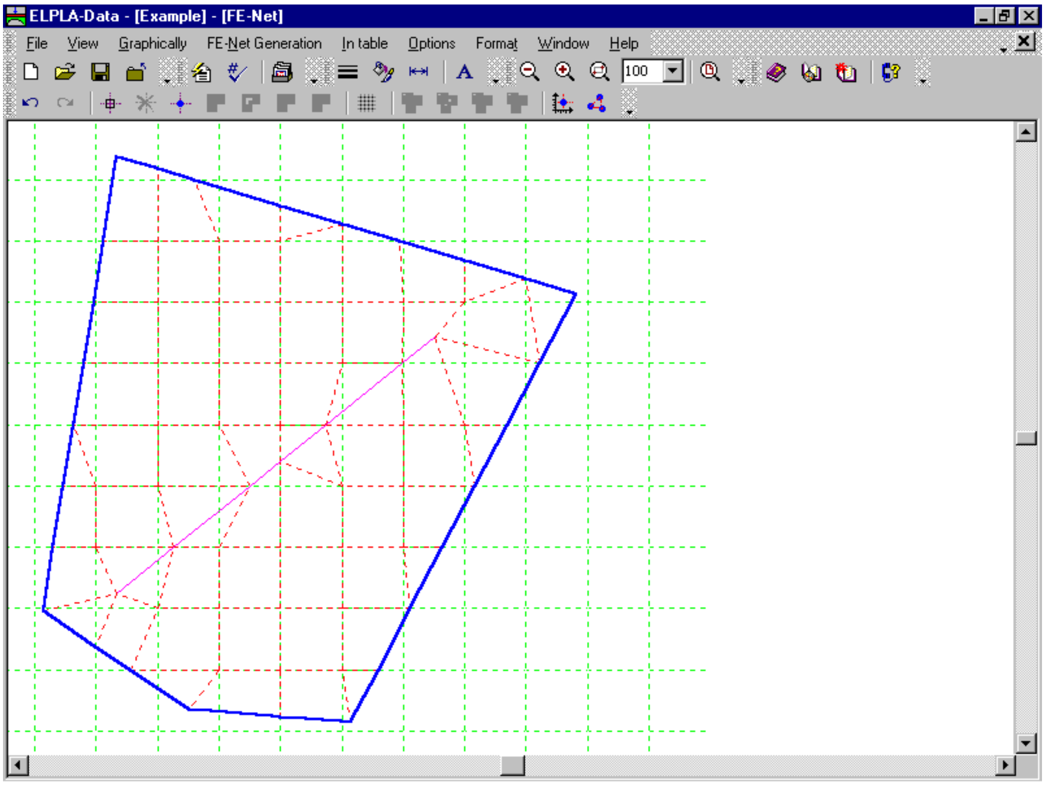


Figure C-30 Reference lines on the FE-Net

Graphically Menu–"Cartesian grid" command

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

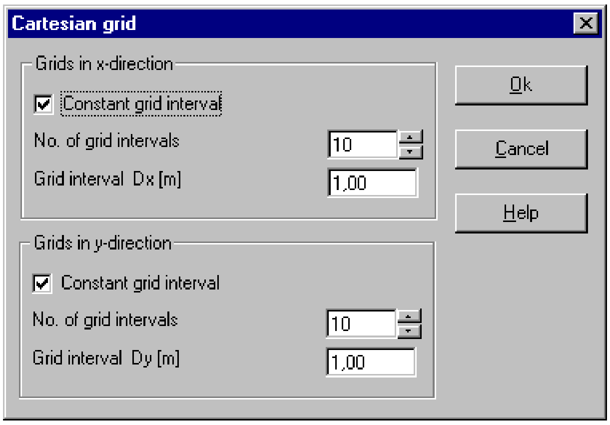


Figure C-31 "Cartesian grid" Dialog box

5.3.4 FE–Net Generation Menu

This menu contains the following commands:

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

FE–Net Generation Menu–"Generation type" command

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

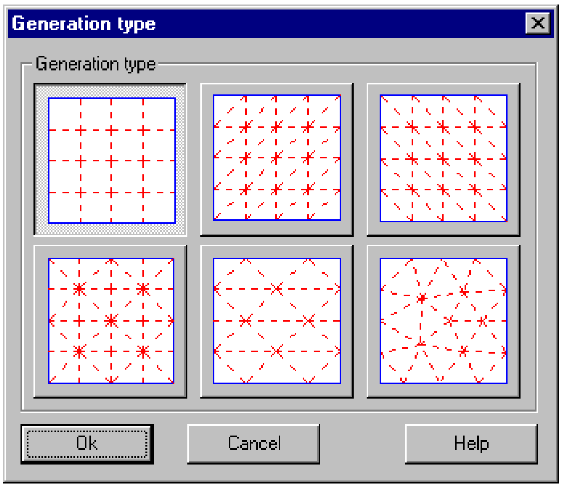


Figure C-32 Generation type for square, rectangular and irregular slabs

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

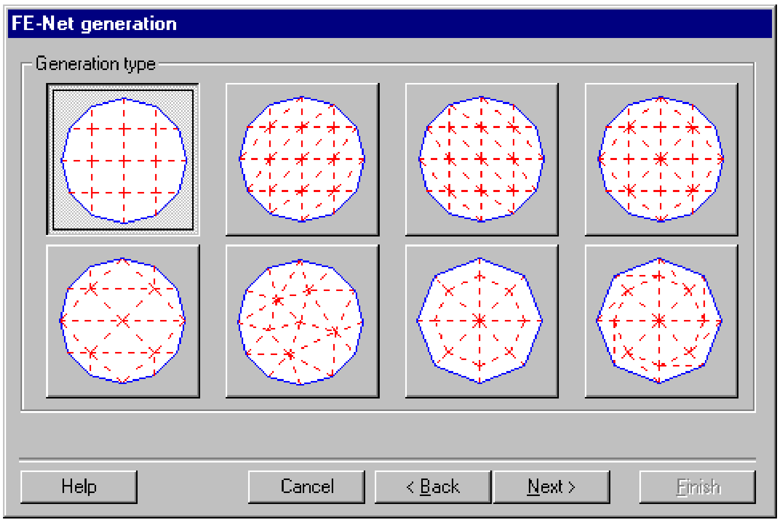


Figure C-33 Generation type for circular and ring slabs

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

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

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

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

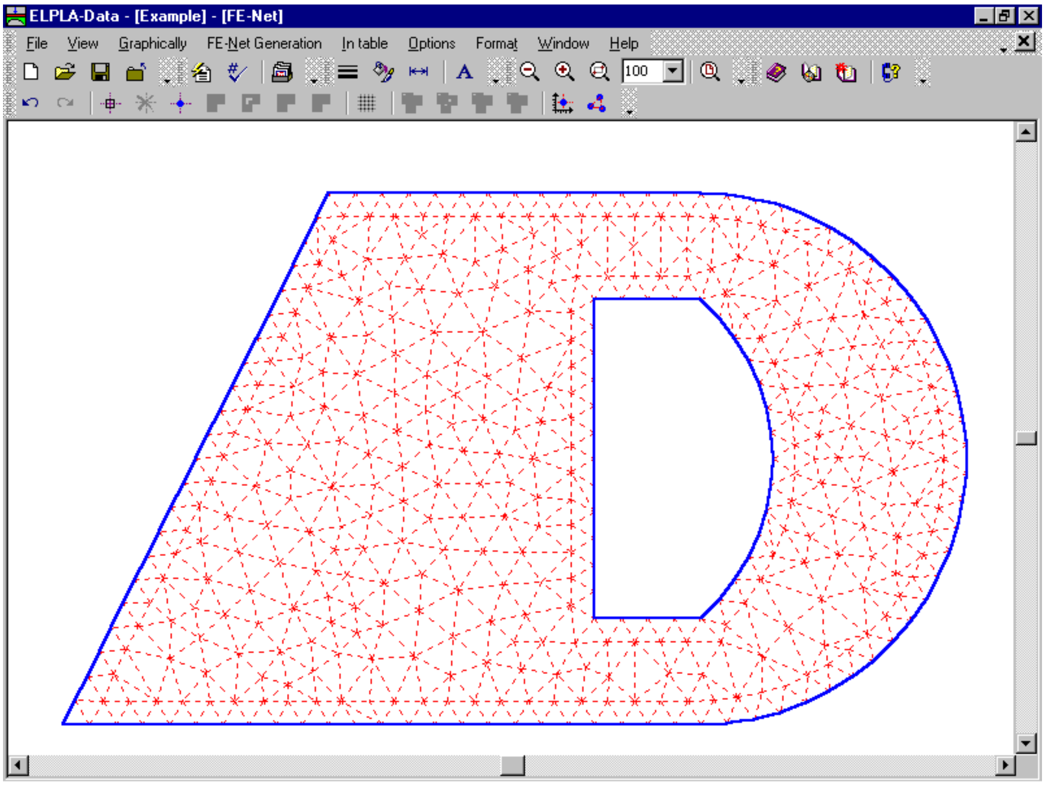


Figure C-34 Directing border elements

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

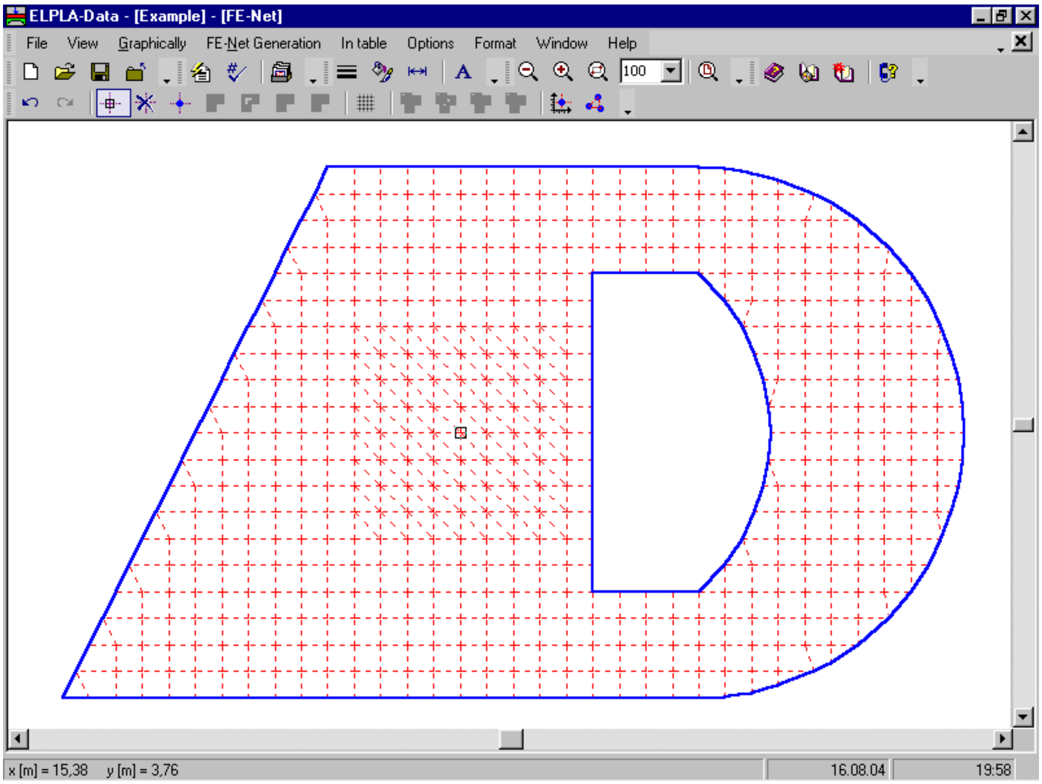


Figure C-35 Refining the mesh around a specified node

5.3.5 In table Menu

This menu contains the following commands:

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

In table Menu –"Slab corners" command

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

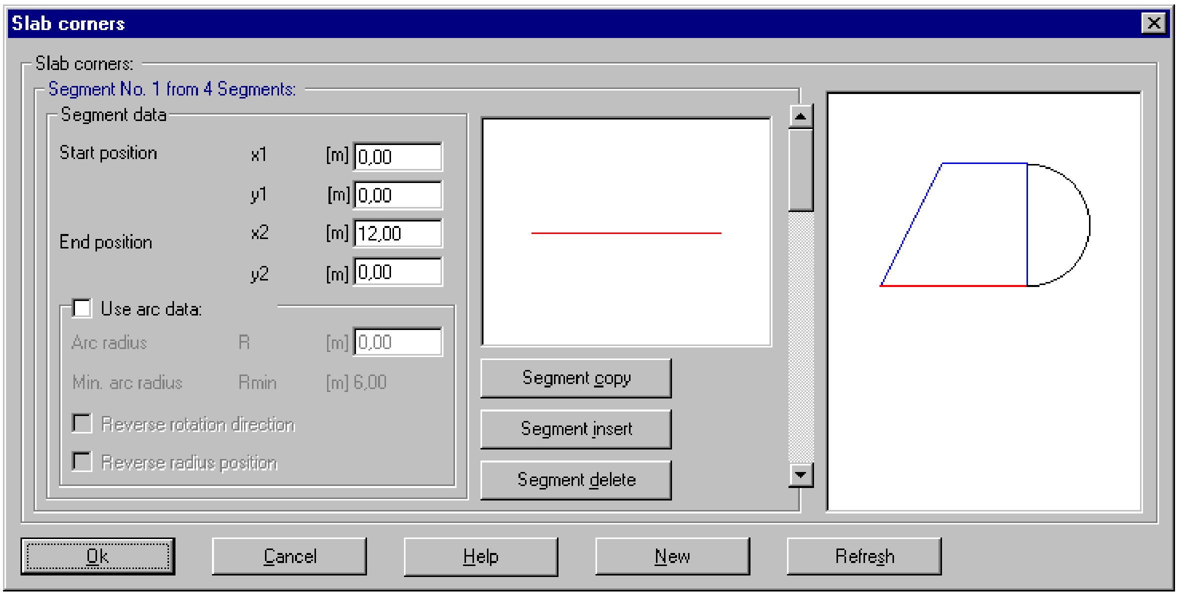


Figure C-36 Defining slab corners in a table

In table Menu–"Opening corners" command

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

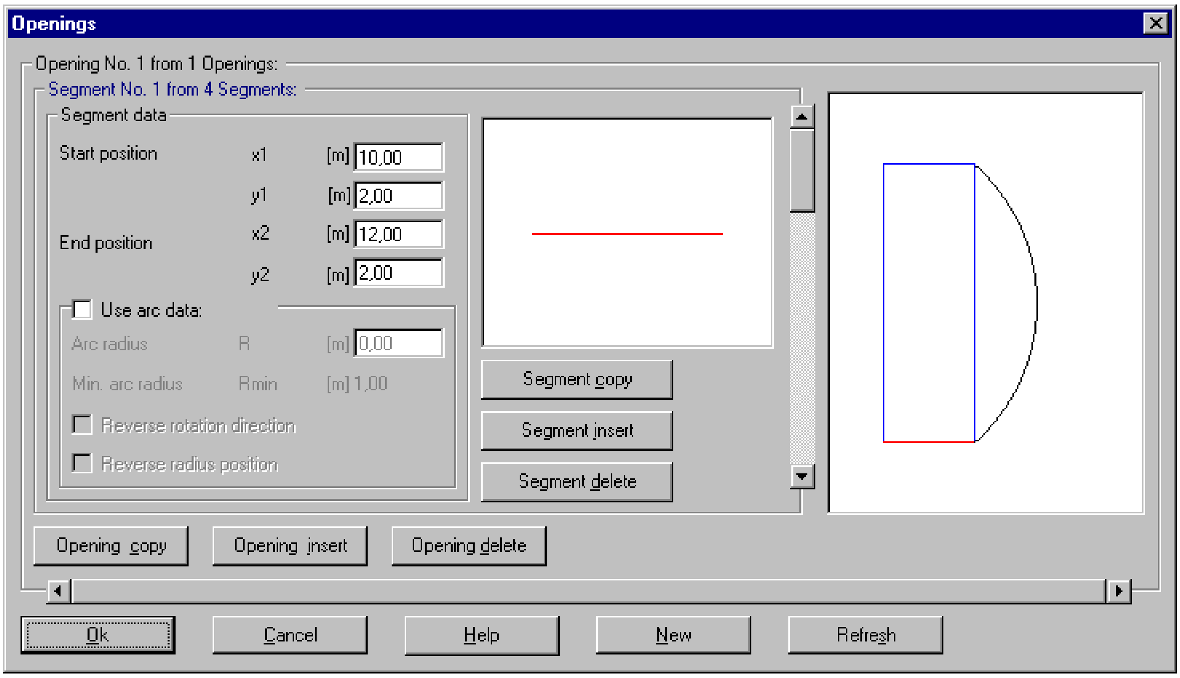


Figure C-37 Defining opening corners in a table

In table Menu–"Reference points" command

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

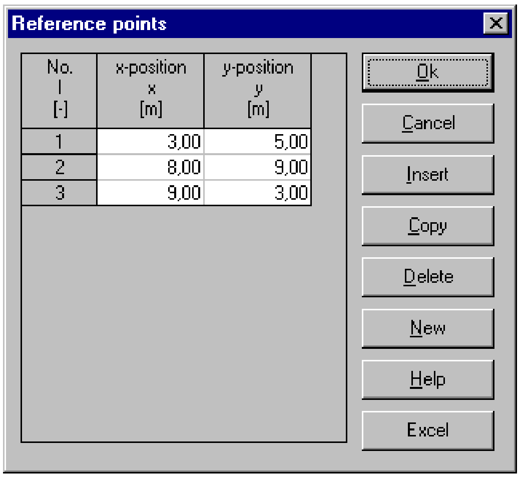


Figure C-38 Defining reference points in Table

In table Menu–"Reference lines" command

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

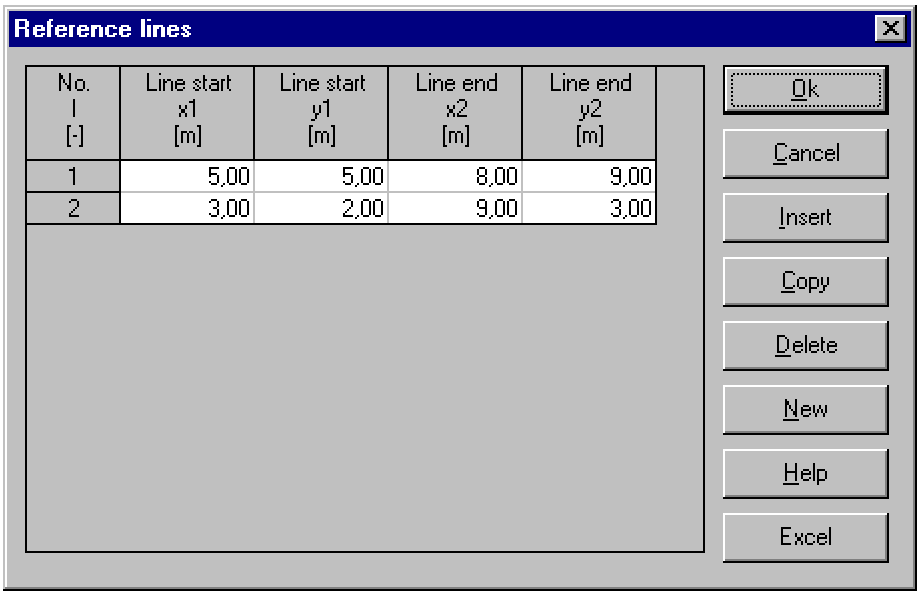


Figure C-39 Defining reference lines in a table

In table Menu–"Node coordinates" command

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

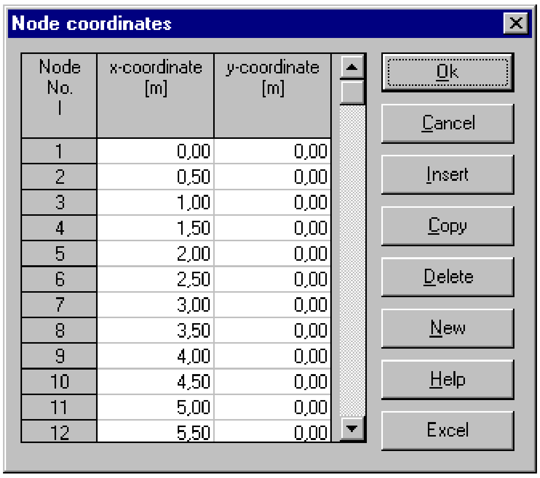


Figure C-40 Table "Node coordinate"

In table Menu–"Connectivity nodes" command

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

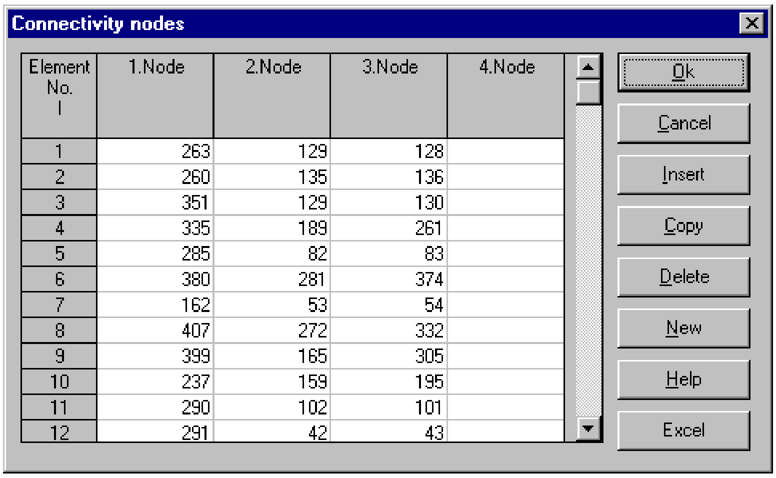


Figure C-41 Table "Net of finite elements"

5.3.6 Options Menu

The Options Menu has the following commands:

- Plot parameters
- Display values
- View grouping

Options Menu–"Plot parameters" command

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

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

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

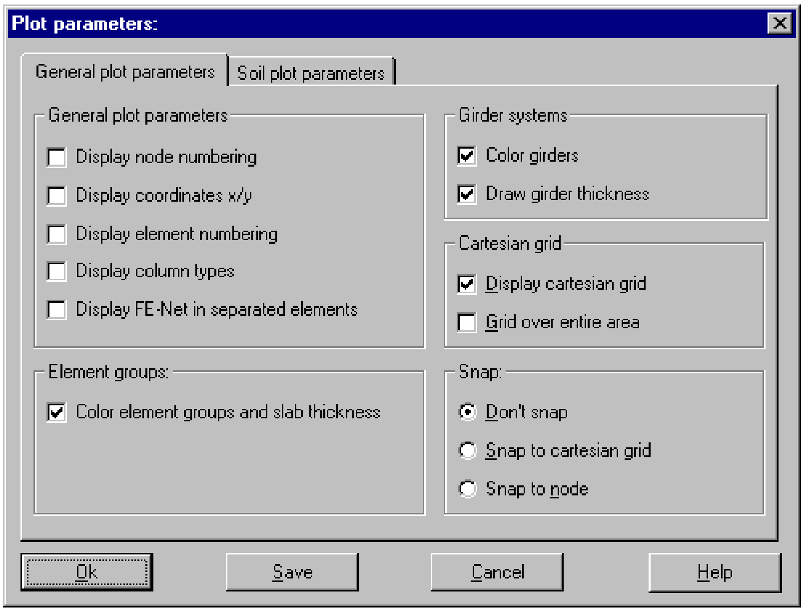


Figure C-42 "Plot parameters" Dialog box

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

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

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

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

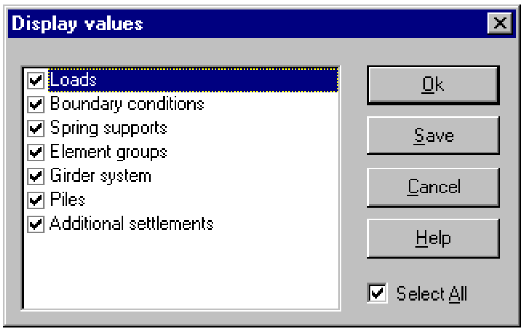


Figure C-43 "Display values" Dialog box

Options Menu–"View grouping" command

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

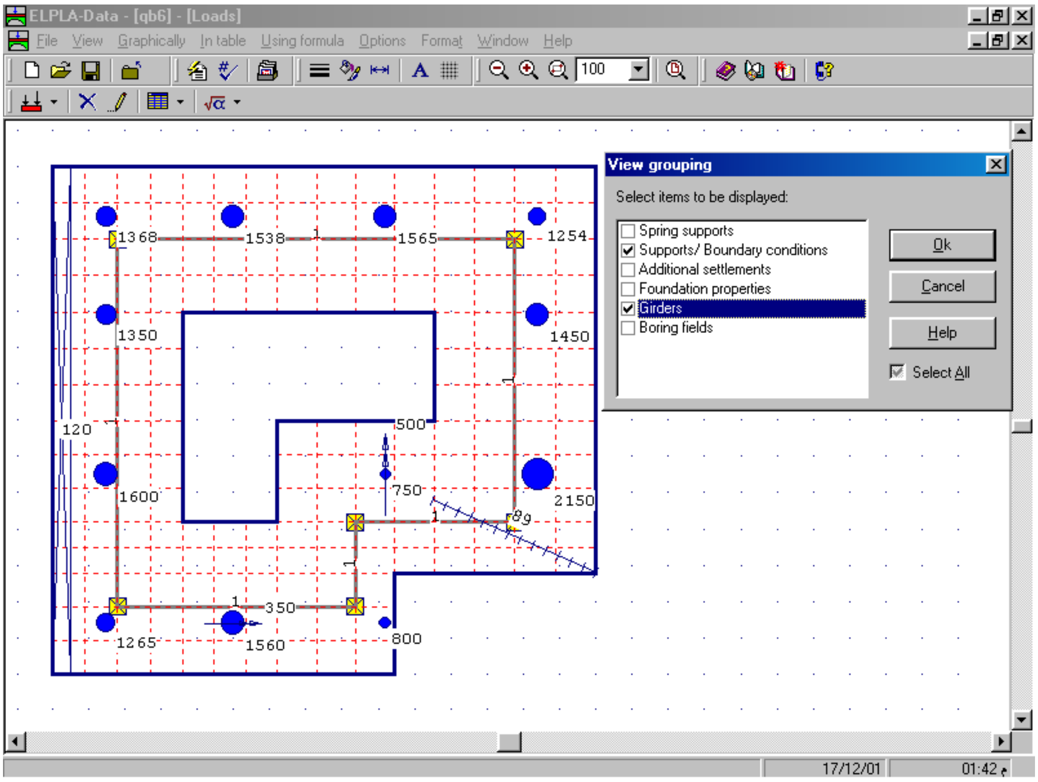


Figure C-44 Group of data together

5.3.7 Format Menu

The Format Menu has the following commands:

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

Format Menu–"Line formats" command

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

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

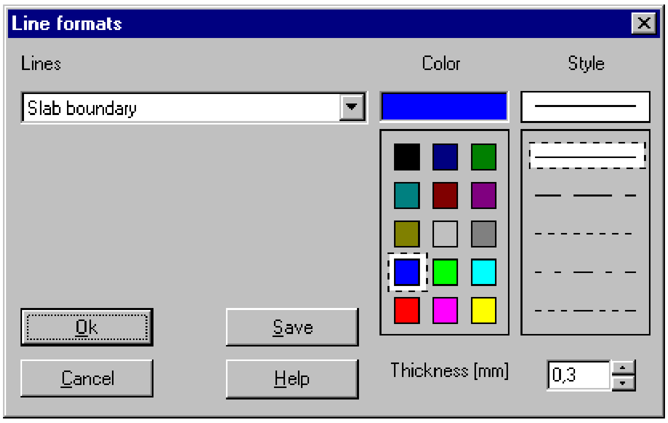


Figure C-45 "Line formats" Dialog box

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

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

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

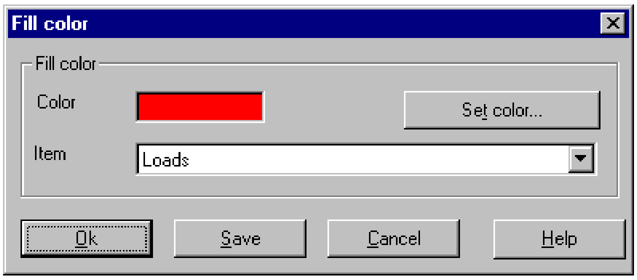


Figure C-46 "Fill color" Dialog box

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

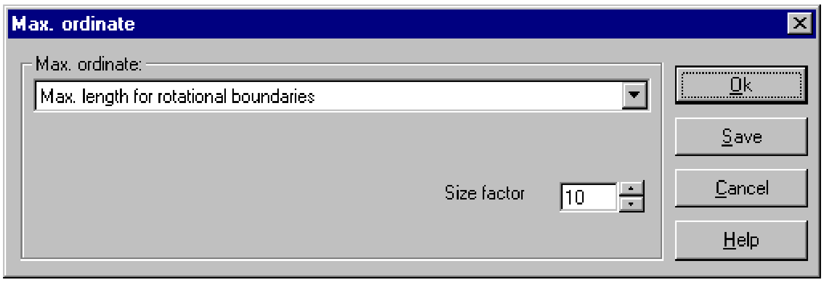


Figure C-47 "Max. ordinate" Dialog box

Format Menu–"Font" command
Here the font size (Figure C-48) and font type (Figure C-49) for the drawing can be defined.

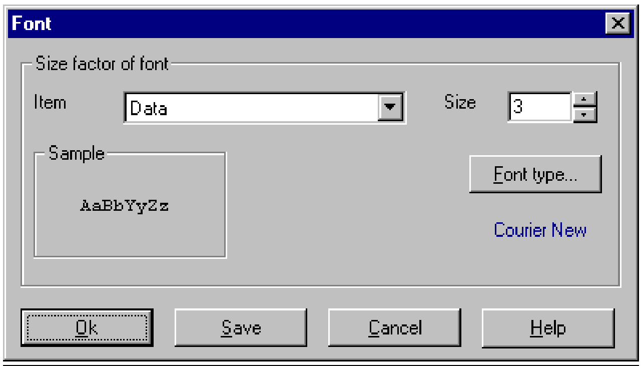


Figure C-48 "Font size" Dialog box

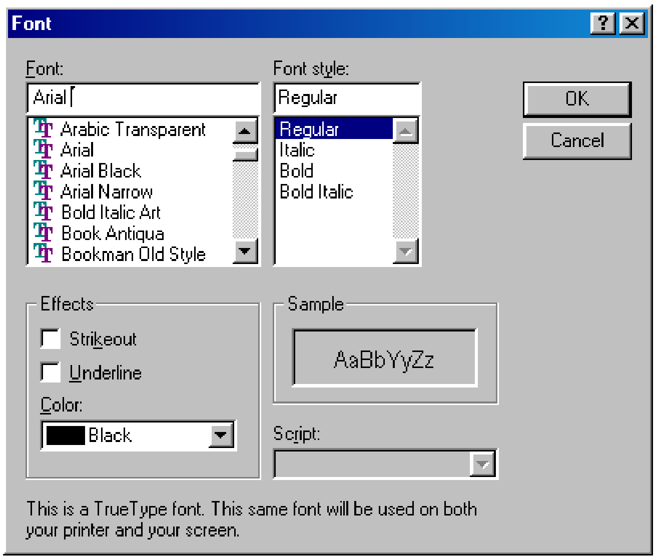


Figure C-49 "Font type" Dialog box

5.3.8 Window Menu

The Window menu has the following commands:

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

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

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

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

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

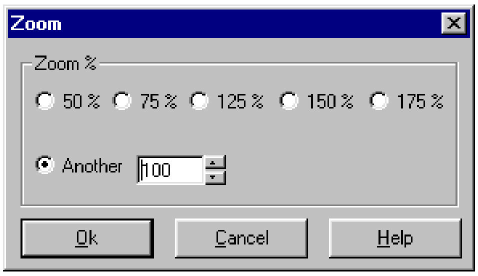


Figure C-50 "Zoom %" Dialog box

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

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

5.3.9 Help Menu

The Help Menu commands are:

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

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

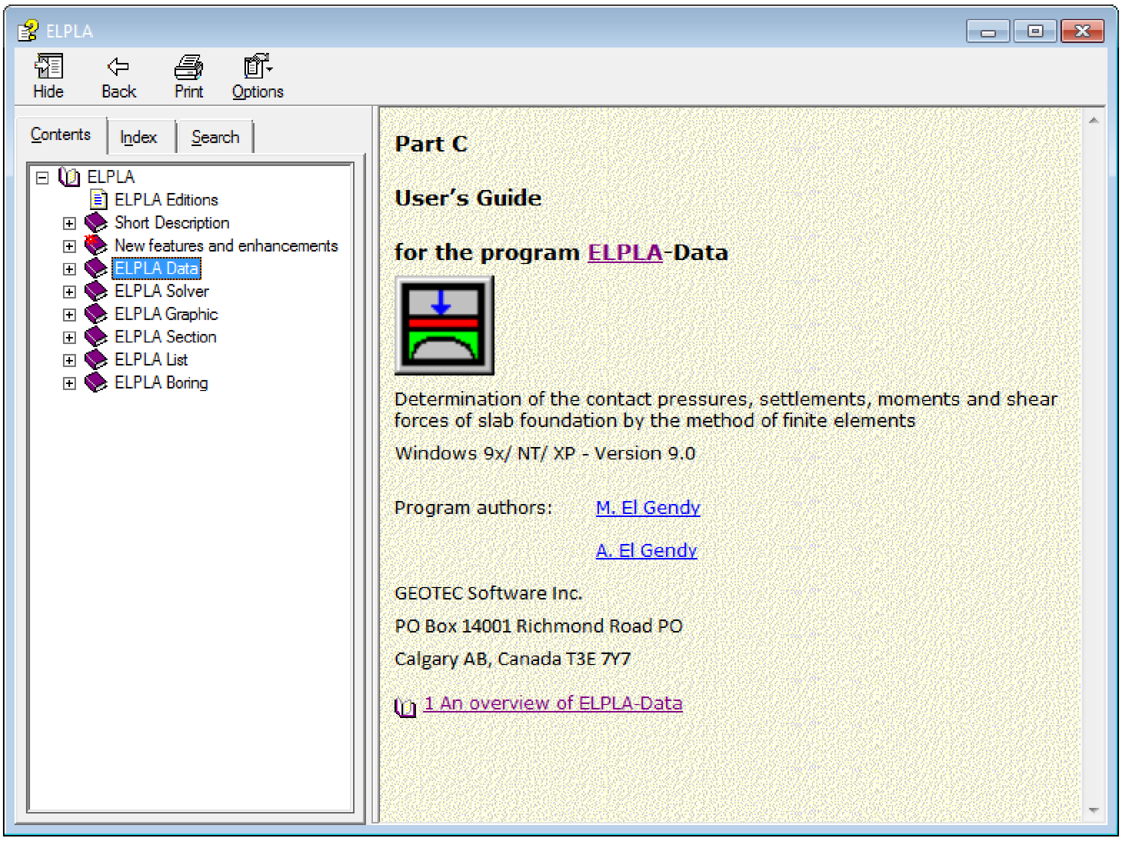


Figure C-51 Menu "Contents"

Help Menu–"Short description of ELPLA" command

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

Help Menu–"New in ELPLA" command

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

Help Menu–"About ELPLA-Data" command

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

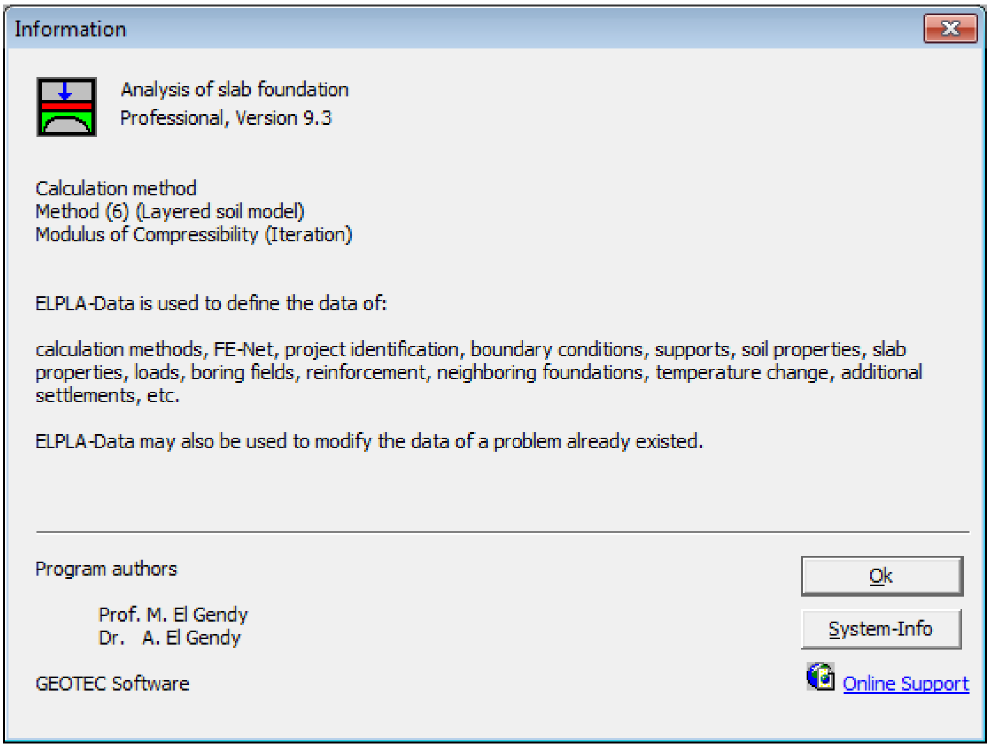


Figure C-52 Information form of program ELPLA-Data

5.3.10 Important notes

Element size

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

Change or modification of FE-Net

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

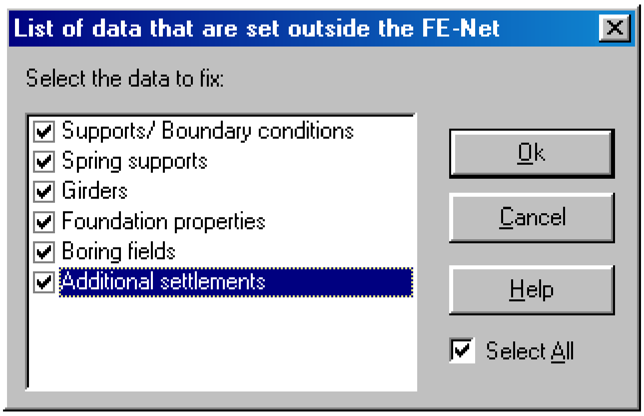


Figure C-53 Select the data to fix Dialog box

5.4 Data Menu–"Girders" command

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

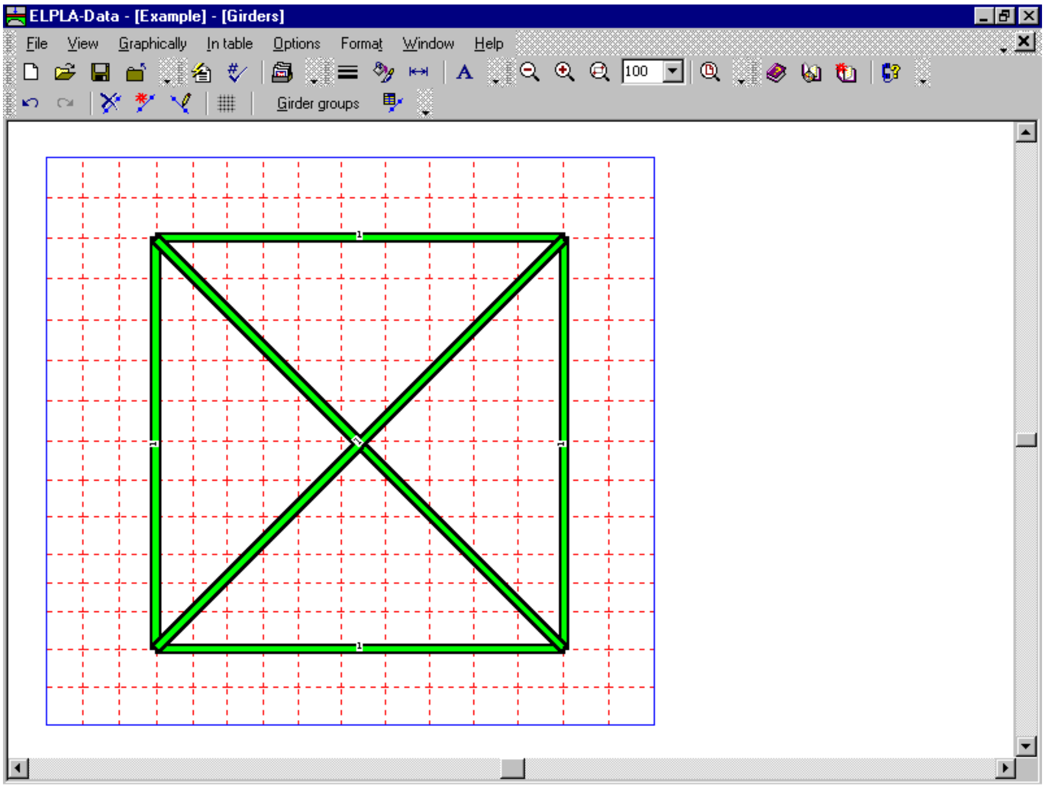


Figure C-54 "Girders" embedded program

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

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

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

5.4.1 File Menu

This menu contains five commands:

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

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

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

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

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

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

5.4.2 View Menu (See paragraph 5.3.2)

5.4.3 Graphically Menu

The menu "Graphically" contains the following commands:

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

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

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

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

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

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

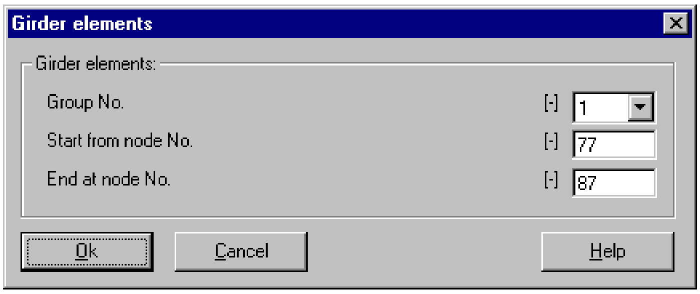


Figure C-55 "Girder elements" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.4.4 In table Menu

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

- Girder groups
- Girders

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

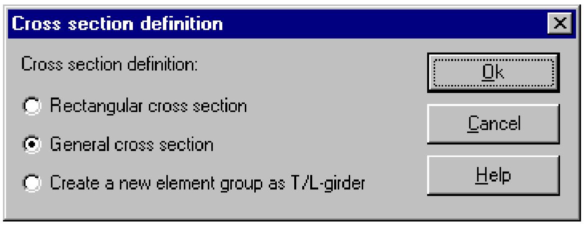


Figure C-56 "Cross section definition" Dialog box

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

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

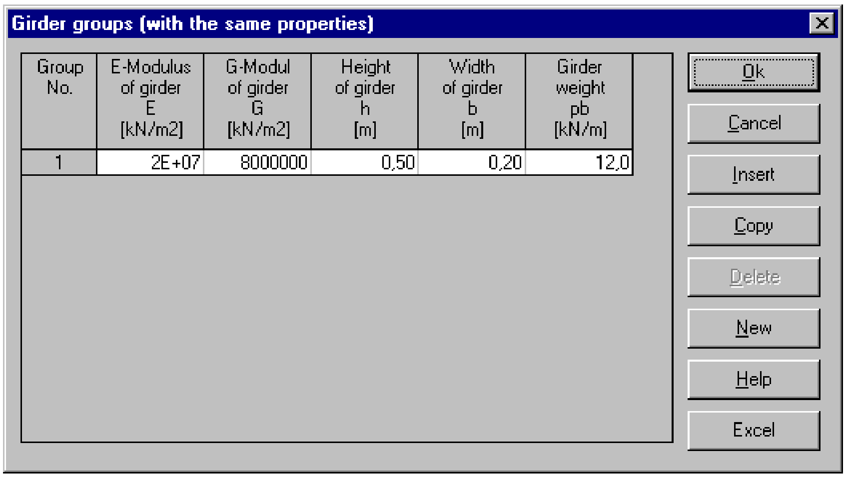


Figure C-57 "Defining girder groups – Rectangular cross section" Dialog box

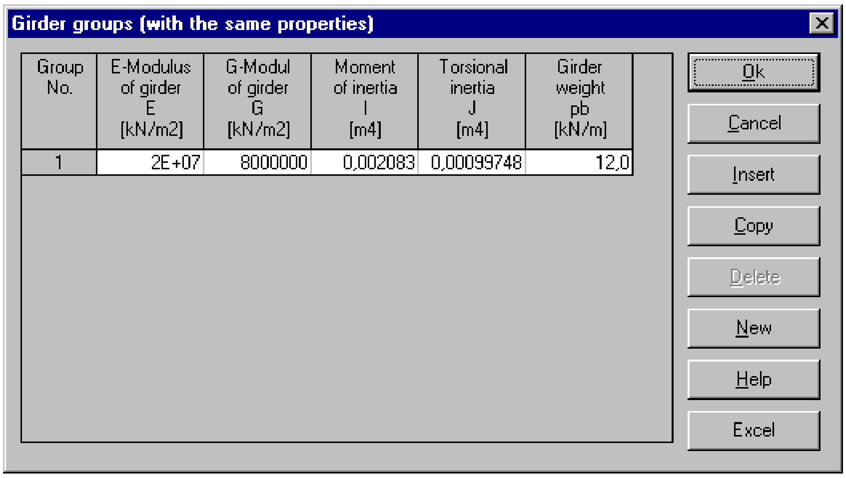


Figure C-58 "Defining girder groups – General cross section" Dialog box

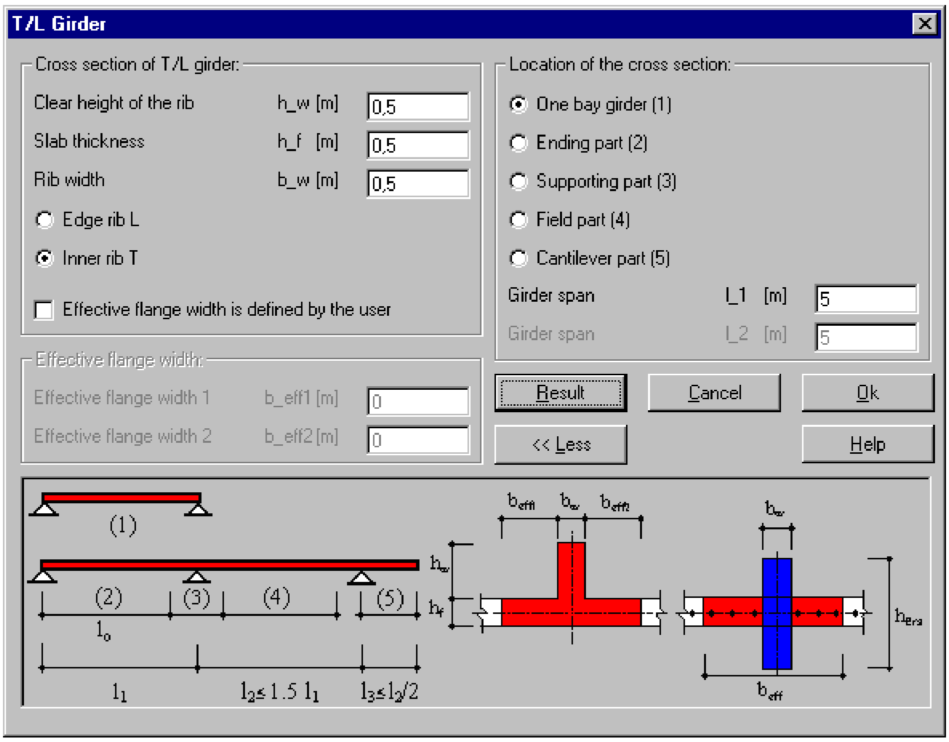
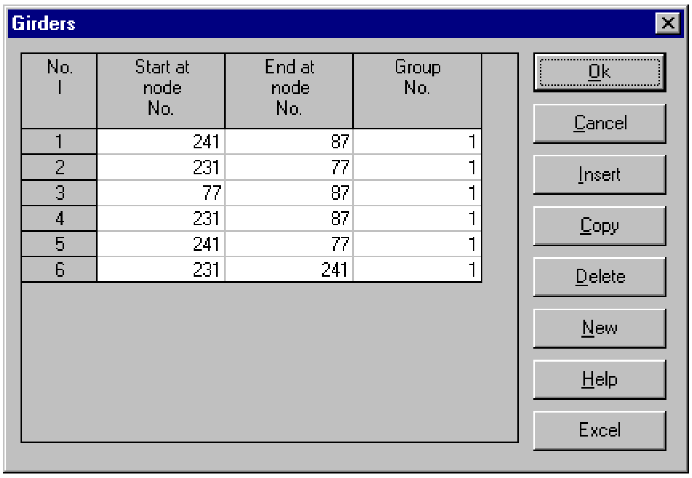


Figure C-59 "Defining girder groups – T/L-Girder" Dialog box

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

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

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



No. I	Start at node No.	End at node No.	Group No.
1	241	87	1
2	231	77	1
3	77	87	1
4	231	87	1
5	241	77	1
6	231	241	1

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

5.4.5 Options Menu (See paragraph 5.3.6)

5.4.6 Format Menu (See paragraph 5.3.7)

5.4.7 Window Menu (See paragraph 5.3.8)

5.4.8 Help Menu (See paragraph 5.3.9)

5.5 Data Menu–"Spring supports" command

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

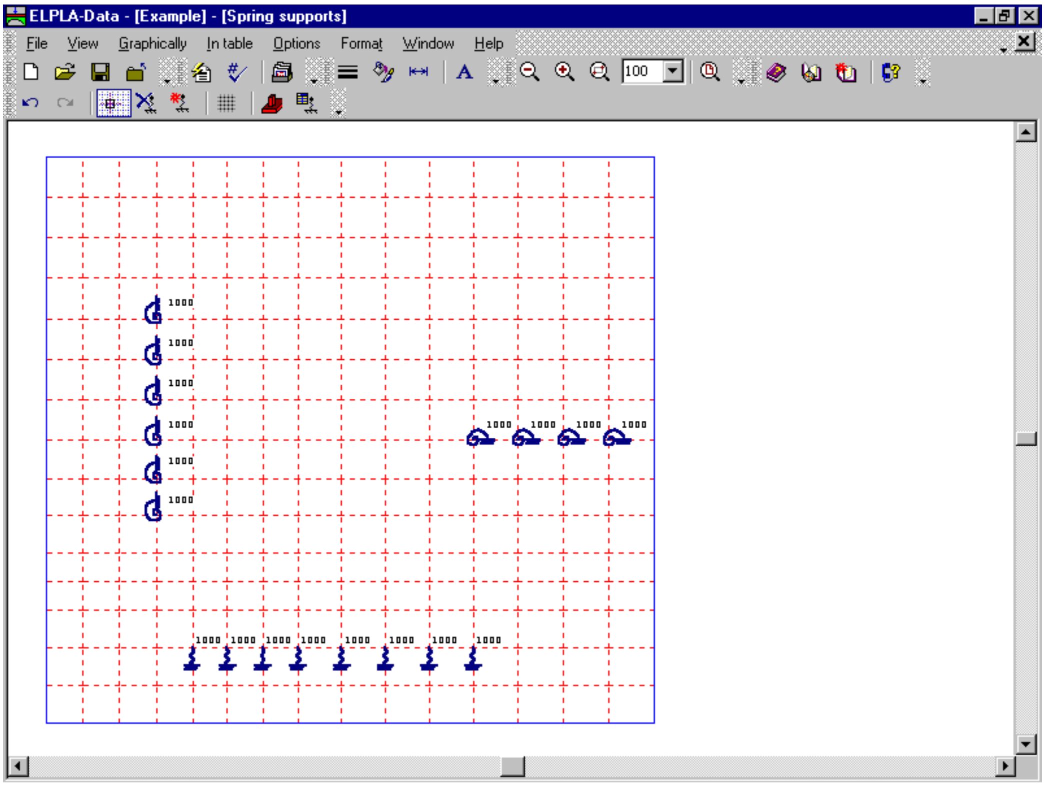


Figure C-61 "Spring supports" embedded program

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

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

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

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

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

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

5.5.1 File Menu

This menu contains five commands:

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

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

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

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

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

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

5.5.2 View Menu (See paragraph 5.3.2)

5.5.3 Graphically Menu

This menu contains the following commands:

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

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

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

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

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

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

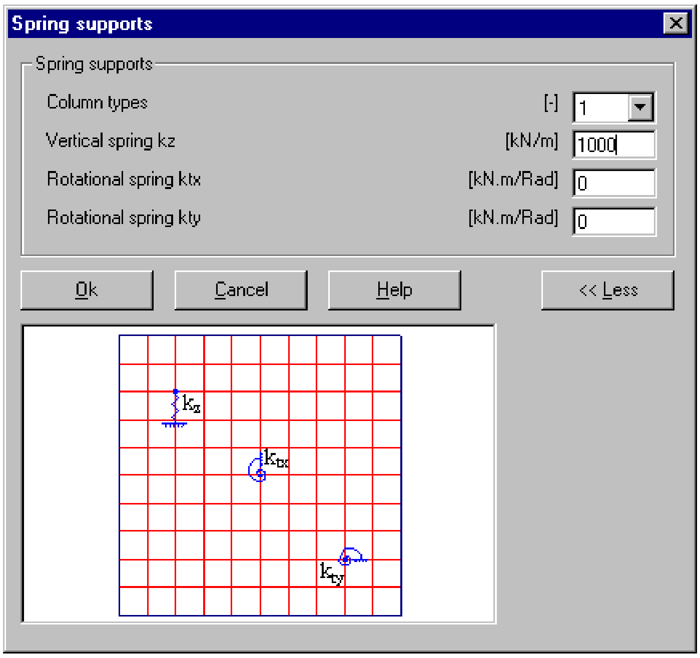


Figure C-62 "Add spring supports" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

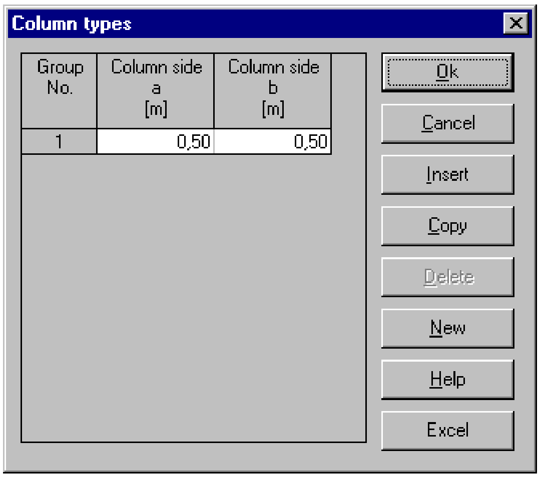
5.5.4 In table Menu

This menu contains the following commands:

- Column types
- Spring supports

In table Menu–"Column types" command

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

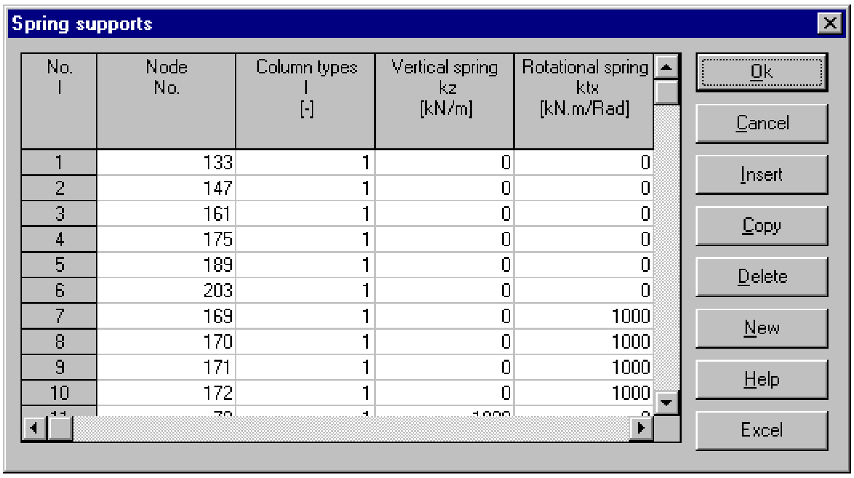


Group No.	Column side a [m]	Column side b [m]
1	0,50	0,50

Figure C-63 Defining column dimensions

In table Menu–"Spring supports" command

When this command is chosen, the following Table in Figure C-64 appears. In this Table the elastic support is described by spring stiffness.



No. I	Node No.	Column types I [-]	Vertical spring kz [kN/m]	Rotational spring ktx [kN.m/Rad]
1	133	1	0	0
2	147	1	0	0
3	161	1	0	0
4	175	1	0	0
5	189	1	0	0
6	203	1	0	0
7	169	1	0	1000
8	170	1	0	1000
9	171	1	0	1000
10	172	1	0	1000

Figure C-64 Defining spring stiffness in a table

5.5.5 Options Menu (See paragraph 5.3.6)

5.5.6 Format Menu (See paragraph 5.3.7)

5.5.7 Window Menu (See paragraph 5.3.8)

5.5.8 Help Menu (See paragraph 5.3.9)

5.6 Data Menu–"Supports/ Boundary conditions" command

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

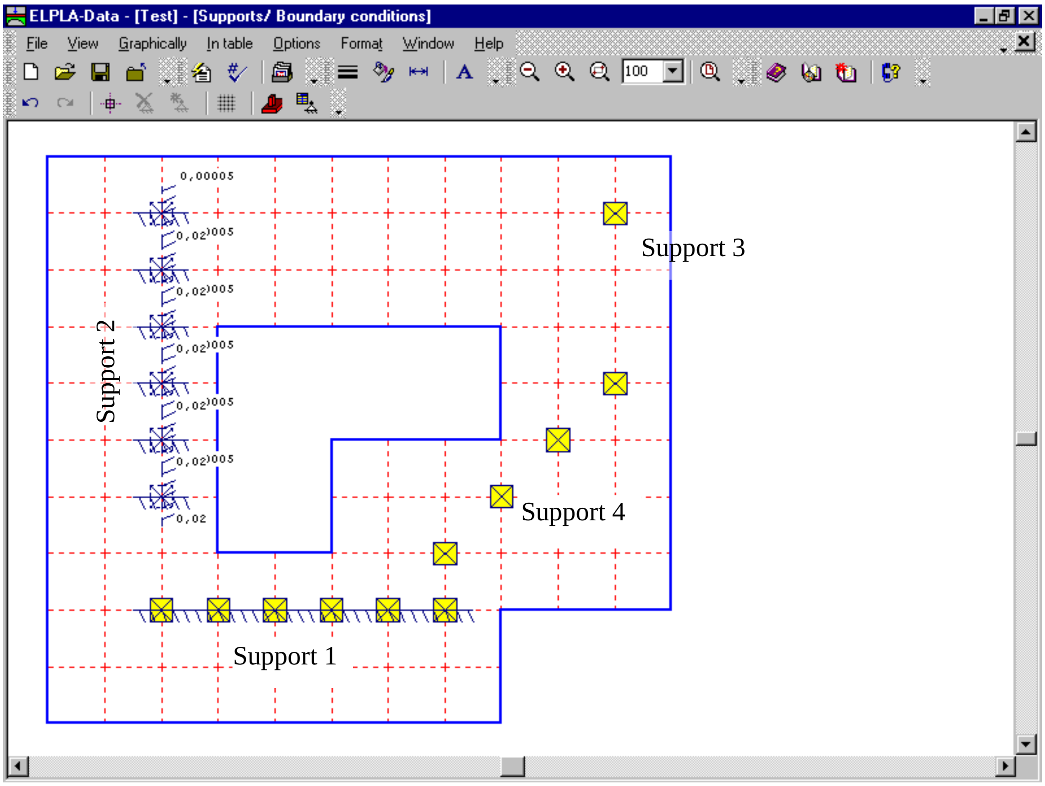


Figure C-65 "Supports/ Boundary conditions" embedded program

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

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

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

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

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

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

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

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

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

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

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

5.6.1 File Menu

This menu contains five commands:

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

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

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

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

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

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

5.6.2 View Menu (See paragraph 5.3.2)

5.6.3 Graphically Menu

This menu contains the following commands:

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

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

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

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

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

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

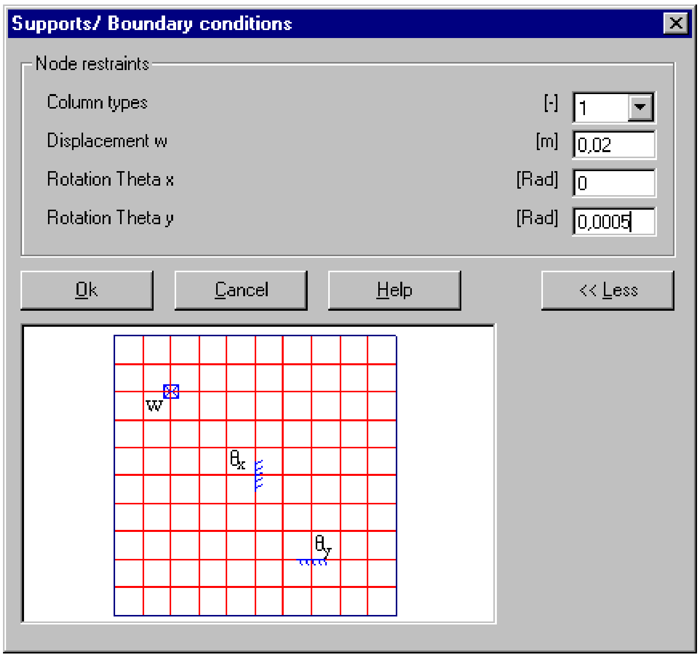


Figure C-66 Defining node restraints

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

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

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.6.4 In table Menu

This menu contains the following commands:

- Column types
- Node restraints

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

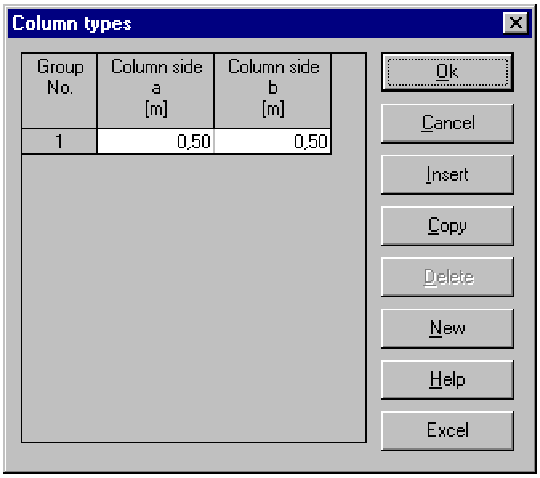


Figure C-67 Defining column dimensions

In table Menu–"Node restraints" command

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

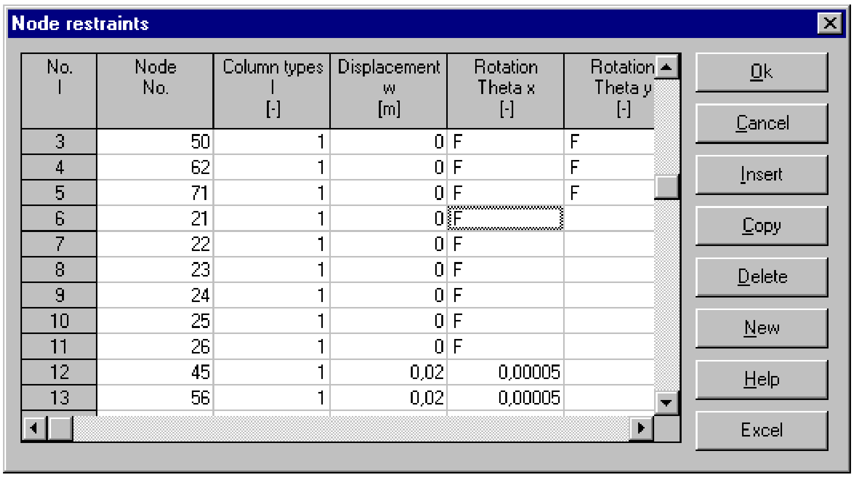


Figure C-68 Defining node restraints in a table

5.6.5 Options Menu (See paragraph 5.3.6)

5.6.6 Format Menu (See paragraph 5.3.7)

5.6.7 Window Menu (See paragraph 5.3.8)

5.6.8 Help Menu (See paragraph 5.3.9)

5.7 Data Menu–"Piles" command

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

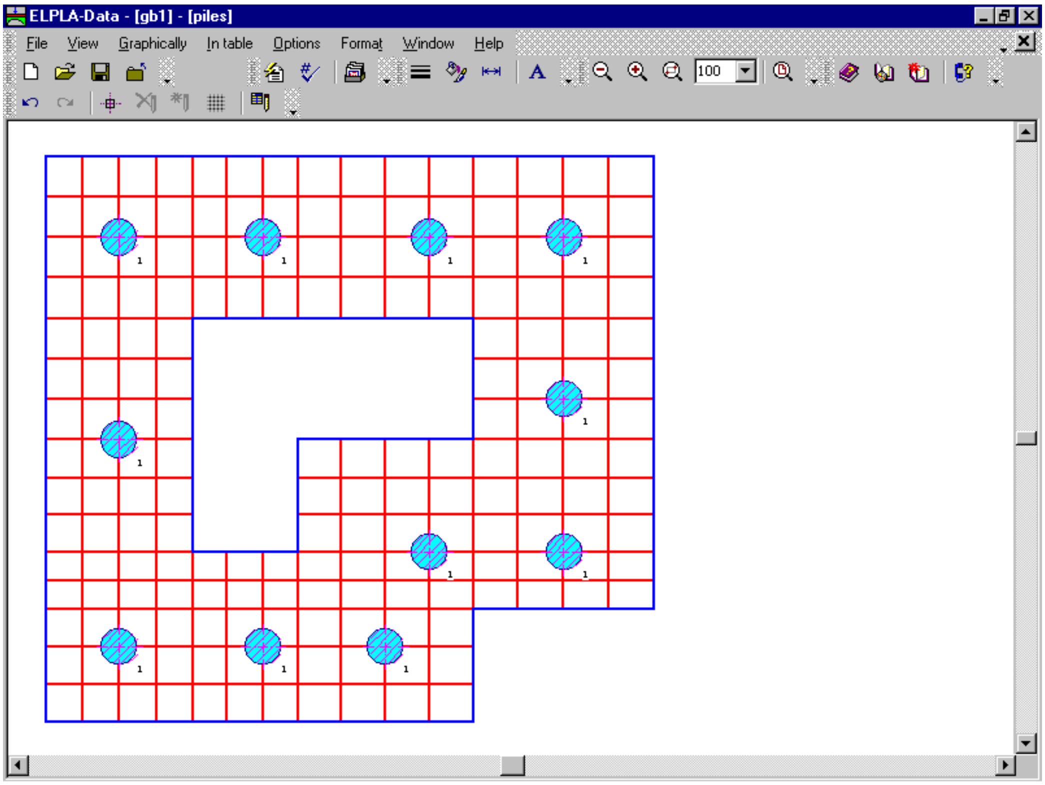


Figure C-69 "Piles" embedded program

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

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

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

5.7.1 File Menu

This menu contains five commands:

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

File Menu–"New piles" command
Defines new piles

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

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

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

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

5.7.2 View Menu (See paragraph 5.3.2)

5.7.3 Graphically Menu

This menu contains the following commands:

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

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

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

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

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

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

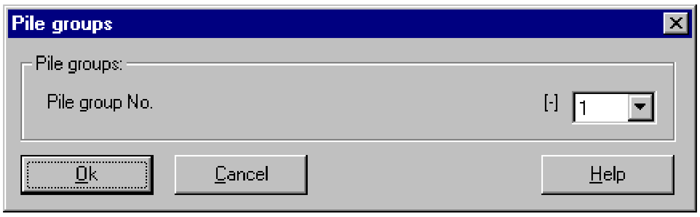


Figure C-70 "Add piles" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.7.4 In table Menu

This menu contains the following commands:

- Pile groups
- Pile location and groups
- Pile material

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

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

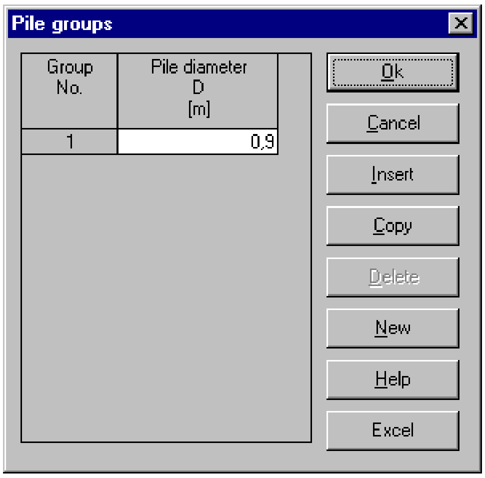


Figure C-71 Defining pile groups for Simple Assumption Model

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

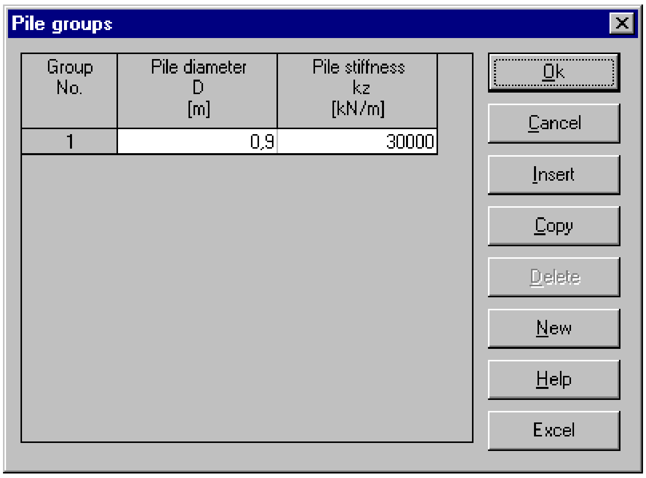


Figure C-72 Defining pile groups for *Winkler*'s Model

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

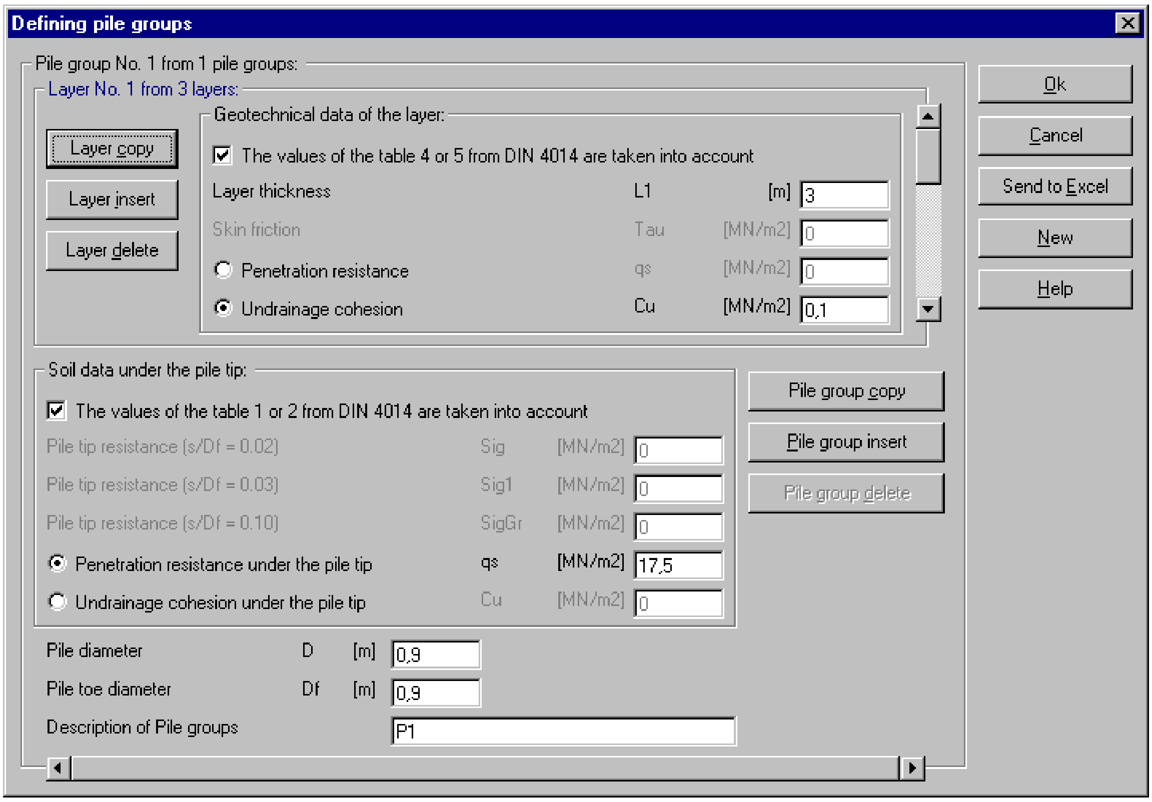
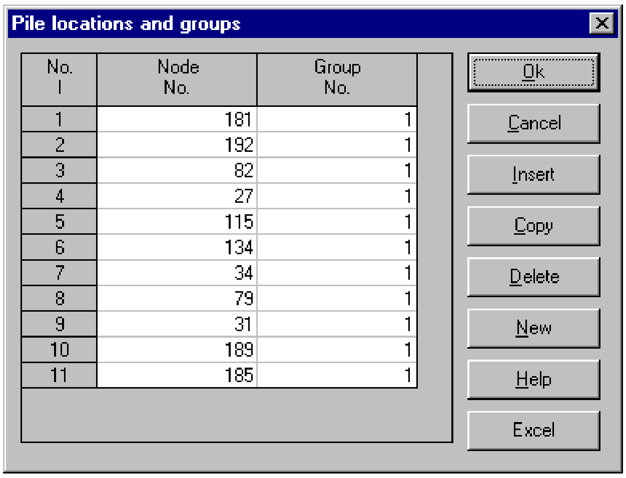


Figure C-73 Defining pile groups for Half-Space and Layered soil Models

In table Menu–"Pile locations and groups" command

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



No. I	Node No.	Group No.
1	181	1
2	192	1
3	82	1
4	27	1
5	115	1
6	134	1
7	34	1
8	79	1
9	31	1
10	189	1
11	185	1

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

In table Menu–"Pile material" command

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

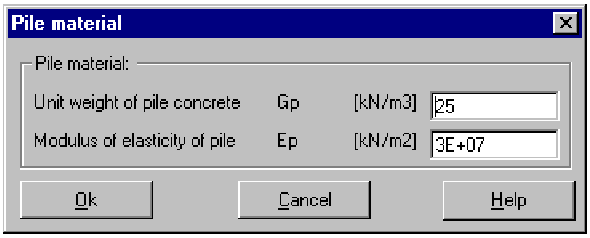


Figure C-75 "Defining element groups" Dialog box

5.7.5 Options Menu (See paragraph 5.3.6)

5.7.6 Format Menu (See paragraph 5.3.7)

5.7.7 Window Menu (See paragraph 5.3.8)

5.7.8 Help Menu (See paragraph 5.3.9)

5.8 Data Menu–"Soil properties" command

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

Soil properties for Simple Assumption Model

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

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

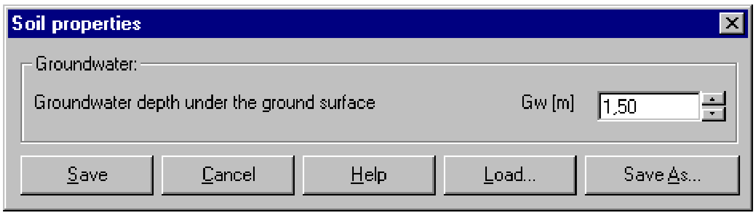


Figure C-76 "Soil properties" Dialog box (method 1)

Soil properties for *Winkler'*s Model

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

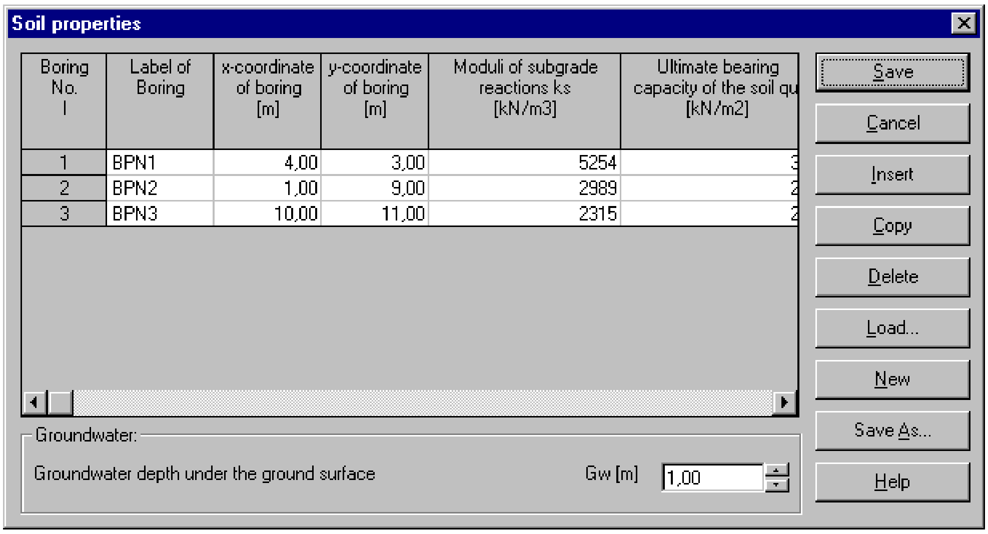


Figure C-77 "Soil properties" Dialog box (methods 2 and 3)

Soil properties for Isotropic Elastic Half-Space Model

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

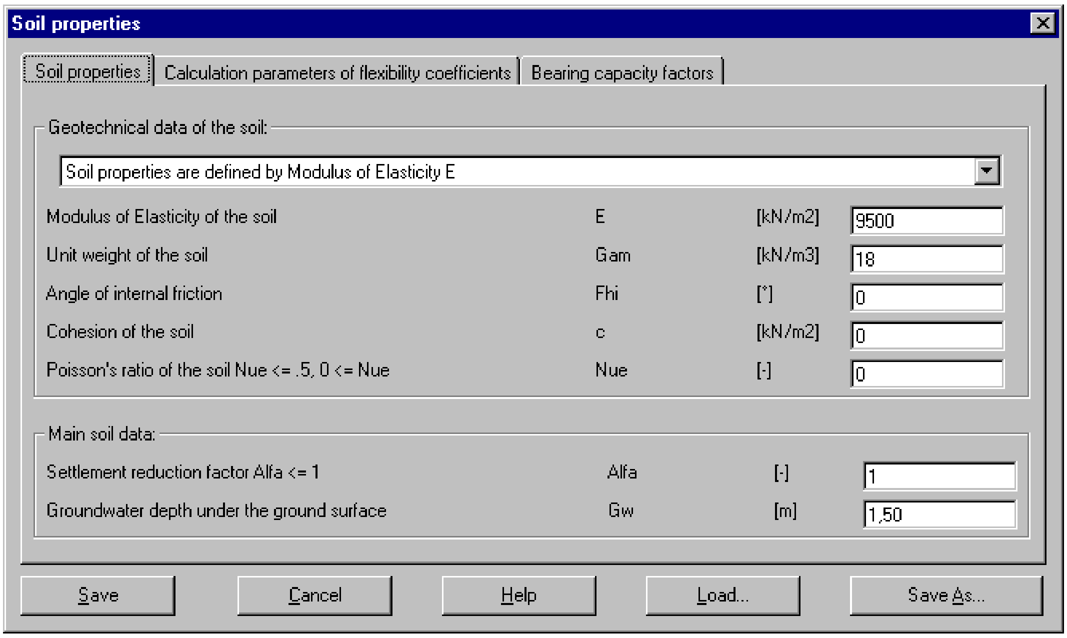


Figure C-78 "Soil properties" Dialog box (methods 2 and 5)

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

Soil properties for Layered soil Model

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

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

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

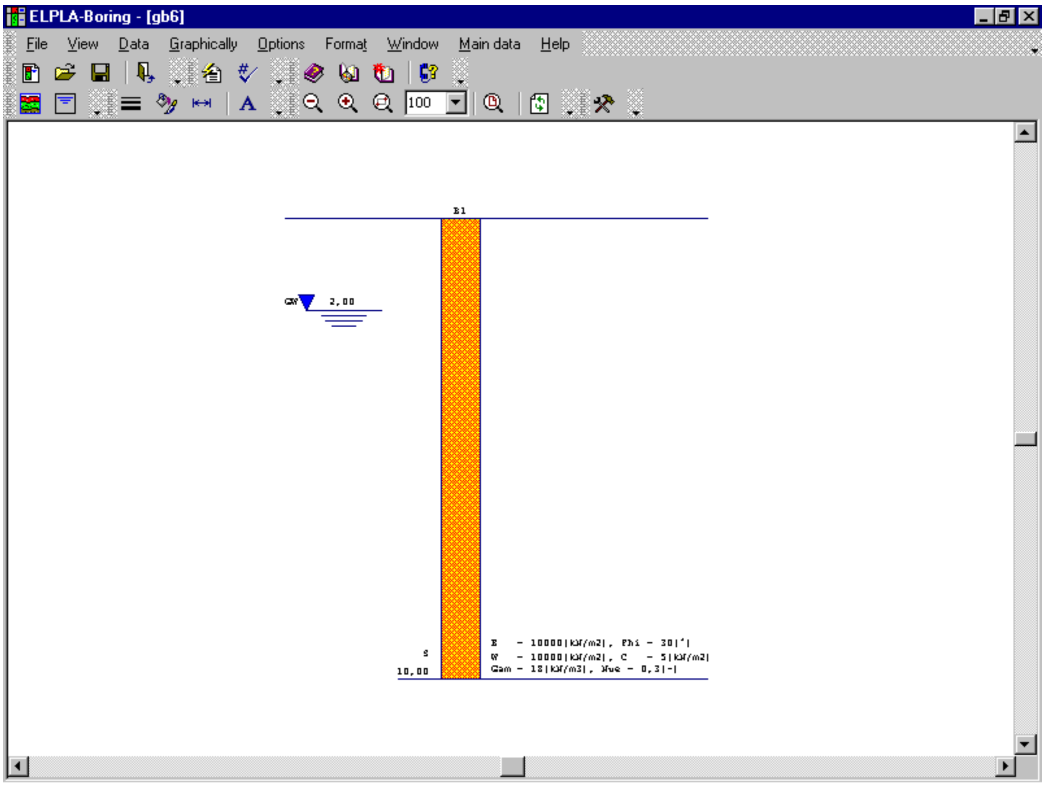


Figure C-79 "Soil properties" embedded program

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

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

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

5.8.1 File Menu

This menu contains five commands:

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

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

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

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

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

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

5.8.2 View Menu (See paragraph 5.3.2)

5.8.3 Data Menu

The menu "Data" contains the following commands:

- Soil data
- Main soil data

Data Menu–"Soil data" command

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

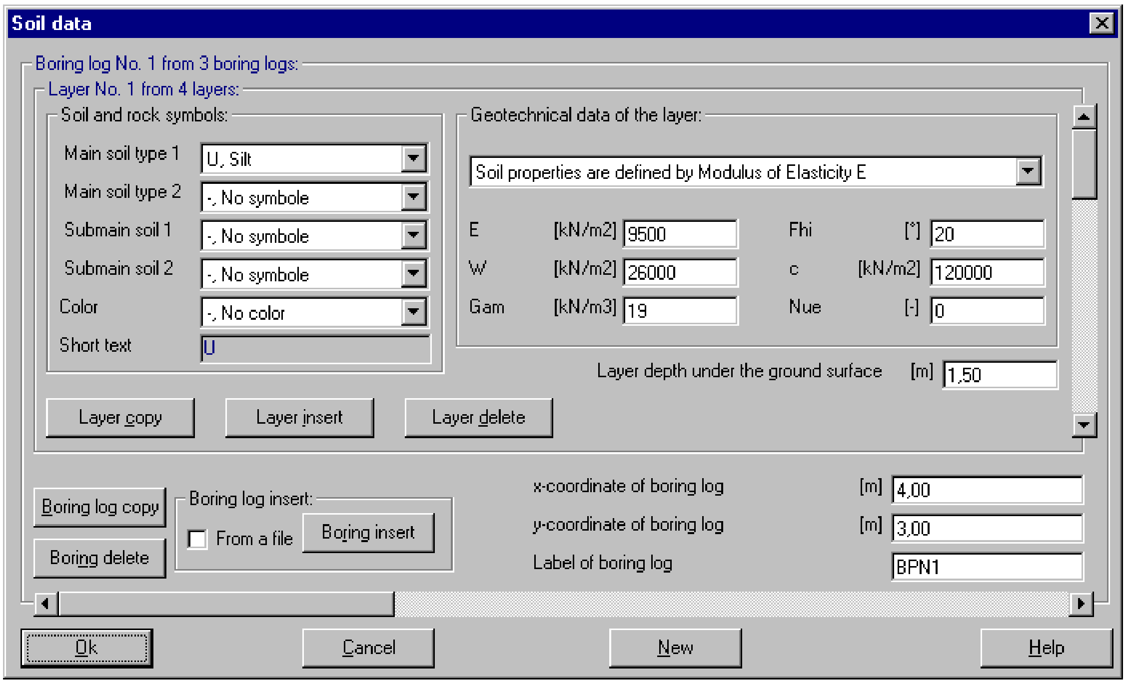


Figure C-80 "Soil data" Dialog box

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

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

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

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

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

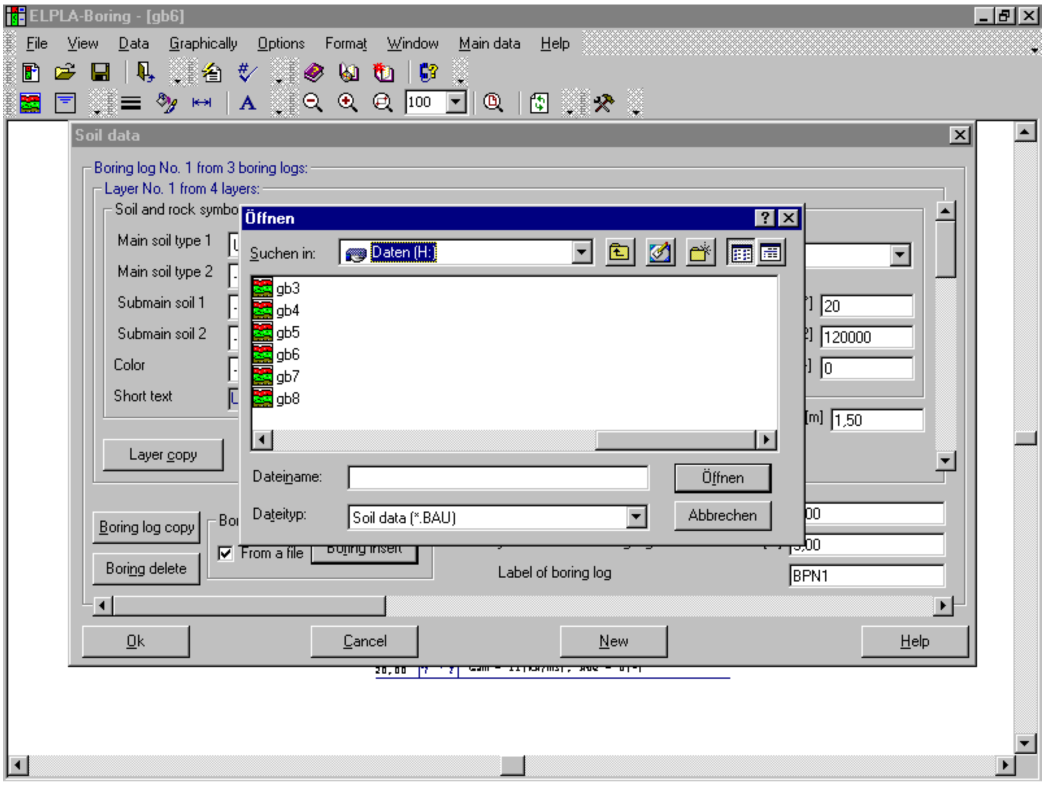


Figure C-81 Boring log can be inserted from a file

Data Menu–"Main soil data" command

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

Settlement reduction factor

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

Sand and silt	$\alpha = 0.66$
Normally and slightly over consolidated clay	$\alpha = 1.0$
Heavily over consolidated clay	$\alpha = 0.5\text{-}1.0$

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

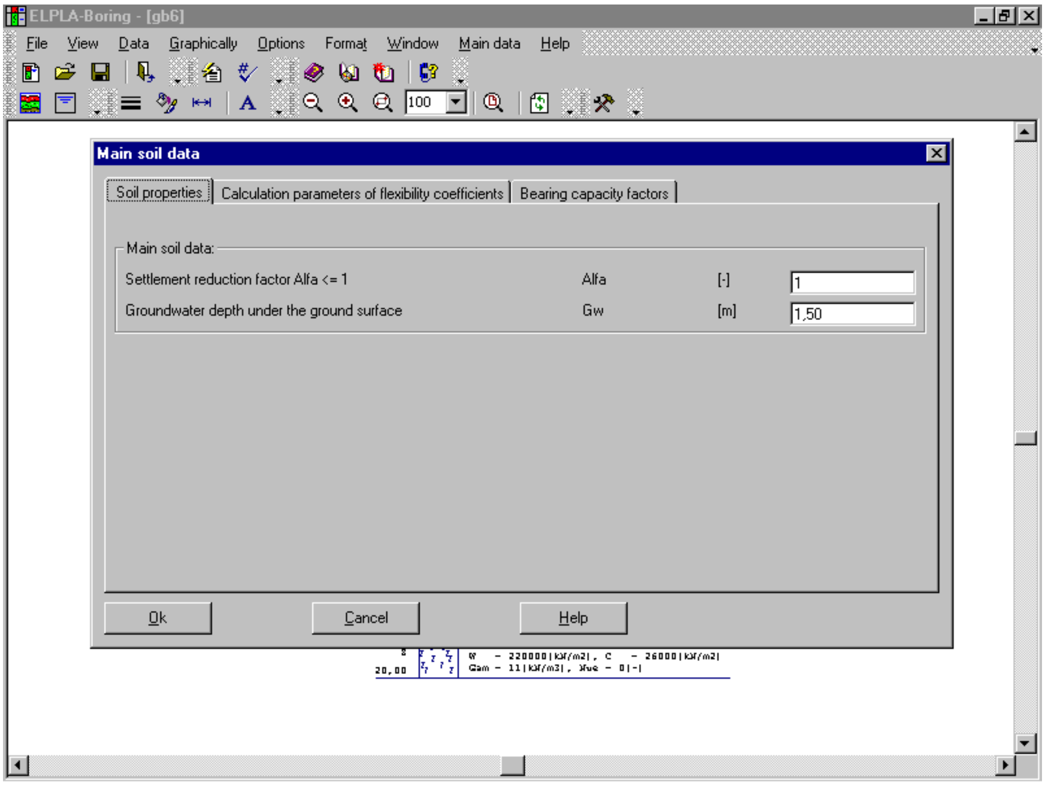


Figure C-82 "Main soil data" dialog box

Bearing capacity factors

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

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

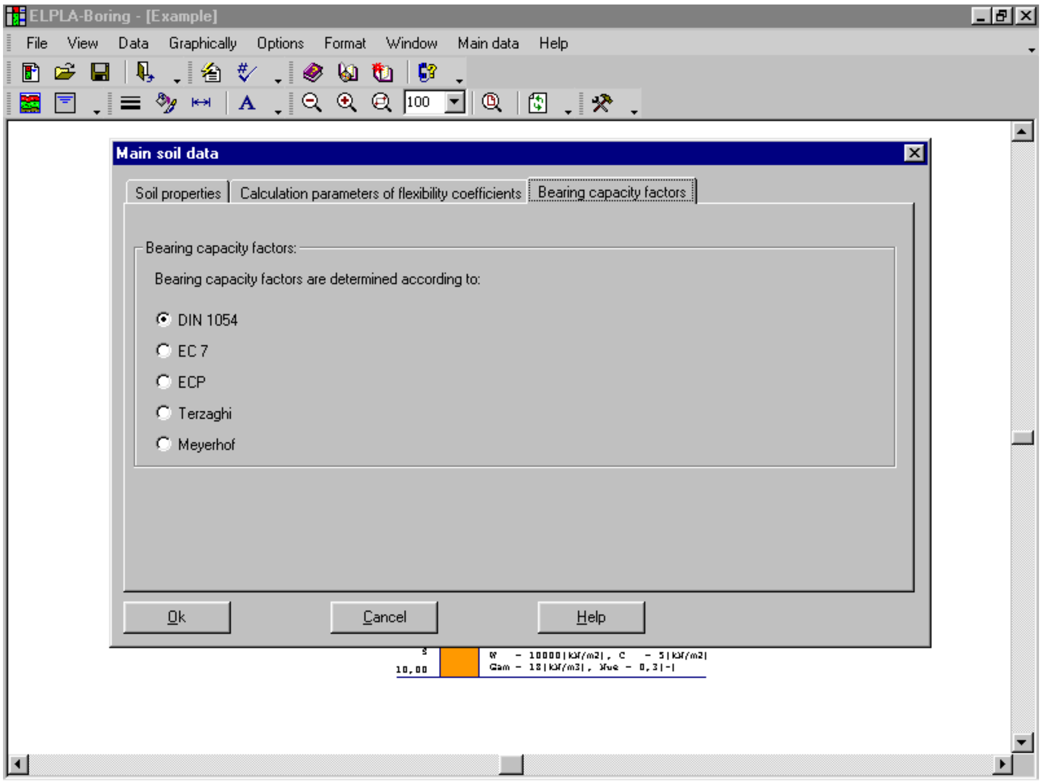


Figure C-83 Menu "Bearing capacity factors"

Flexibility coefficients for interior nodes

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

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

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

Flexibility coefficients for exterior nodes

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

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

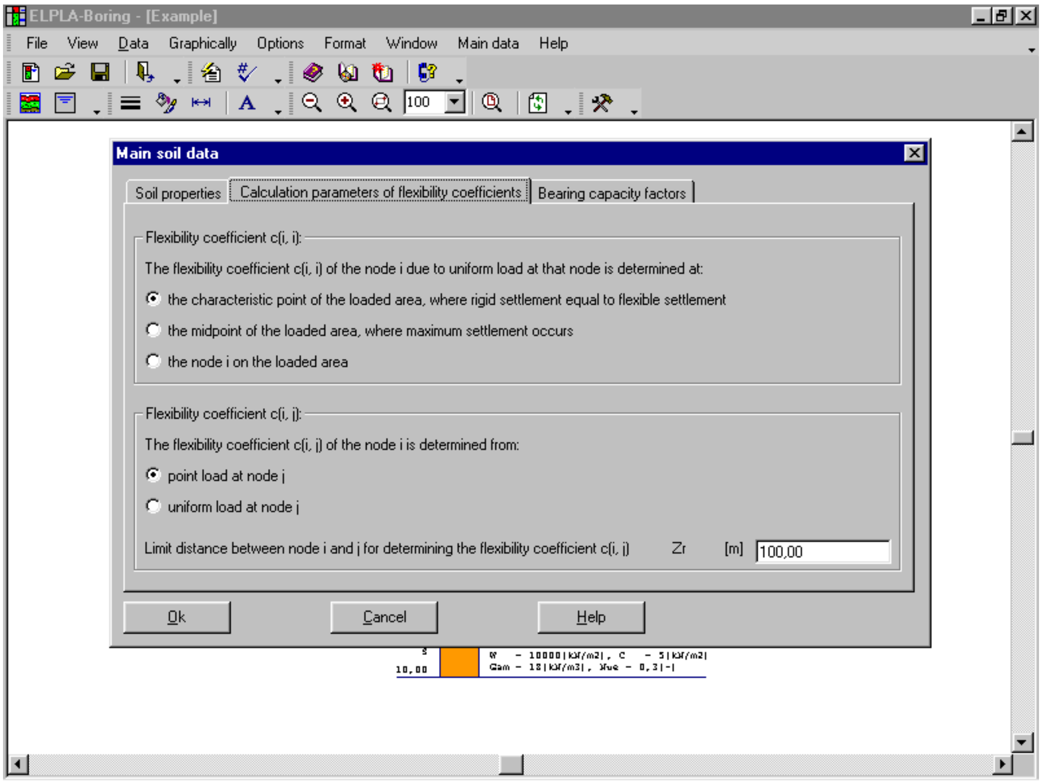


Figure C-84 Menu "Flexibility coefficients"

5.8.4 Graphically Menu

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

The menu "Graphically" contains one command:

- Drawing boring logs

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

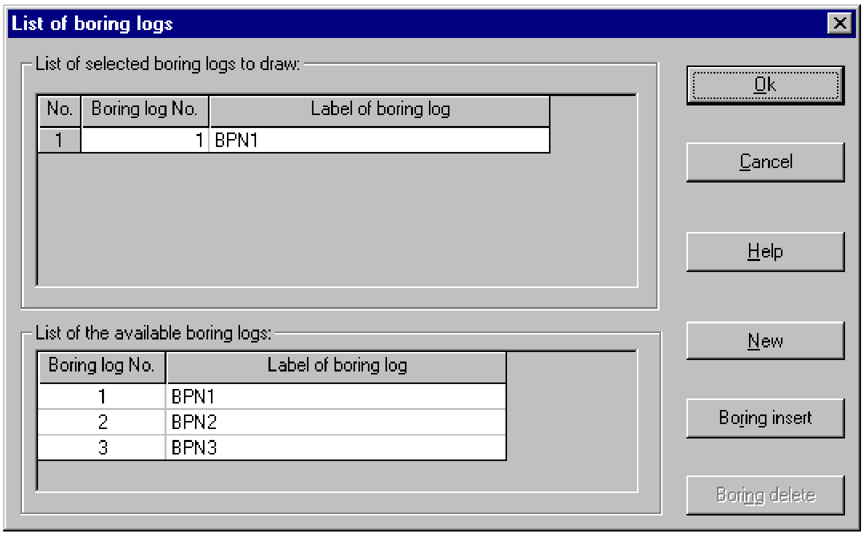


Figure C-85 "List of boring logs" Dialog box

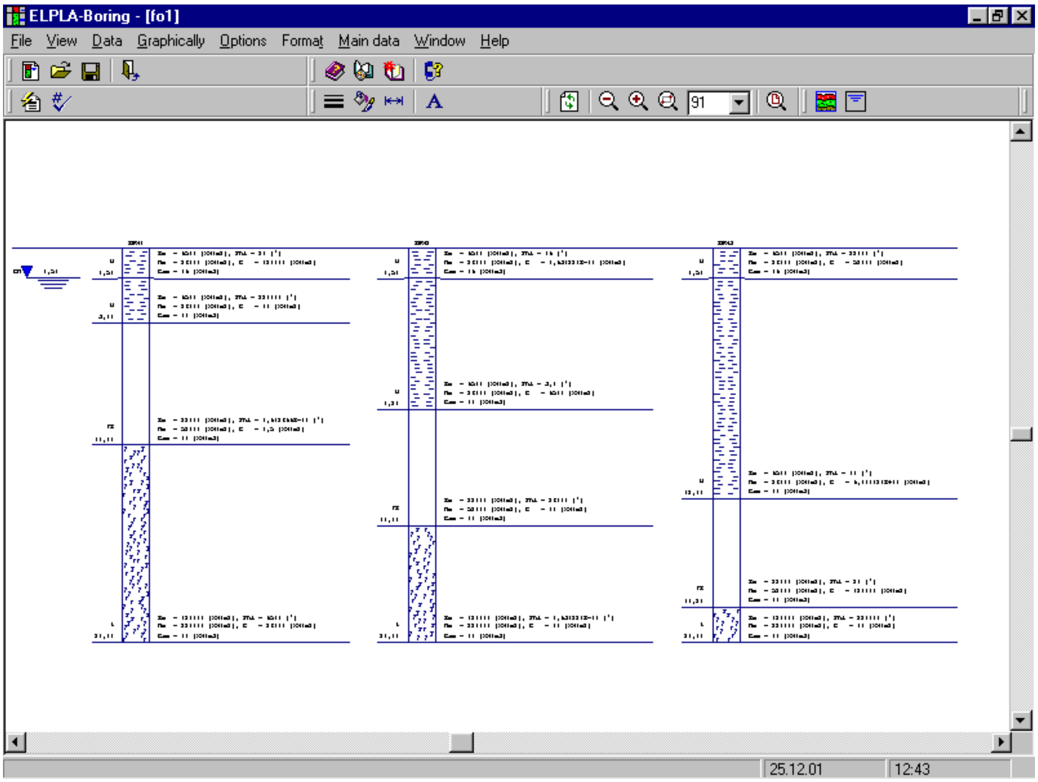


Figure C-86 Boring logs on the screen

Definition of boring logs graphically

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

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

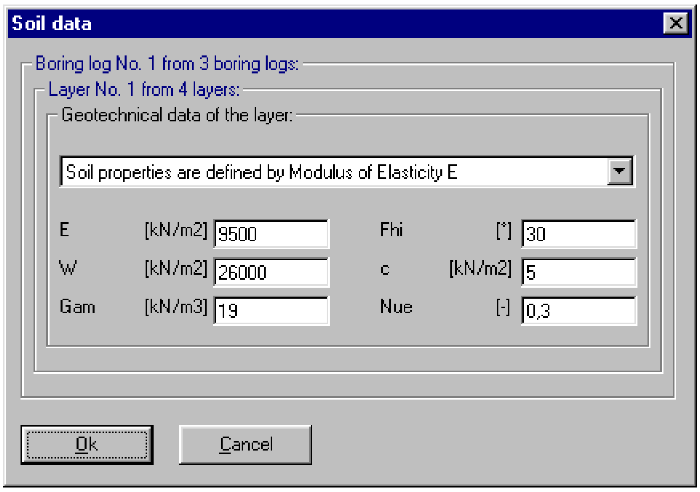


Figure C-87 "Geotechnical data of the layer" Dialog box

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

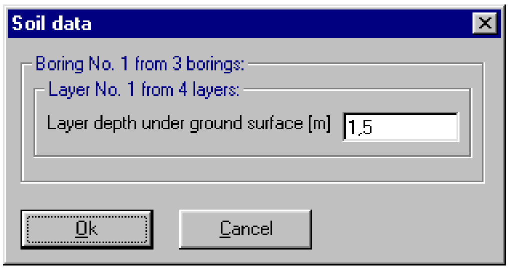


Figure C-88 "Layer depth under the ground surface" Dialog box

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

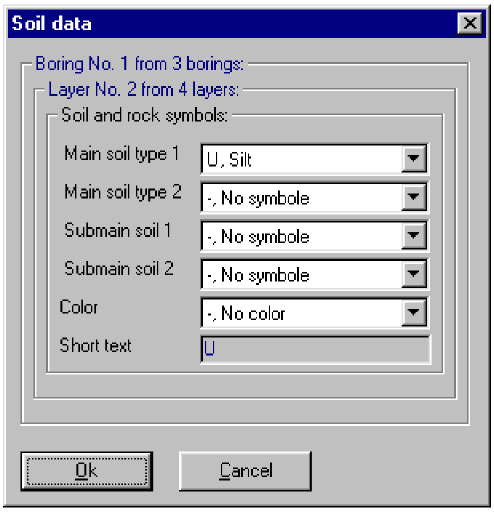


Figure C-89 "Sand and rock symbols" Dialog box

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

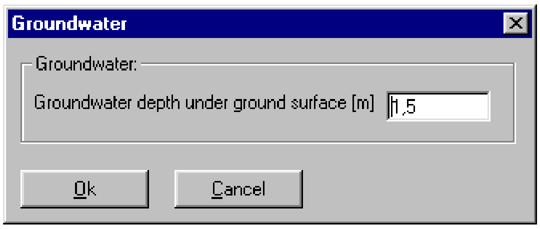


Figure C-90 "Groundwater" Dialog box

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

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

5.8.5 Options Menu

The Options Menu has the following commands:

- Plot parameters
- Display values

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

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

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

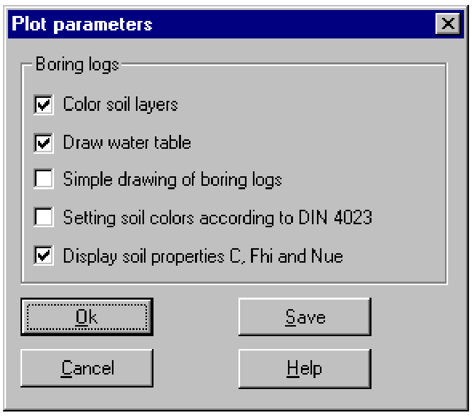


Figure C-92 "Plot parameters" Dialog box

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

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

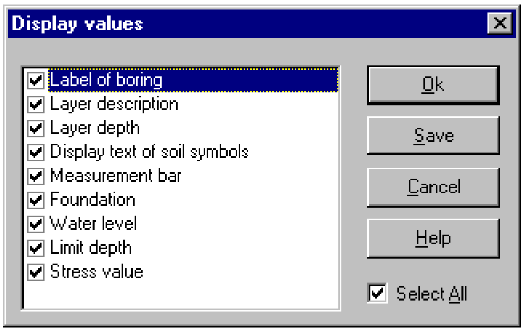


Figure C-93 "Display values" Dialog box

5.8.6 Format Menu

The Format Menu has the following commands:

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

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

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

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

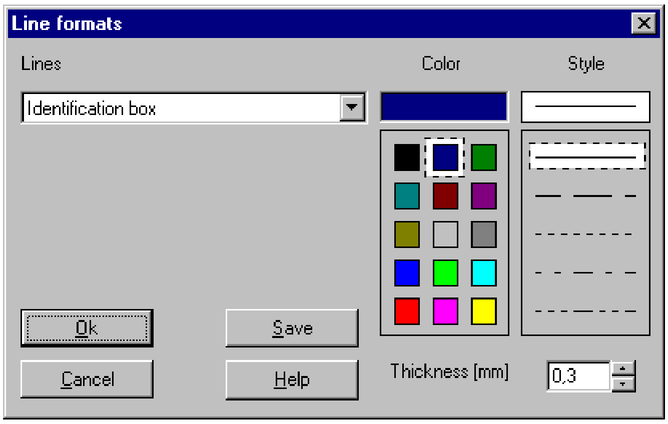


Figure C-94 "Line formats" Dialog box

Format Menu–"Fill color" command

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

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

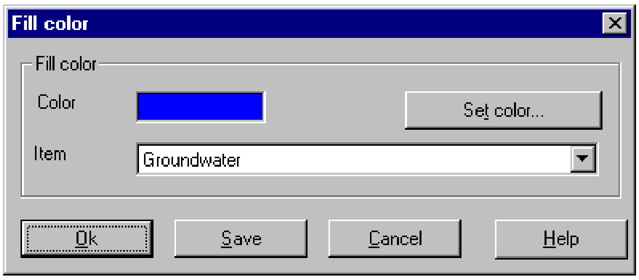


Figure C-95 "Fill color" Dialog box

Format Menu–"Max. width" command

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

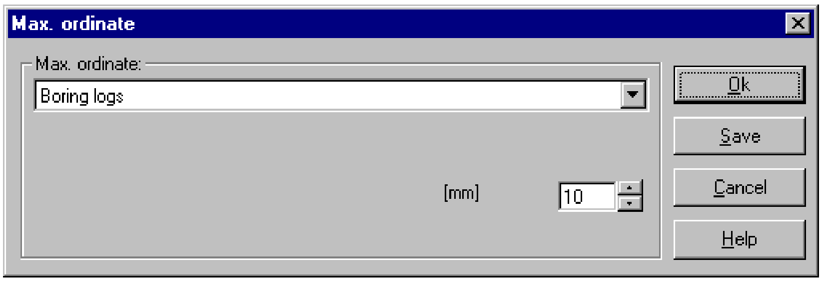


Figure C-96 "Max. width" Dialog box

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

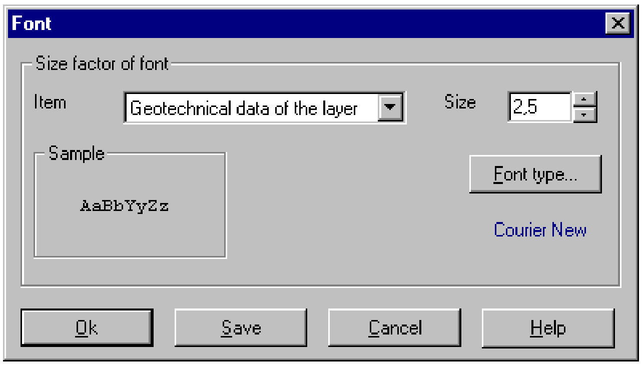


Figure C-97 "Font size" Dialog box

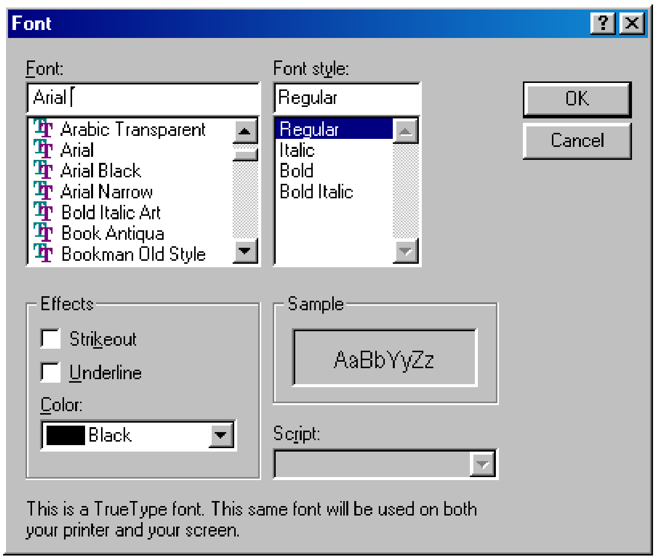


Figure C-98 "Font type" Dialog box

5.8.7 Main data Menu

The Main data Menu has the following command:

- Preferences

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

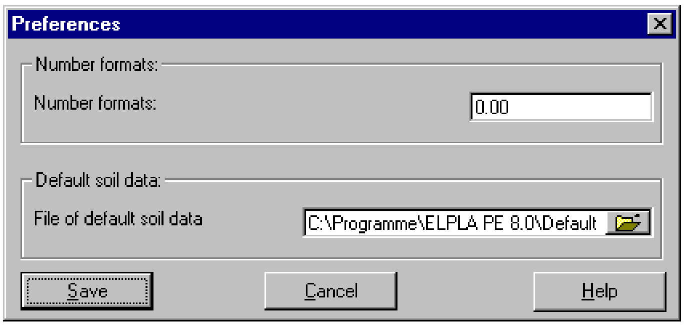


Figure C-99 "Preferences" Dialog box

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

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

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

  The following examples describe the number formats:

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

5.8.8 **Window Menu** (See paragraph 5.3.8)

5.8.9 **Help Menu** (See paragraph 5.3.9)

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

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

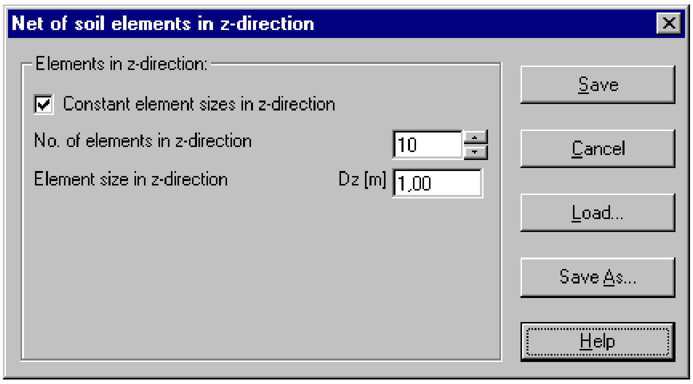


Figure C-100 "Net of soil elements in z-direction" Dialog box

5.10 Data Menu–"Limit depth" command

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

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

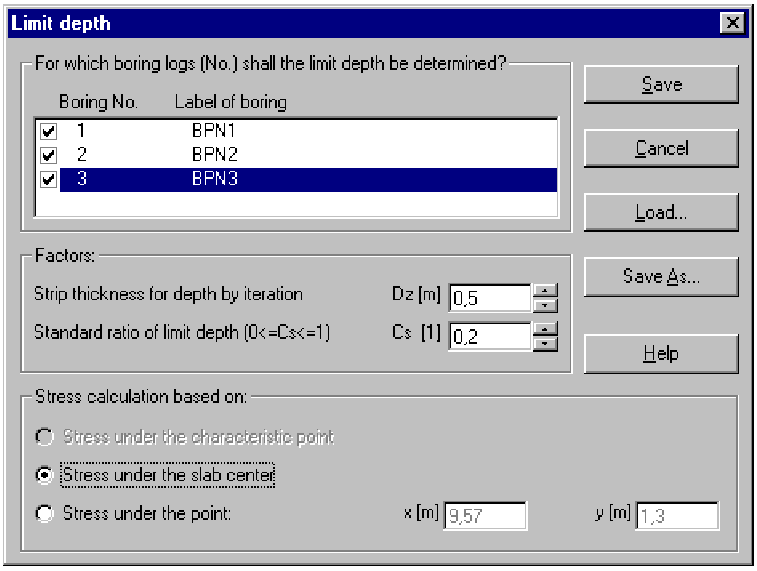


Figure C-101 "Limit depth" Dialog box

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

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

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

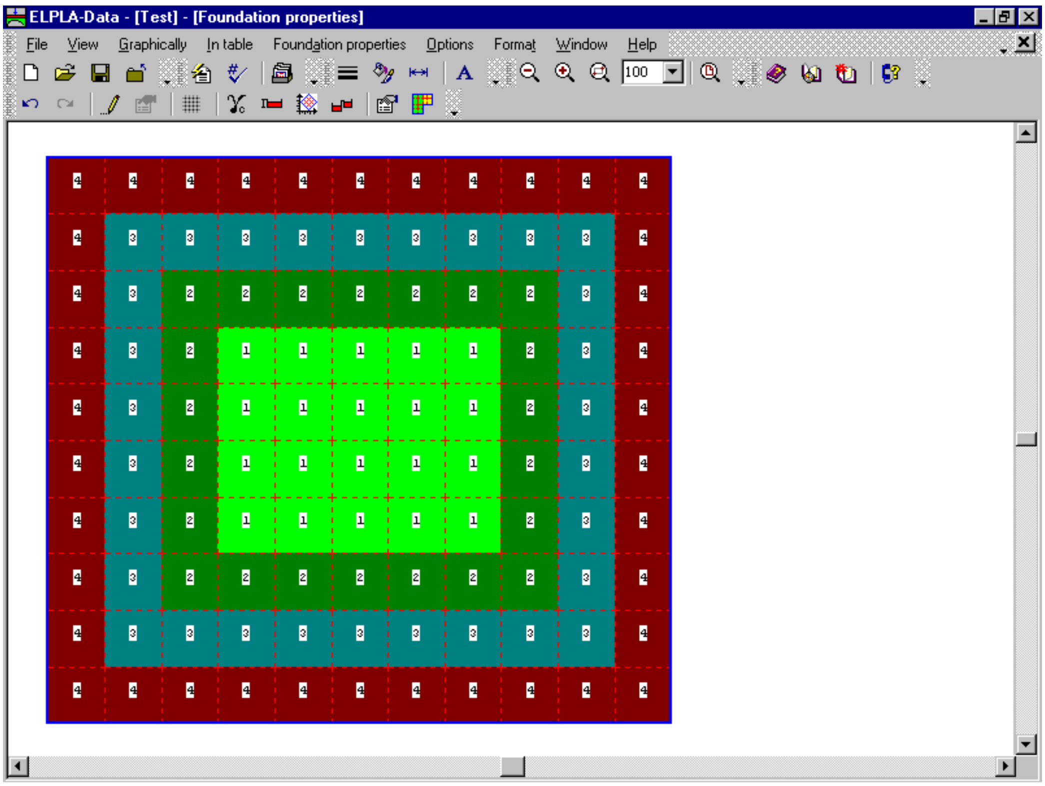


Figure C-102 "Foundation properties" embedded program

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

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

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

5.11.1 File Menu

This menu contains five commands:

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

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

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

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

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

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

5.11.2 View Menu (See paragraph 5.3.2)

5.11.3 Graphically Menu

The menu "Graphically" contains the following commands:

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

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

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

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

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

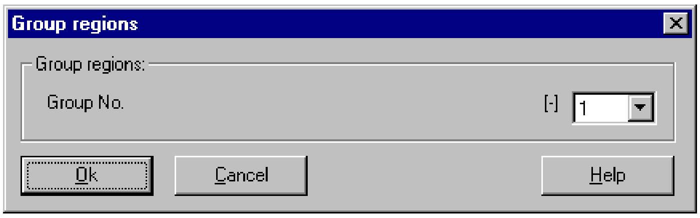


Figure C-103 Defining group No.

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.11.4 In table Menu

The menu "In table" contains the following commands:

- Element groups
- Group regions

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

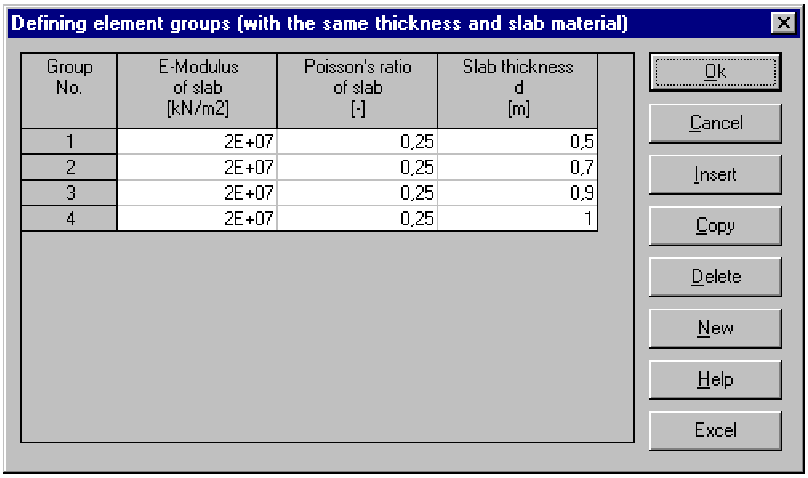


Figure C-104 "Defining element groups" Dialog box

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

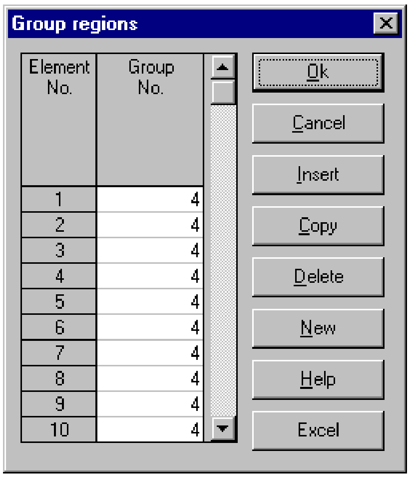


Figure C-105 "Group regions" Dialog box

5.11.5 Foundation (Slab) properties Menu

The menu "Foundation properties" contains the following commands:

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

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

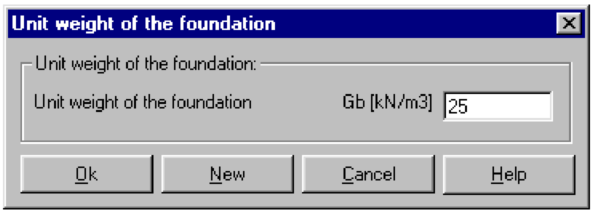


Figure C-106 "Unit weight of the foundation" Dialog box

Foundation properties Menu–"Foundation depth" command

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

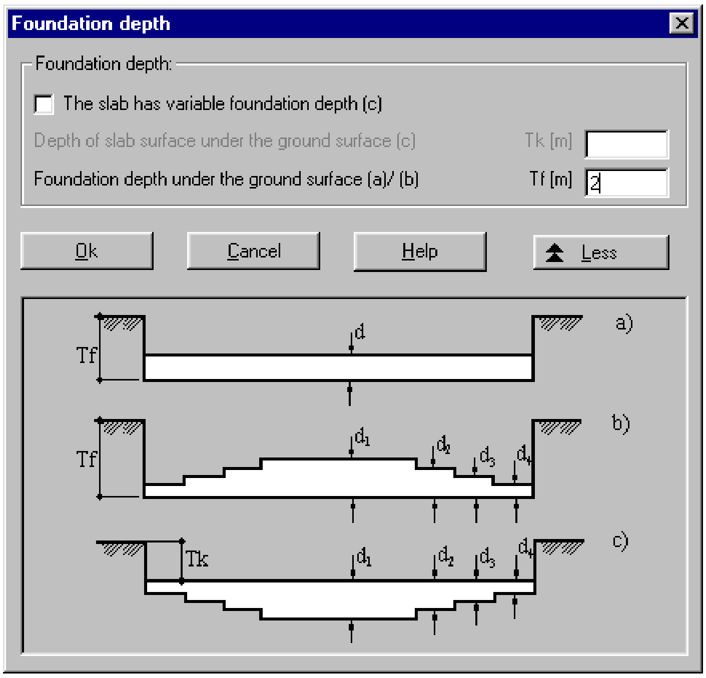


Figure C-107 "Foundation depth" Dialog box

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

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

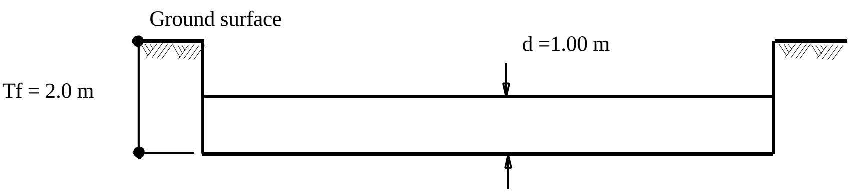


Figure C-108 The slab thickness d is constant for the entire slab

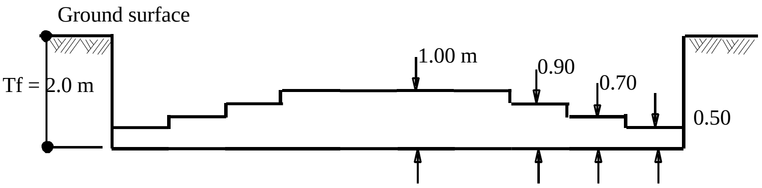


Figure C-109 Variable slab thickness with constant foundation level

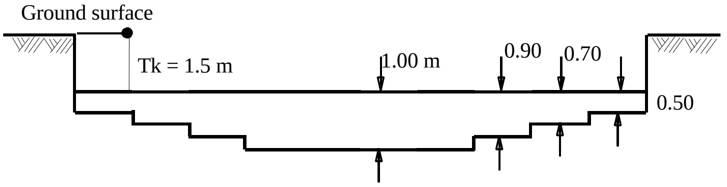


Figure C-110 Variable slab thickness with variable foundation level

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

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

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

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

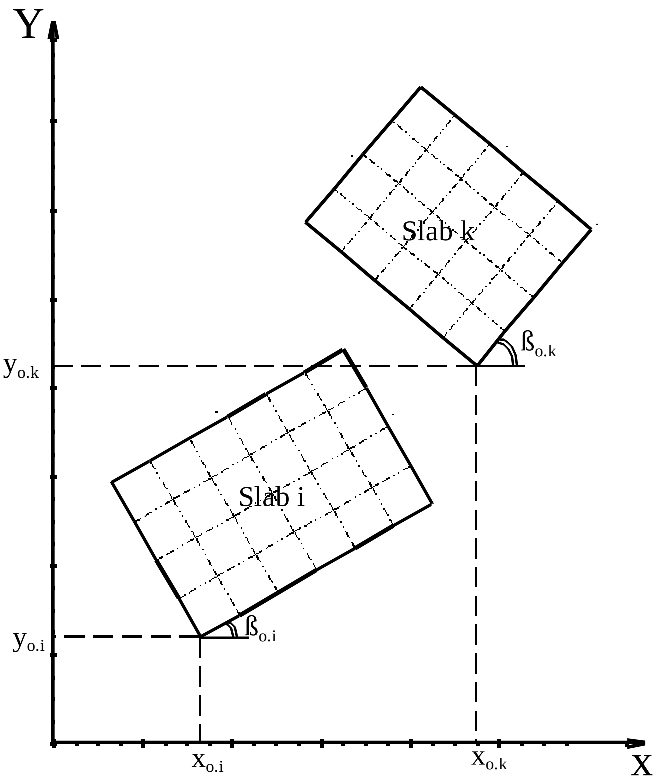


Figure C-111 Geometrical plan by studding the influence of slab k on the slab i

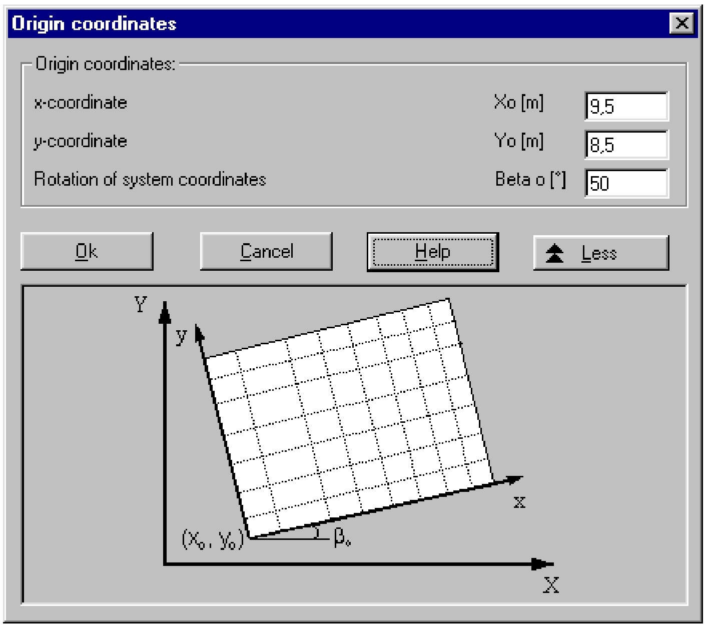


Figure C-112 "Origin coordinates" Dialog box

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

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

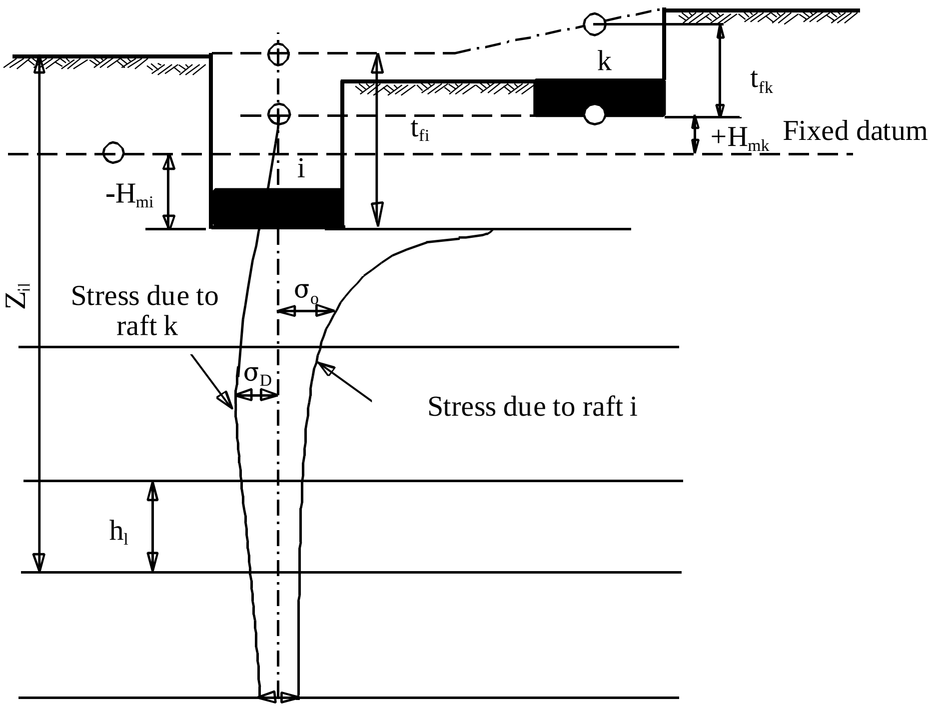


Figure C-113 Influence of slab k on the slab i

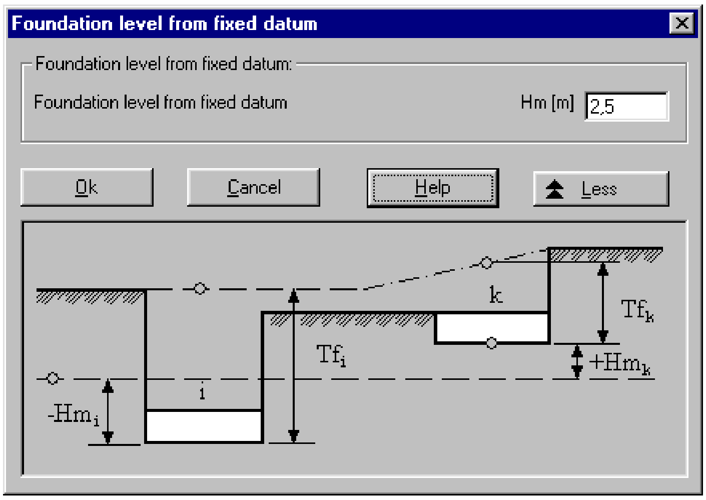


Figure C-114 "Foundation level from a fixed datum" Dialog box

5.11.6 Options Menu (See paragraph 5.3.6)

5.11.7 Format Menu (See paragraph 5.3.7)

5.11.8 Window Menu (See paragraph 5.3.8)

5.11.9 Help Menu (See paragraph 5.3.9)

5.12 Data Menu–"Reinforcement data" command

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

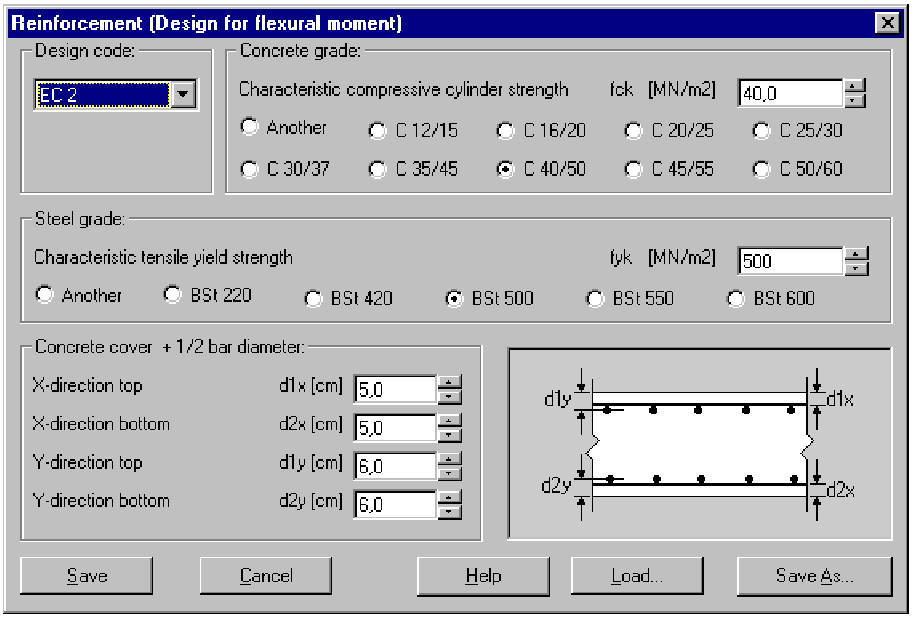


Figure C-115 "Reinforcement" dialog box

5.13 Data Menu–"Boring fields" command

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

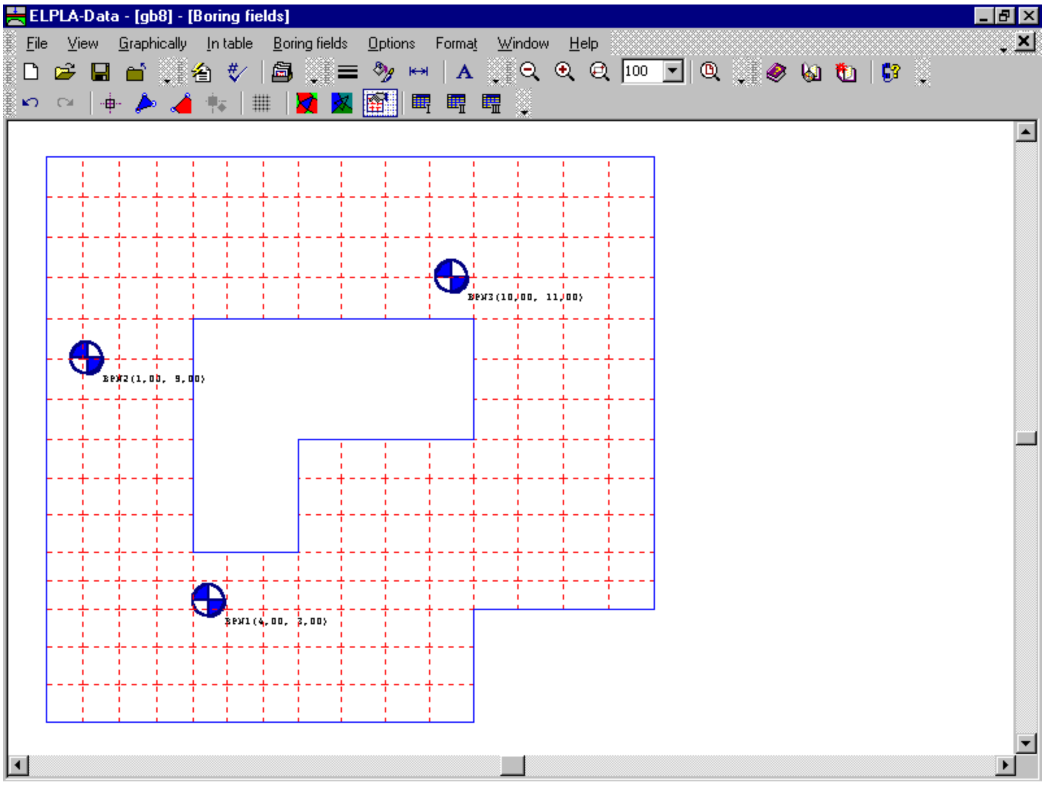


Figure C-116 "Boring fields" embedded program

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

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

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

5.13.1 File Menu

This menu contains five commands:

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

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

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

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

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

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

5.13.2 View Menu (See paragraph 5.3.2)

5.13.3 Graphically Menu

This menu contains the following commands:

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

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

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

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

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

Graphically Menu–"Zone type I" command

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

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

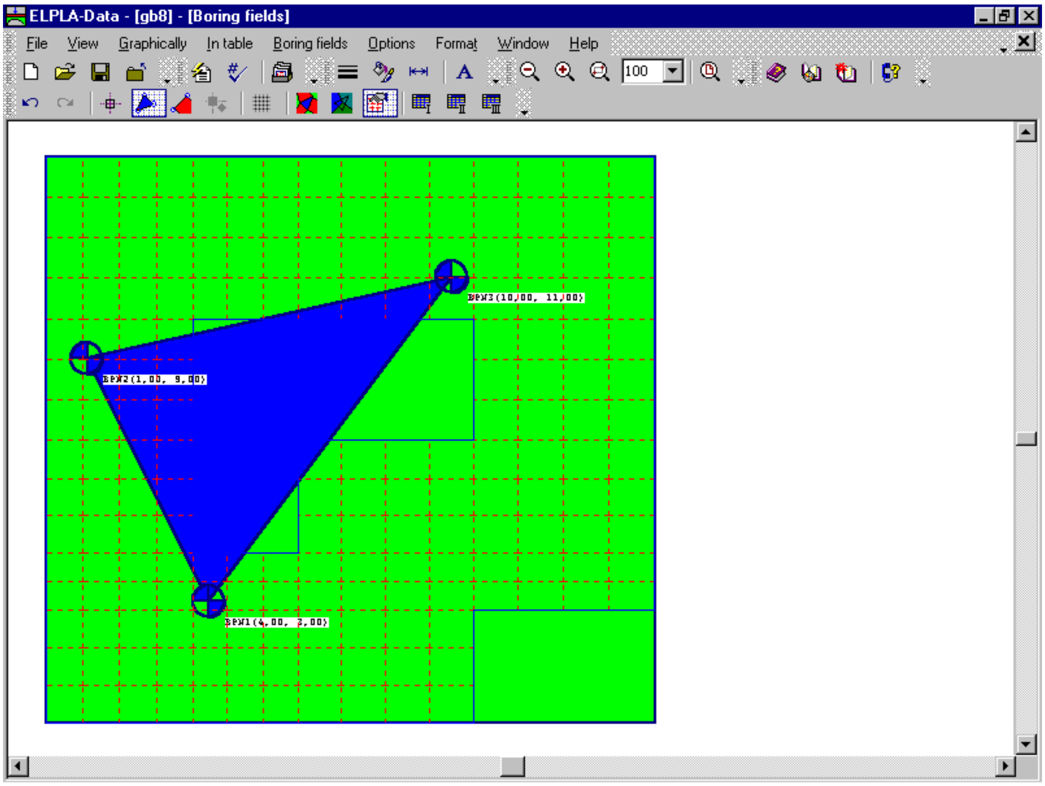


Figure C-117 Zone type I

Graphically Menu–"Zone type II" command

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

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

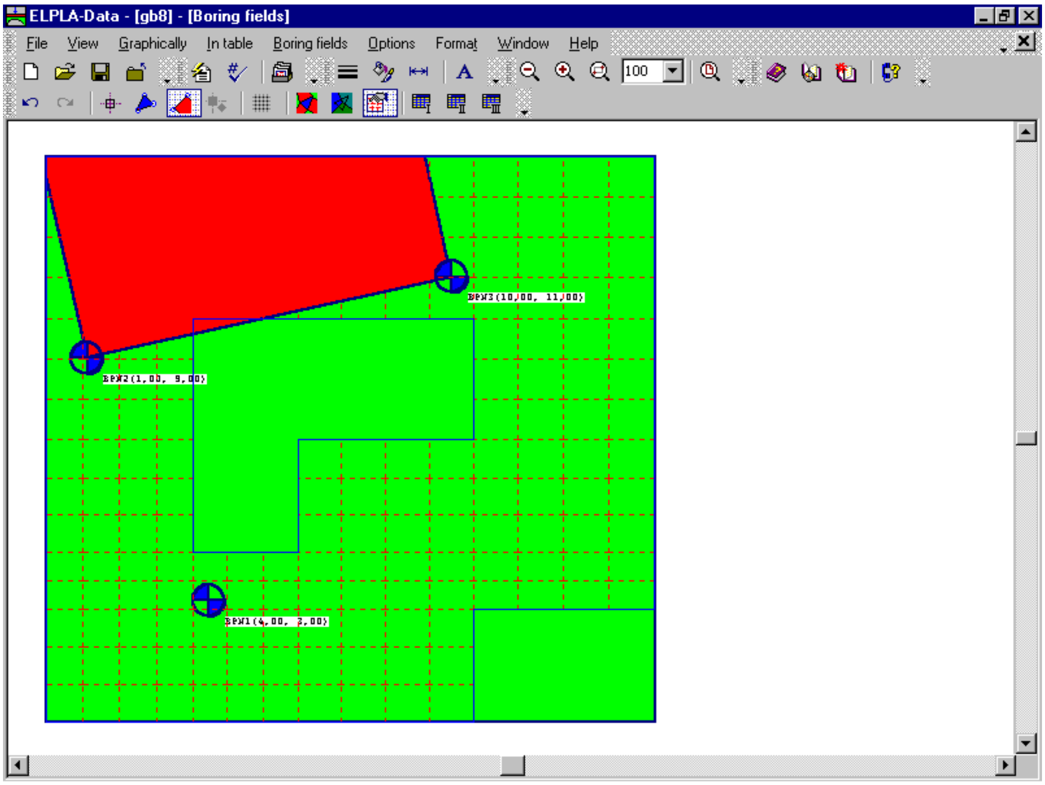


Figure C-118 Zone type II

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

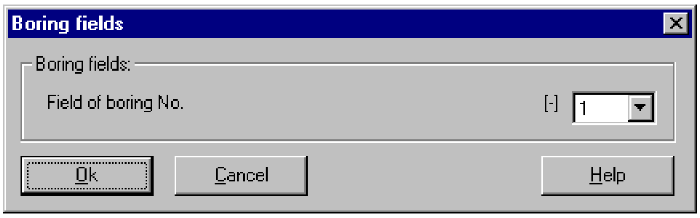


Figure C-119 "Definition of boring fields" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.13.4 In table Menu

This menu contains the following commands:

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

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

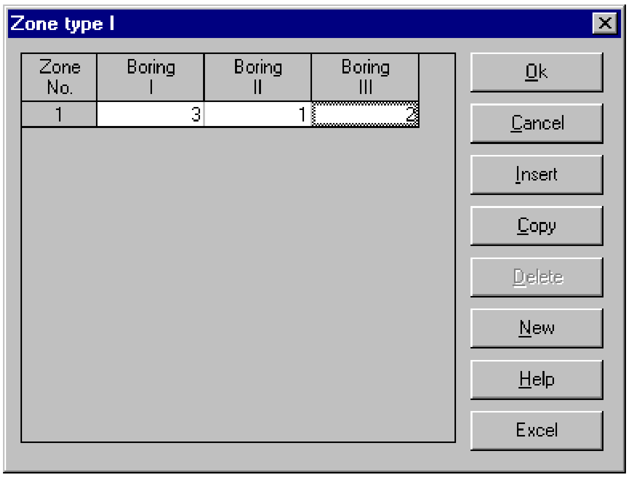


Figure C-120 "Zone type I" Dialog box

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

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

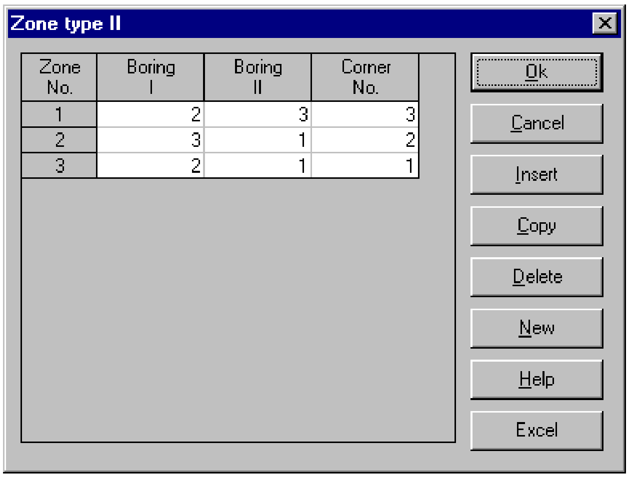


Figure C-121 "Zone type II" Dialog box

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

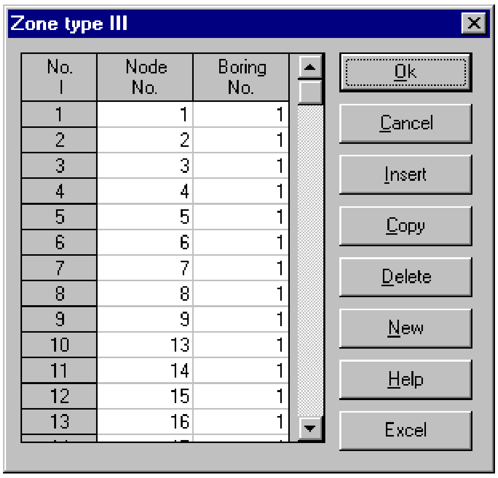


Figure C-122 "Zone type III" Dialog box

5.13.5 Boring fields Menu

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

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

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

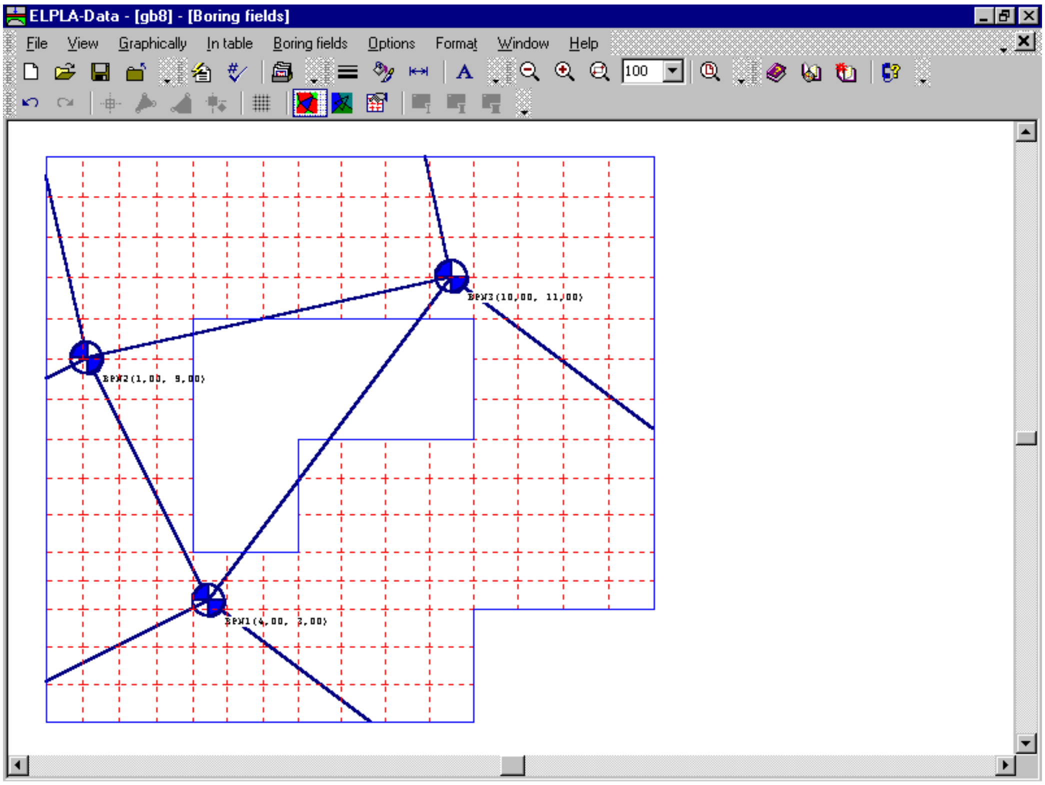


Figure C-123 Boring locations and region types by Interpolation method

Boring fields Menu–"Subarea method" command

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

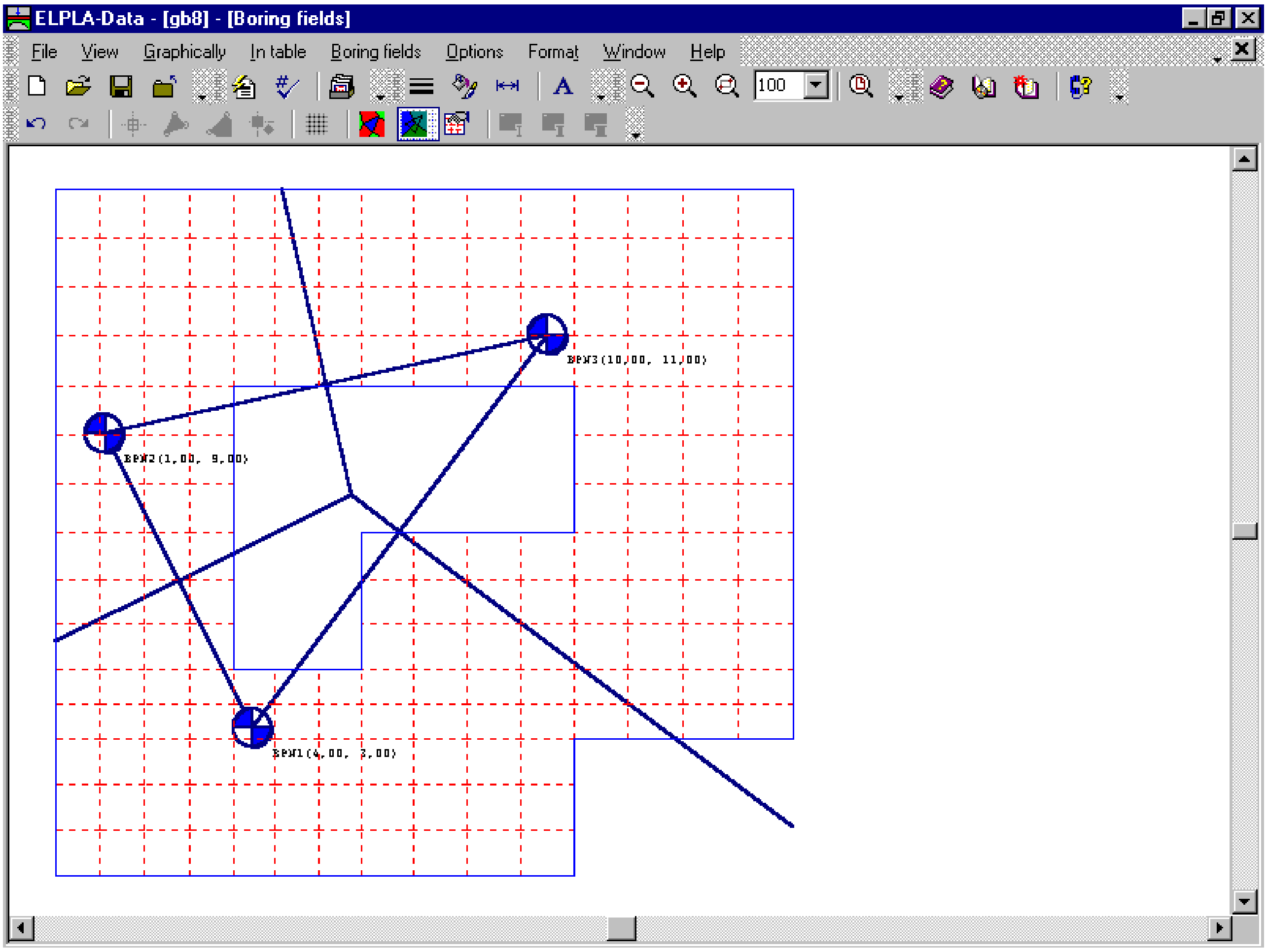


Figure C-124 Boring locations and subareas by Subarea method

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

5.13.6 Options Menu (See paragraph 5.3.6)

5.13.7 Format Menu (See paragraph 5.3.7)

5.13.8 Window Menu (See paragraph 5.3.8)

5.13.9 Help Menu (See paragraph 5.3.9)

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

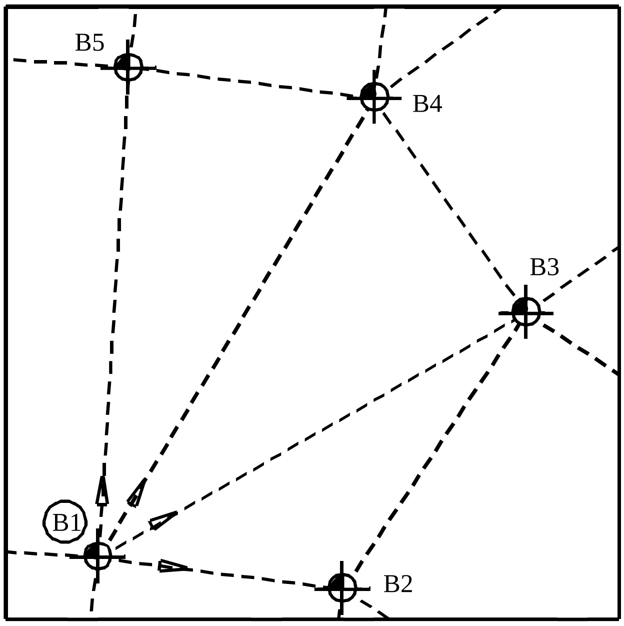


Figure C-125 Division of interpolation zones (numbering of boring a)

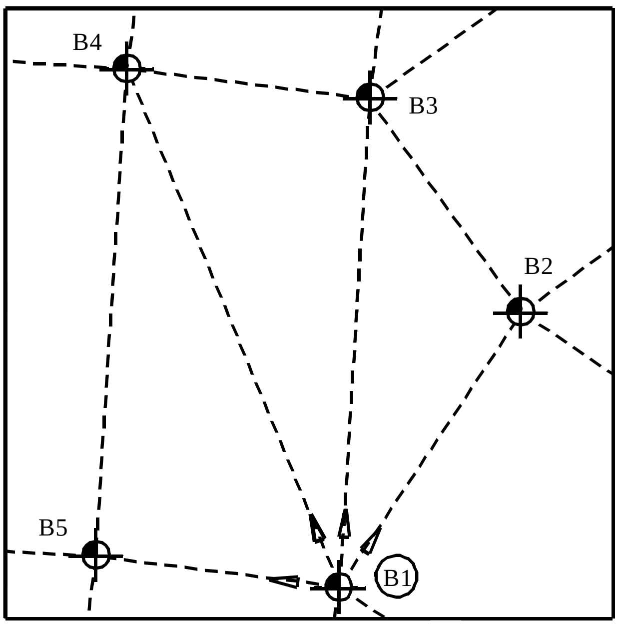


Figure C-126 Division of interpolation zones (numbering of boring b)

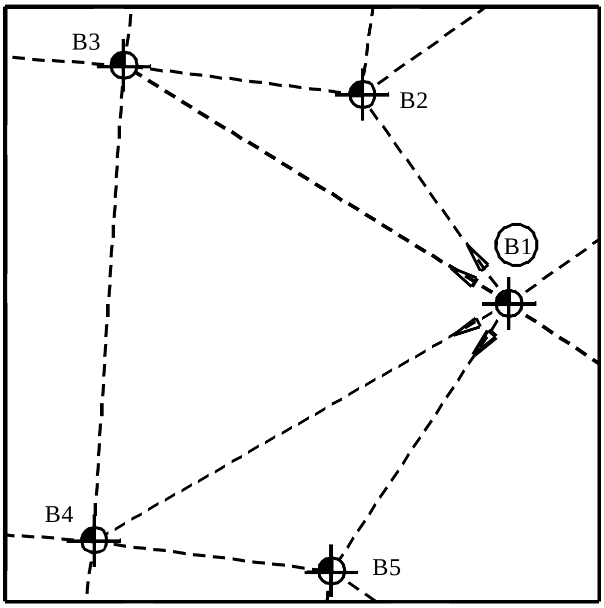


Figure C-127 Division of interpolation zones (numbering of boring c)

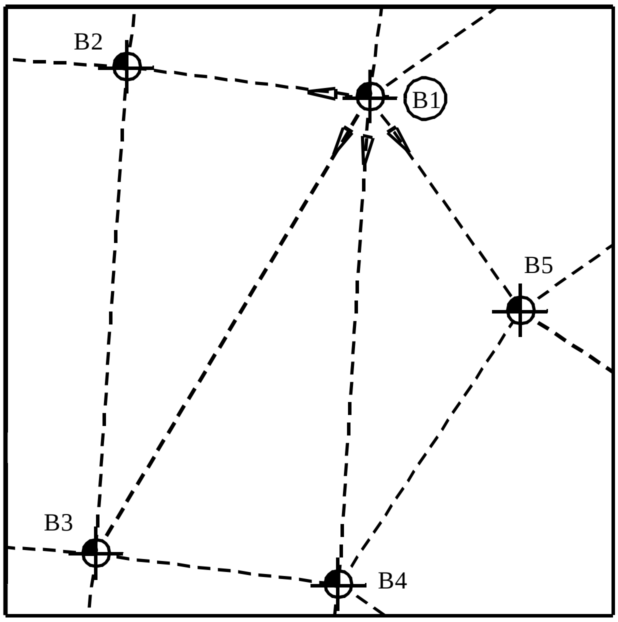


Figure C-128 Division of interpolation zones (numbering of boring d)

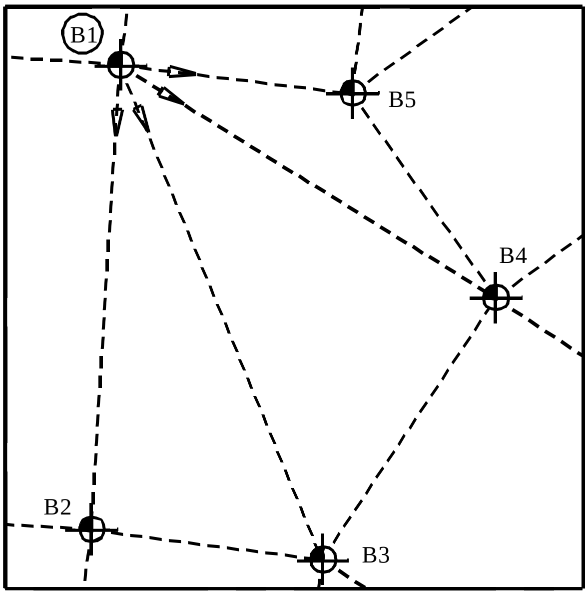


Figure C-129 Division of interpolation zones (numbering of boring e)

5.14 Data Menu–"Loads" command

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

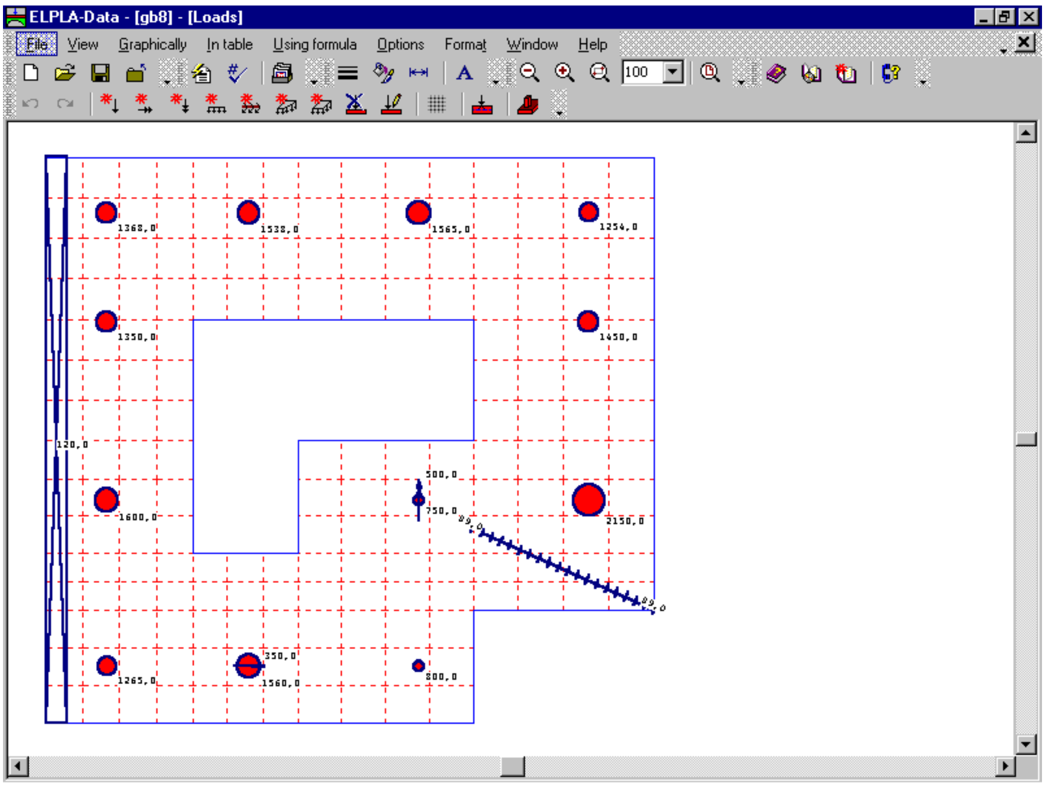


Figure C-130 "Loads" embedded program

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

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

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

5.14.1 File Menu

This menu contains five commands:

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

File Menu–"New loads" command
Defines new loads

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

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

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

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

5.14.2 View Menu (See paragraph 5.3.2)

5.14.3 Graphically Menu

This menu contains the following commands:

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

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

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

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

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

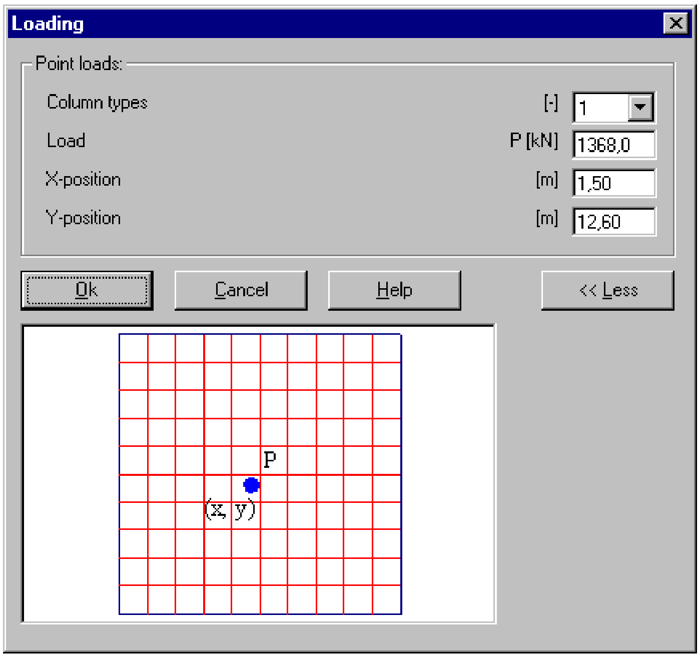


Figure C-131 "Point loads P" Dialog box

Graphically Menu–"Moments M_x" command

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

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

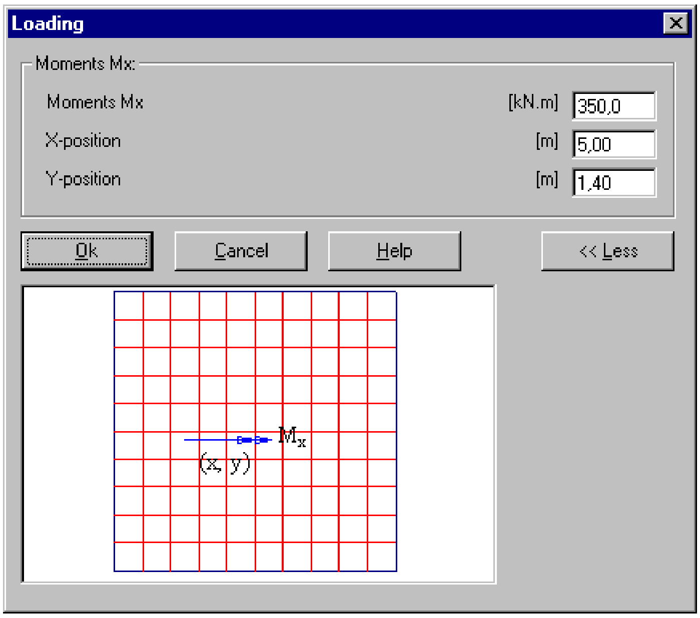


Figure C-132 "Moments M_x" Dialog box

Graphically Menu–"Moments M_y" command

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

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

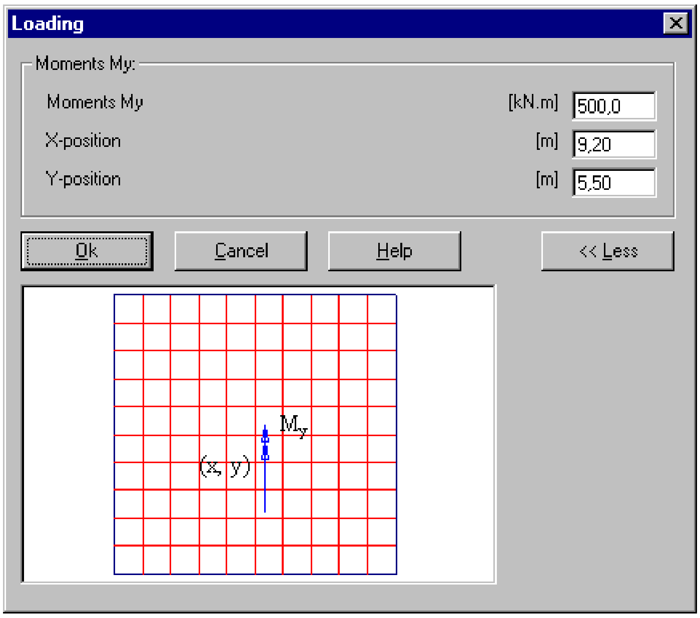


Figure C-133 "Moments M_y" Dialog box

Graphically Menu–"Line loads" command

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

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

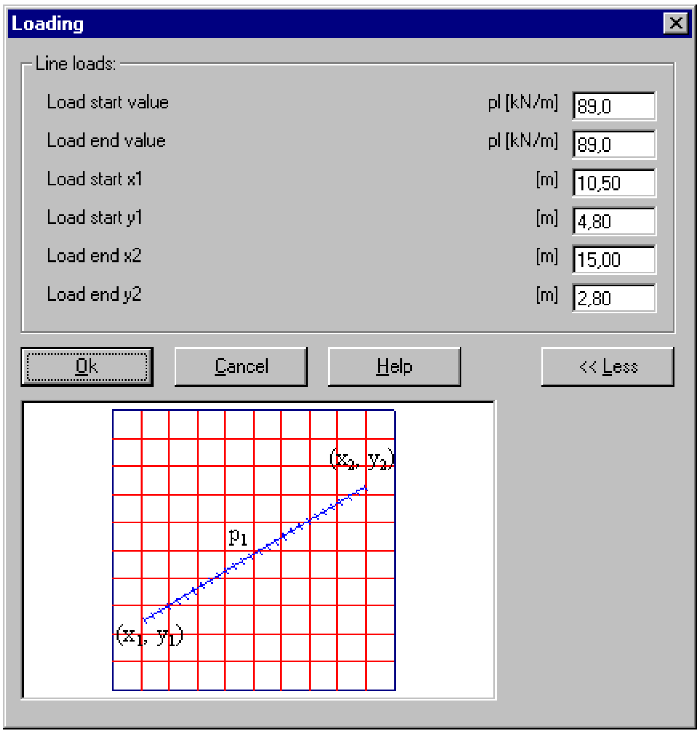


Figure C-134 "Line loads pl" Dialog box

Graphically Menu–"Line moments" command

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

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

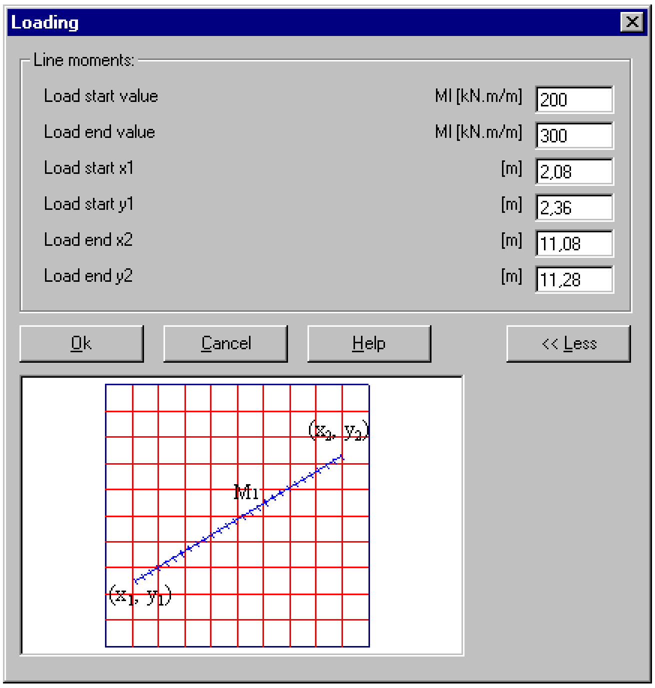


Figure C-135 "Line moments Ml" Dialog box

Graphically Menu–"Distributed loads (Polygon)" command

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

- Load intensity at corners
- Corners position of the polygon

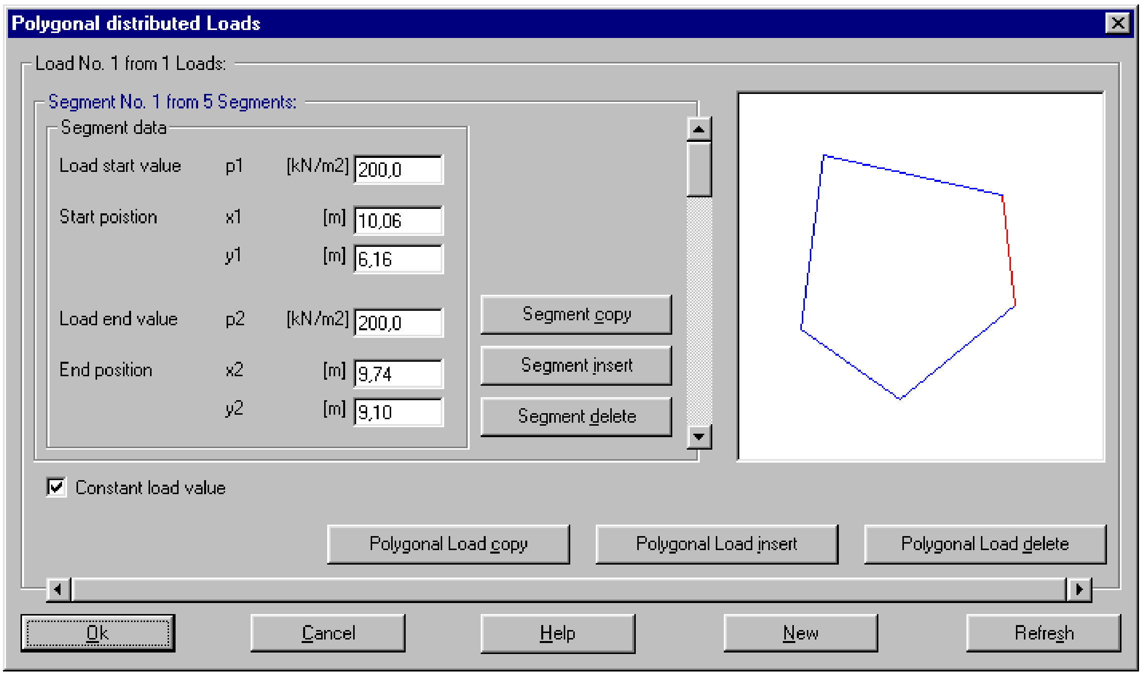


Figure C-136 "Distributed loads p" Dialog box

Graphically Menu–"Distributed loads (Rectangle)" command

By this command the applied load per Meter square can be defined at any diagonal position (x_1, y_1) to (x_2, y_2). The position of the applied load is independent on the FE-Net. When "Distributed loads (Rectangle)" command is chosen, the cursor is changed from an arrow to a cross hair. Then, the load can be defined by holding the left mouse button down at the starting point of the distributed load. As the mouse is dragged, a box appears, indicating a distributed load is being defined. When the left mouse button is released, the following Dialog box of Figure C-137 appears and the following data is required to define:

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

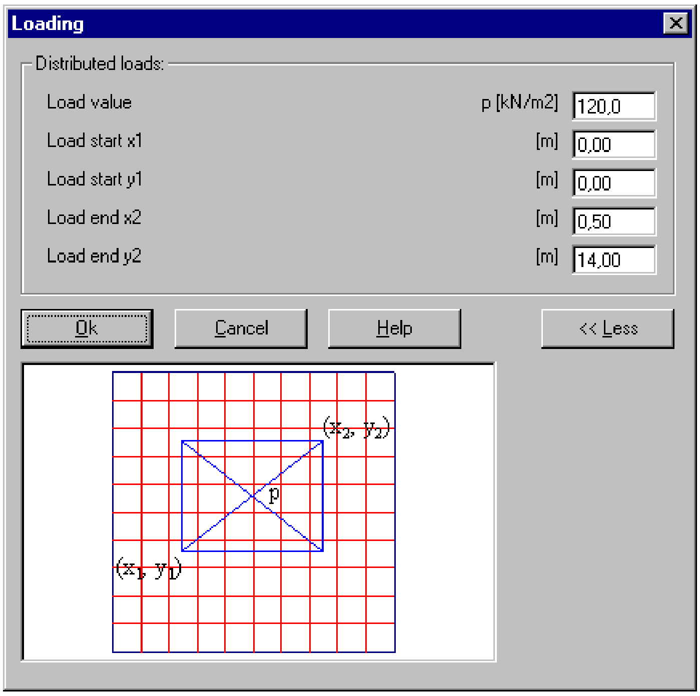


Figure C-137 "Distributed loads (Rectangle)" Dialog box

Graphically Menu–"Remove loads" command

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

Graphically Menu–"Edit Loads" command

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

Note

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

Graphically Menu–"Cartesian grid" command

(See paragraph 5.3.3)

5.14.4 In table Menu

This menu contains the following commands:

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

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

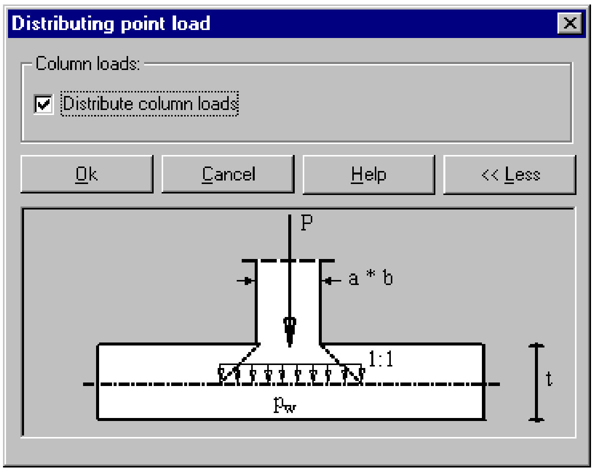
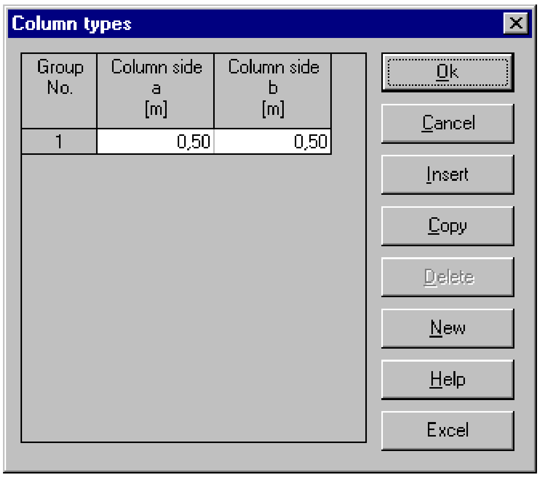


Figure C-138 Distributing the column point load over an appropriate area

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

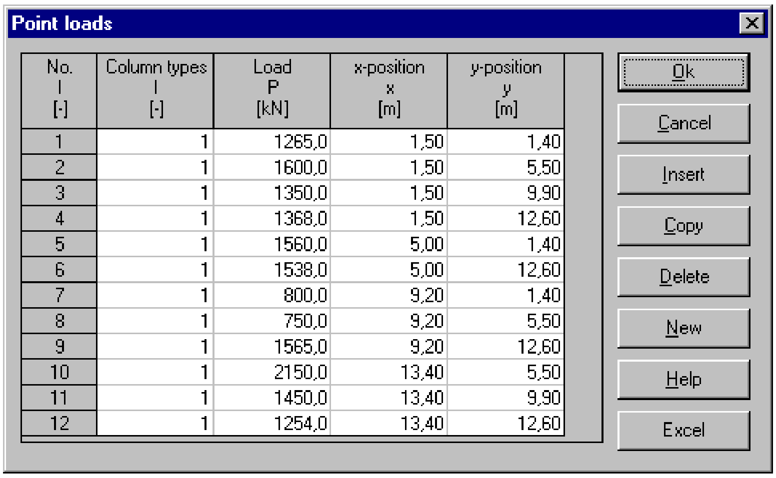


Group No.	Column side a [m]	Column side b [m]
1	0,50	0,50

Figure C-139 Defining column dimensions

In table Menu –"Point loads" command

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



No. I [-]	Column types I [-]	Load P [kN]	x-position x [m]	y-position y [m]
1	1	1265,0	1,50	1,40
2	1	1600,0	1,50	5,50
3	1	1350,0	1,50	9,90
4	1	1368,0	1,50	12,60
5	1	1560,0	5,00	1,40
6	1	1538,0	5,00	12,60
7	1	800,0	9,20	1,40
8	1	750,0	9,20	5,50
9	1	1565,0	9,20	12,60
10	1	2150,0	13,40	5,50
11	1	1450,0	13,40	9,90
12	1	1254,0	13,40	12,60

Figure C-140 "Point loads P" Dialog box

In table Menu–"Moments M_x" command

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

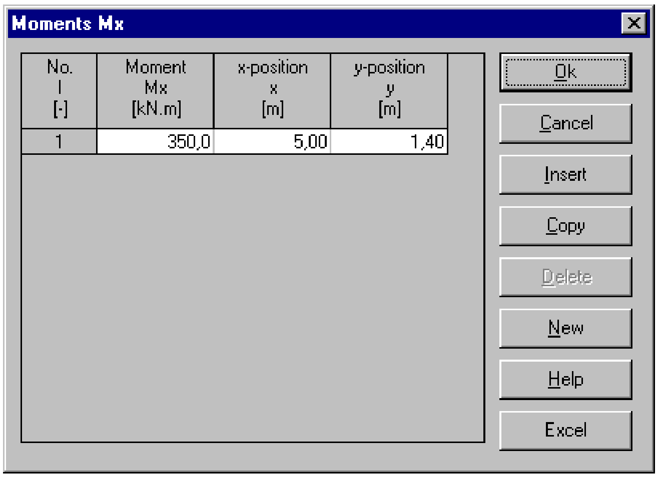


Figure C-141 "Moments M_x" Dialog box

In table Menu–"Moments M_y" command

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

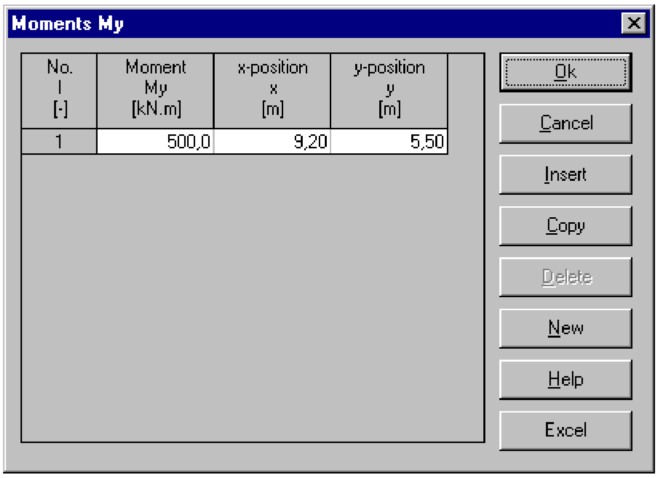


Figure C-142 "Moments M_y" Dialog box

In table Menu–"Line loads" command

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

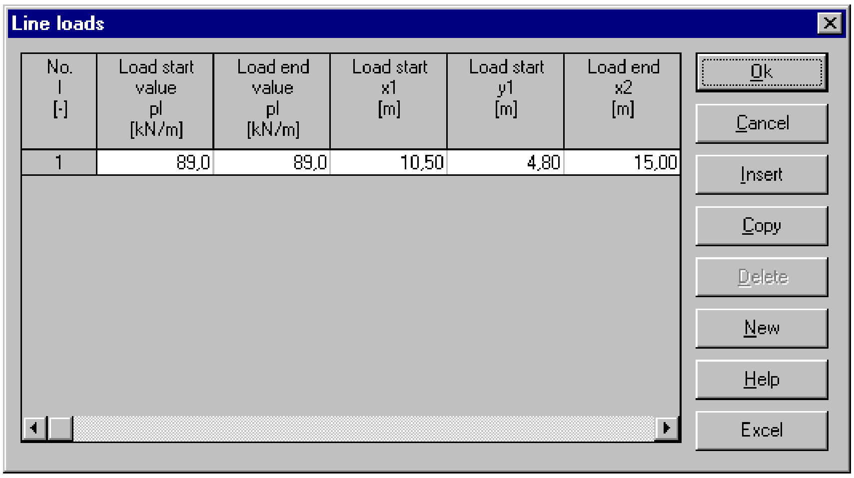


Figure C-143 "Line loads pl" Dialog box

In table Menu–"Line moments" command

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

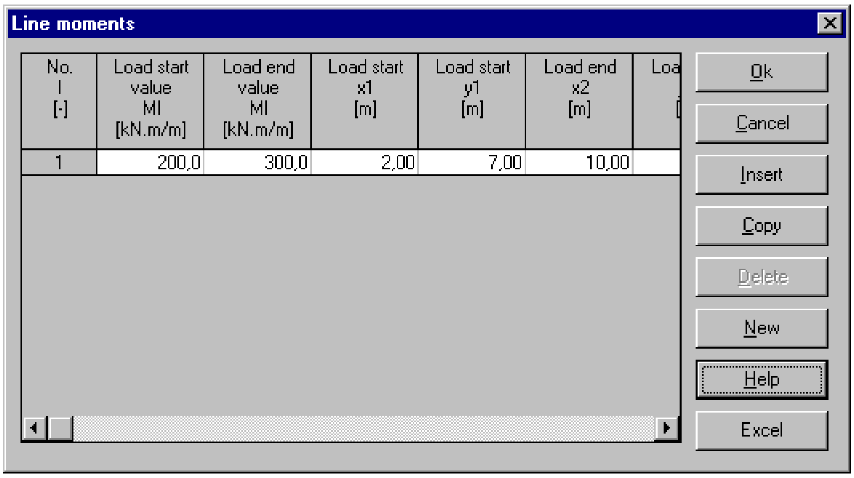


Figure C-144 "Line moments Ml" Dialog box

In table Menu–"Distributed loads (Polygon)" command

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

In table Menu–"Distributed loads (Rectangle)" command

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

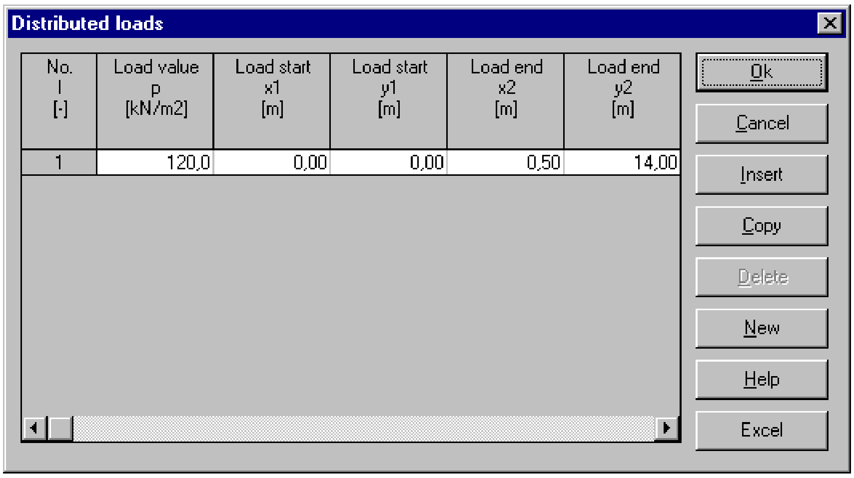


Figure C-145 "Distributed loads p" Dialog box

Note

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

5.14.5 Using formula Menu

This menu contains the following commands:

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

Using formula–"Point loads" command

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

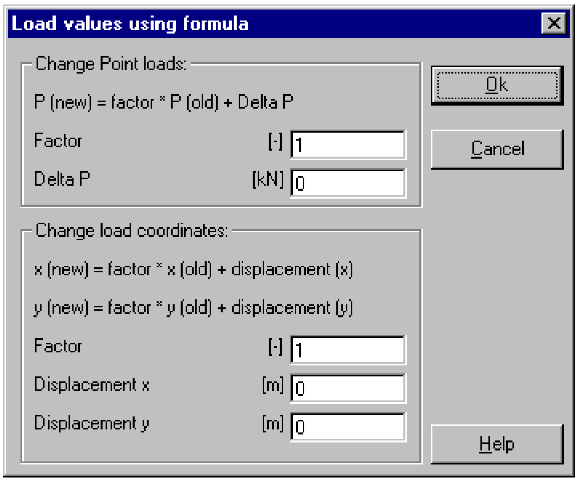


Figure C-146 "Load values using formula" Dialog box

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

5.14.6 **Options Menu** (See paragraph 5.3.6)

5.14.7 **Format Menu** (See paragraph 5.3.7)

5.14.8 **Window Menu** (See paragraph 5.3.8)

5.14.9 **Help Menu** (See paragraph 5.3.9)

5.15 Data Menu–"Neighboring foundations" command

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

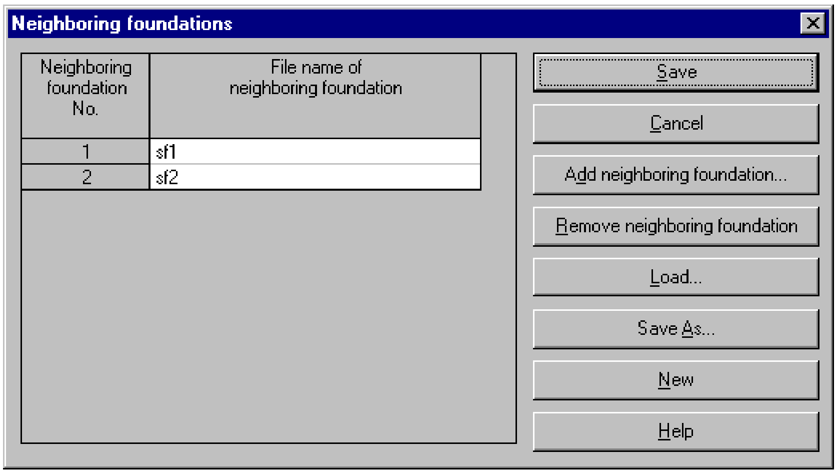


Figure C-147 "Neighboring foundations" Dialog box

5.16 Data Menu–"Temperature change" command

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

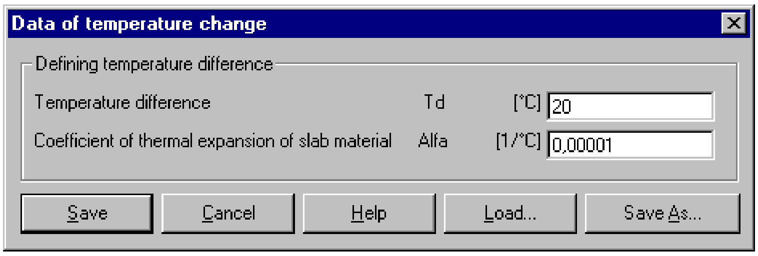


Figure C-148 "Data of temperature change" Dialog box

5.17 Data Menu–"Additional settlements" command

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

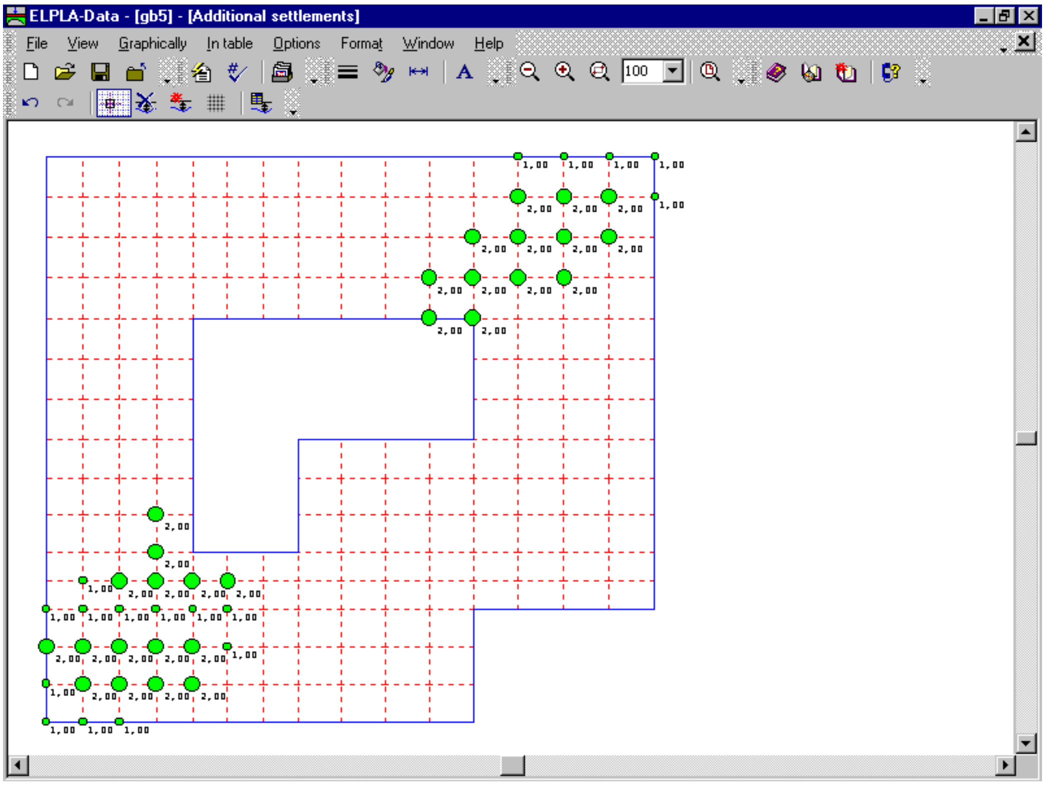


Figure C-149 "Additional settlements" embedded program

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

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

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

5.17.1 File Menu

This menu contains five commands:

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

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

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

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

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

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

5.17.2 View Menu (See paragraph 5.3.2)

5.17.3 Graphically Menu

This menu contains the following commands:

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

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

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

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

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

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

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

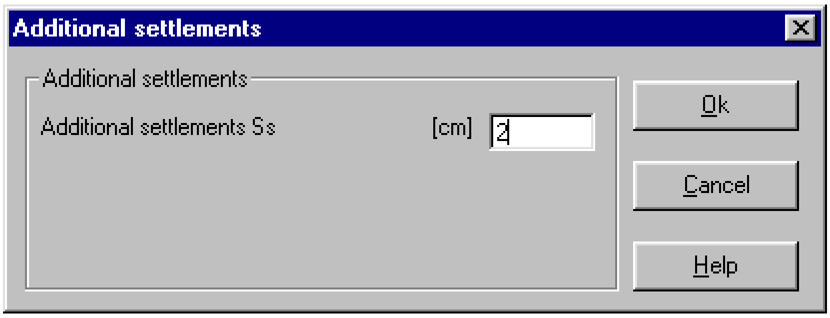


Figure C-150 "Add additional settlements" Dialog box

Graphically Menu–"Cartesian grid" command
See paragraph 5.3.3

5.17.4 In table Menu

This menu contains the following command:

- Additional settlements

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

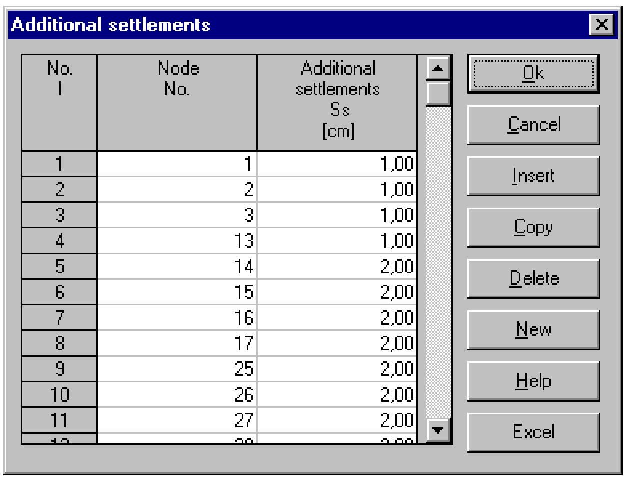


Figure C-151 Defining additional settlements in a table

5.17.5 **Options Menu** (See paragraph 5.3.6)

5.17.6 **Format Menu** (See paragraph 5.3.7

5.17.7 **Window Menu** (See paragraph 5.3.8)

5.17.8 **Help Menu** (See paragraph 5.3.9)

6 **View Menu** (See paragraph 5.3.2)

7 Main Data Menu

The Main Data Menu has the following commands:

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

7.1 Main Data–"Firm header" command

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

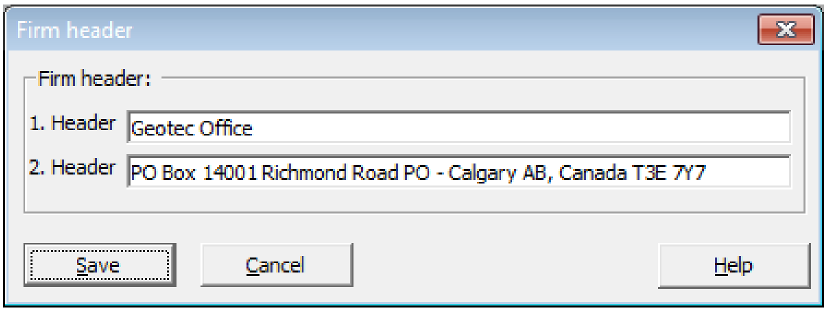


Figure C-152 "Firm header" Dialog box

7.2 Main Data–"Directory of data" command

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

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

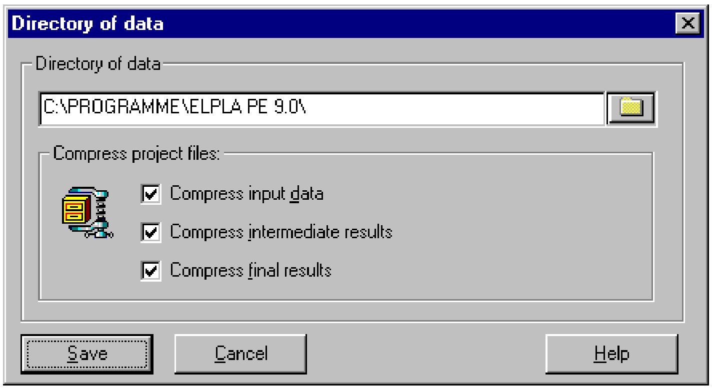


Figure C-153 "Directory of data" Dialog box

7.3 Main Data–"Preferences" command

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

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

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

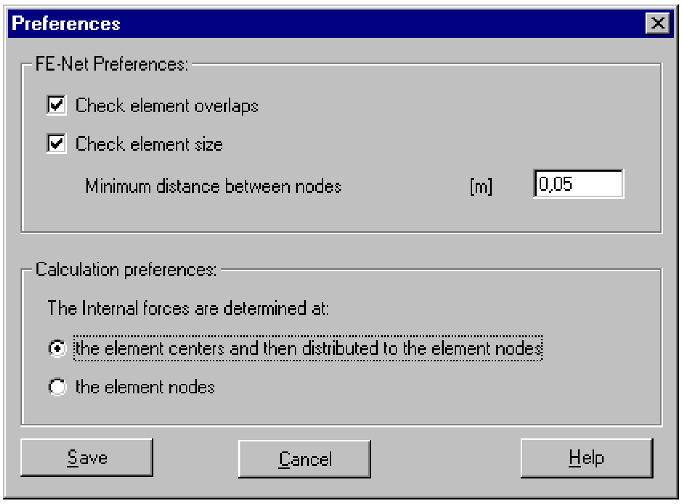


Figure C-154 "Preferences" Dialog box

7.4 Main Data–"Help language setting" command

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

Figure C-155 "Help language setting" Dialog box

7.5 Main Data–"System of unit" command

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

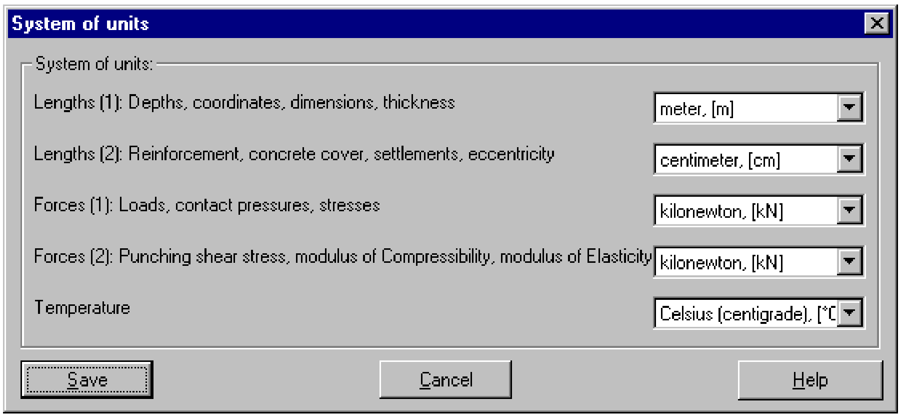


Figure C-156 "System of units" Dialog box

7.6 Main Data–"Number formats" command

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

The following examples describe the number formats:

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

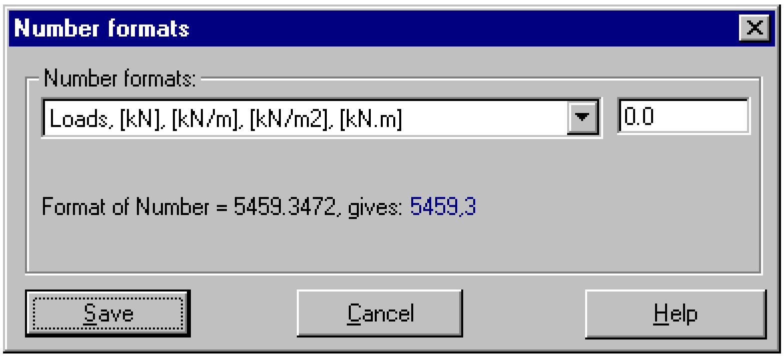


Figure C-157 "Number formats" Dialog box

7.7 Main data–"Design code parameters" command

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

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

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

- ACI
  American Concrete Institute Building Code Requirements for Structural Concrete

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

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

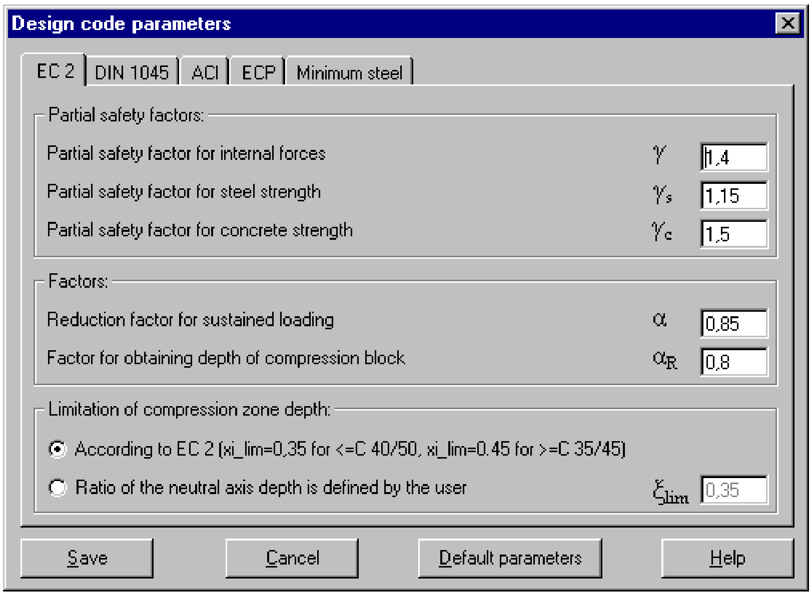


Figure C-158 "Design code parameters" Dialog box

8 Help Menu (See paragraph 5.3.9)

9 Tips and Tricks

9.1 Keyboard

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

Table C-5 Shortcut keys of menu head

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

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

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

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

Table C-7 Shortcut keys of View-Command

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

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

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

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

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

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

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

Table C-11 Shortcut keys of Data-Command

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

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

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

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

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

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

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

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

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

Table C-16 Shortcut keys of Options-Command

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

Table C-17 Shortcut keys of Format-Command

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

Table C-18 Shortcut keys of Window-Command

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

Table C-19 Shortcut keys of Help-Command

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

9.2 Mouse

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

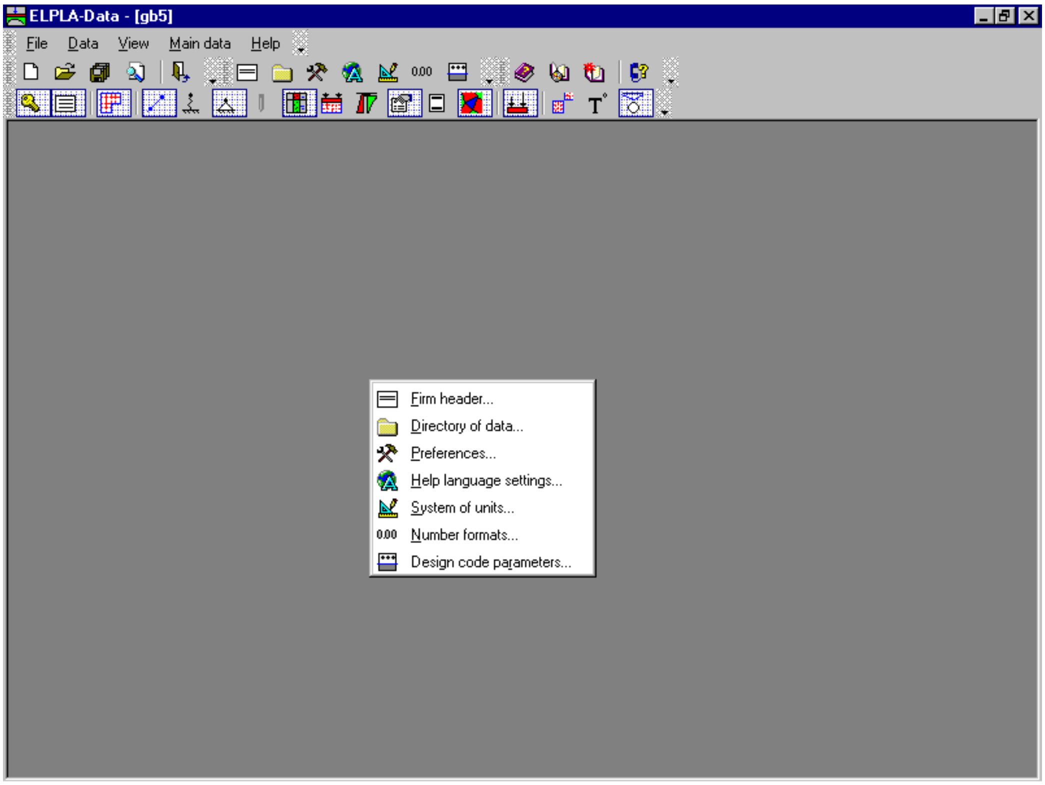


Figure C-159 Menu "Popup-Main data"

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

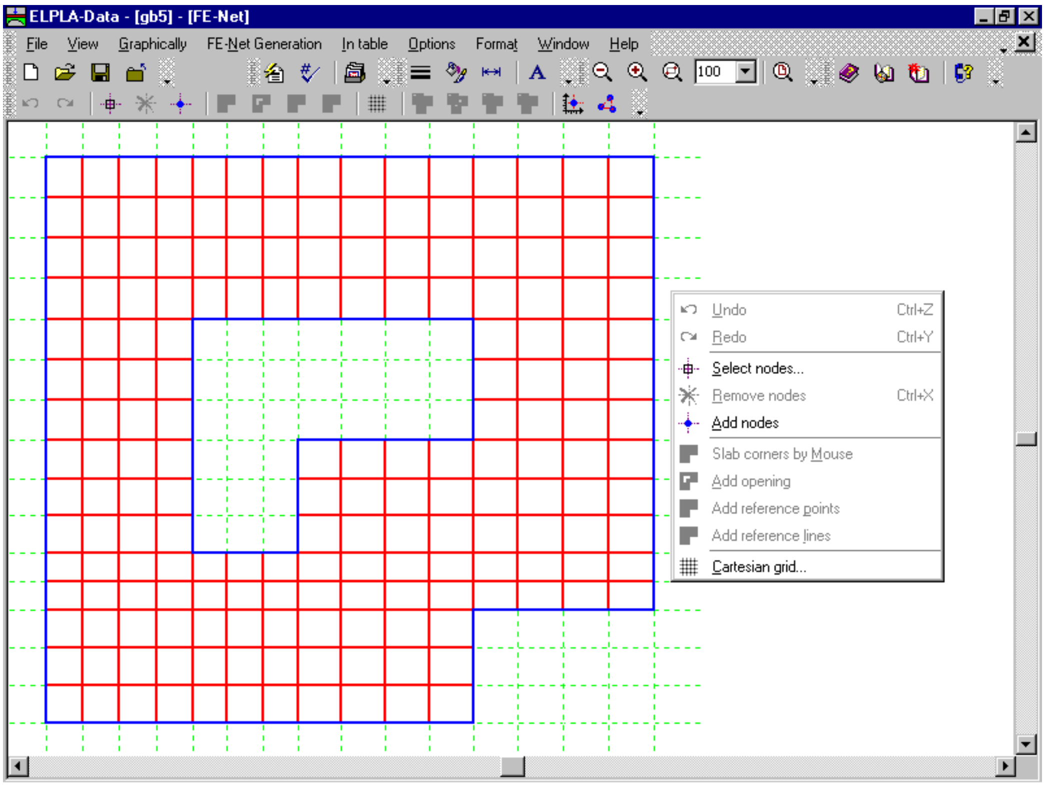


Figure C-160 Menu "Popup-Options"

10 References

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

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

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

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

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

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

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

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

11 Index

A

B

C

D

E

F

G

H

I

L

M

N

O

P

R

S

T

U

V

Z

Part D

User's Guide
for the program ELPLA-Solver

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

Version 9.2

Program authors: M. El Gendy
A. El Gendy

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

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

Contents Page

1 An Overview of ELPLA-Solver

When project data were defined and stored, ELPLA-Solver can carry out many calculations and finally solve the system of linear equations. The calculation steps, which are carried out by ELPLA-Solver, can be listed as follows:

1 Assembling the load vector
2 Preparing the calculation
3 Determining the ultimate bearing capacity
4 Determining the limit depth
5 Determining the modulus of subgrade reaction
6 Determining flexibility coefficients of piles
7 Determining flexibility coefficients of the soil
8 Determining flexibility coefficients for system of rafts
9 Assembling the soil stiffness matrix
10 Influence of neighboring foundations on settlements
11 Influence of the temperature change on raft
12 Influence of additional settlements on raft
13 Assembling the girder stiffness matrix
14 Assembling the slab stiffness matrix
15 Solving the system of linear equations
16 Iteration process
17 Analysis of the rigid raft
18 Analysis of the flexible foundation
19 Performing the nonlinear analysis
20 Performing the nonlinear analysis of piled raft foundation
21 Determining deformation, internal forces, contact pressures
22 Design of the slab
23 Determining displacements, stresses and strains in soil
24 Analysis of the plane frame
25 Analysis of the plane stress
26 Computation of all

All results are saved as ASCII-format in separate files and can therefore be read with any text editor. The file format is straightforward so that an interface for any self-developed or commercial package can be easily used.

2 Description of ELPLA-Solver

ELPLA-Solver is a 32-bit, analysis and design software product that operates under Microsoft Windows 9x/ NT/ ME/ XP. The common "what you see is what you get" of Windows applications makes it easy to learn how to use ELPLA-Solver, especially if you are already familiar with the Windows environment.

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

The usage of ELPLA is typically such that first data files are created describing a certain problem by ELPLA-Data, then the project problem is analyzed by using ELPLA-Solver. Finally, the results can be presented as graphical drawing, graphs and tables using 5 separate programs ELPLA-Graphic, ELPLA-Section, ELPLA-List, ELPLA-Boring and GEOTEC-Editor.

Table D-1 Names and descriptions of the 7 separate programs

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

In order to use ELPLA-Solver, first the user must define the project data by ELPLA-Data. Table D-2 gives a list of files, which are read or created by ELPLA-Solver. The files can be classified in four groups.

Table D-2 Names of file groups

Group	Saved from the program
A Main data files	ELPLA-Data
B Project data files	ELPLA-Data
C Intermediate result files	ELPLA-Solver
D Final result files	ELPLA-Solver

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

Table D-3 Names and contents of files

A Main data files

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

B Project data files

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

C Intermediate result files

Filename	Contents
*. PL3	Indicators for old/ new nodes
*. PL4	Area around nodes
*. PL5	Node type and node art
*. PL7	Element areas
*. PL9	Node coordinates, element types and element groups
*. PC3	Groundwater pressure on the raft
*. PC4	Overburden pressure
*. PC5	Load vector (Part 1)
*. PC6	Load vector (Part 2)
*. PC7	Foundation properties
*. PC8	Average contact pressure, eccentricity and area of slab
*. PC9	Coordinates of element centers
*. P33	Vector of supports/ boundary conditions
*. PI1	Vector of contact pressures (Iteration method)
*. PI2	Flexibility band matrix (Iteration method)
*. PT2	Load vector due to temperature change
*. PP2	Load vector due to neighboring foundations
*. QUB	Main ultimate bearing capacity (q_b)
*. PW2	Main moduli of subgrade reactions (k_b)
*. GF1	Girder stiffness matrix
*. PE1	Soil stiffness matrix
*. PE2	Load vector due to reloading
*. PD1	Flexibility soil matrix for loading for raft
*. PD2	Flexibility soil matrix for reloading for raft
*ji. PD3	Flexibility coefficients of raft j due to contact pressures of raft i
*. FP1	Flexibility soil matrix due to end bearing of piles for loading
*. FP2	Flexibility soil matrix due to end bearing of piles for reloading
*. FP3	Flexibility soil matrix due to skin friction of piles for loading
*. FP4	Flexibility soil matrix due to skin friction of piles for reloading
*. FP5	Stiffness vector of piles
*. PF1	Slab stiffness matrix
*. PF2	Load vector from special cases
*. PG1	Deformation vector (w, θ_x, θ_y)
*. PS1	Deformations of the rigid raft (w_o, tan θ_{xo}, tan θ_{xo})

D Final result files

Filename	Contents
*. PT3	Displacements due to temperature change (s_t)
*. PP3	Settlements due to neighboring foundations (s_e)
*. PV2	Load vector due to additional settlements
*. LD1	Limit depth
*. GH1	Internal forces of girders
*. QUN	Ultimate bearing capacities at nodes (q_{ul})
*. PW1	Modulus of subgrade reaction (k_s)
*. PH1	Settlements (s)
*. PH2	Contact pressures (q)
*. PH3	Moments (m_x)
*. PH4	Moments (m_y)
*. PH5	Moments (m_{xy})
*. PH6	Shear forces (Q_x)
*. PH7	Shear forces (Q_y)
*. PH8	Settlements due to reloading (s_w)
*. PH9	Overburden pressures (Q_u)
*. H10	Support reactions (V)
*. H11	Support reactions (M_y)
*. H12	Support reactions (M_x)
*. H13	Reinforcement of the slab (A_{sx1})
*. H14	Reinforcement of the slab (A_{sx2})
*. H15	Reinforcement of the slab (A_{sy1})
*. H16	Reinforcement of the slab (A_{sy2})
*. THX	Rotations about x-axis (θ_x)
*. THY	Rotations about y-axis (θ_y)
*. THZ	Rotations about z-axis (θ_z)
*.U_X	X-Displacements in soil (u)
*.V_Y	Y-Displacements in soil (v)
*.W_Z	Z-Displacements in soil (w=s)
*.S_X	X-Stresses in soil (σ_x)
*.S_Y	Y-Stresses in soil (σ_y)
*.S_Z	Z-Stresses in soil (σ_z)
*.TXY	XY-Shear stresses in soil (τ_{xy})
*.TXZ	XZ-Shear stresses in soil (τ_{xz})
*.TYZ	YZ-Shear stresses in soil (τ_{yz})
*.VAX	X-Strains in soil (ε_x)
*.VAY	Y-Strains in soil (ε_y)
*.VAZ	Z-Strains in soil (ε_z)
*.VXY	XY-Shear strains in soil (γ_{xy})
*.VXZ	XZ-Shear strains in soil (γ_{xz})
*.VYZ	YZ-Shear strains in soil (γ_{yz})
*. PPU	Punching results
*. FP6	Settlement vector of piles
*. PEI	Pile loads and displacements

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

Next paragraphs describe the purpose and function of each ELPLA-Solver command.

3 Starting ELPLA-Solver

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

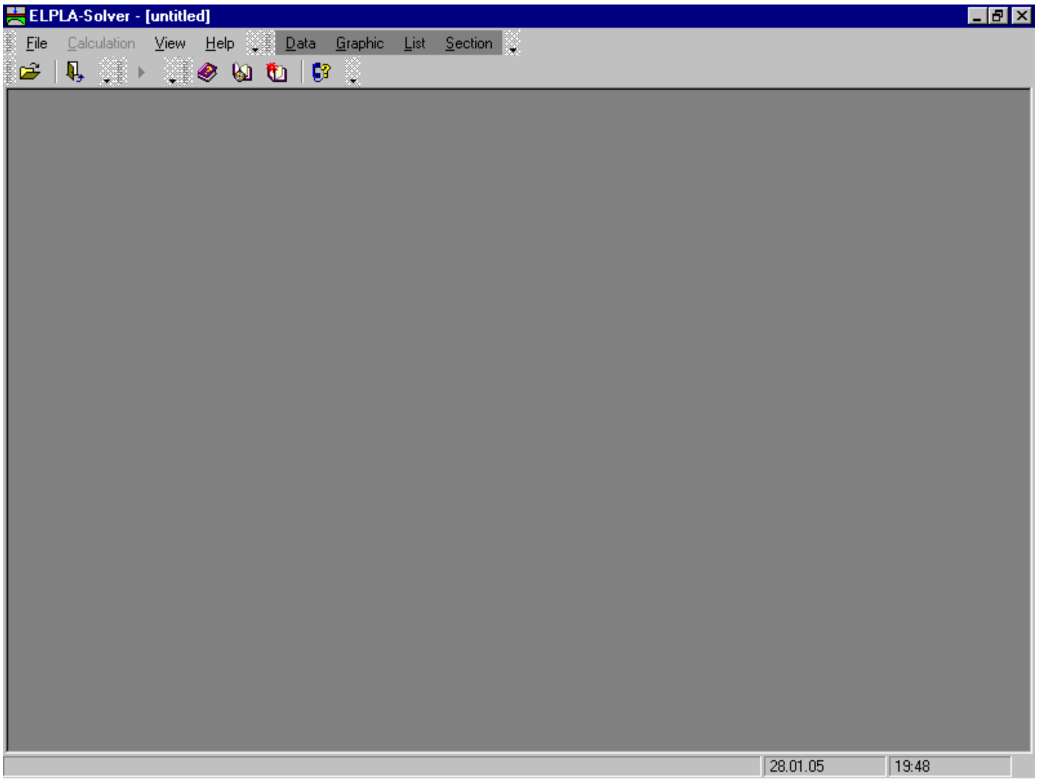


Figure D-1 Introduction screen of the program ELPLA-Solver

The menu head of Figure D-1 contains the following four commands:

- File
- Calculation
- View
- Help

After clicking one of the four menu commands other sub-commands or options become available. The four menu commands and their sub-commands are presented and described in the following paragraphs 4 to 8.

4 File Menu

The File Menu commands are:

- Open
- Files 1, 2, 3, 4
- Exit

4.1 File Menu–"Open" command

By clicking "Open" command the current project is closed, if one is loaded, and an existing project is opened. Figure D-2 shows "Open" Dialog box used to open a specified project.

ELPLA is used to analyze not only an isolated raft but also a system of rafts. Therefore the program can read two types of file names. One has the extension PO1, which represents the isolated raft and the other has the extension PO2, which represents the system of rafts.

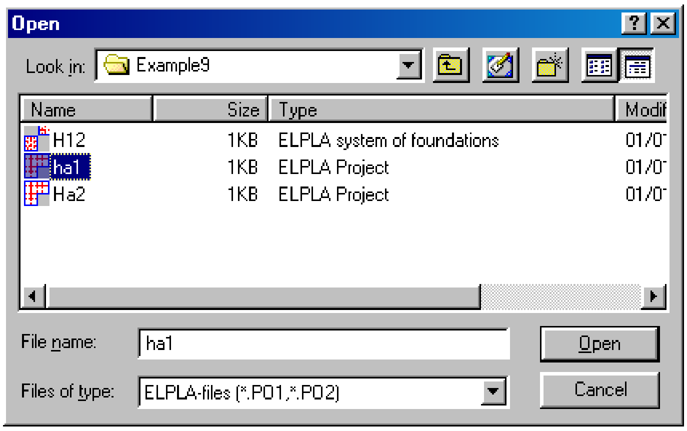


Figure D-2 "Open project" Dialog box

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

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

4.3 File Menu–"Exit" command

Here the current project is closed and ELPLA-Solver is quitted, Figure D-3.

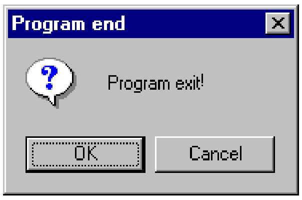


Figure D-3 "Exit" Message box

5 Calculation Menu

The Calculation Menu is the main menu, which is used to carry out the problem analysis. The Calculation Menu commands are:

- Assembling the load vector
- Preparing the calculation
- Determining the ultimate bearing capacity
- Determining the limit depth
- Determining the modulus of subgrade reaction
- Determining flexibility coefficients of piles
- Determining flexibility coefficients of the soil

- Determining flexibility coefficients for system of rafts
- Assembling the soil stiffness matrix
- Influence of neighboring foundations on settlements
- Influence of the temperature change on raft
- Influence of additional settlements on raft
- Assembling the girder stiffness matrix
- Assembling the slab stiffness matrix
- Solving the system of linear equations
- Iteration process
- Analysis of the rigid raft
- Analysis of the flexible foundation
- Performing the nonlinear analysis
- Performing the nonlinear analysis of piled raft foundation
- Determining deformation, internal forces, contact pressures
- Design of the slab
- Determining displacements, stresses and strains in soil
- Analysis of the plane frame
- Analysis of the plane stress
- Computation of all

5.1 Calculation Menu–"Assembling the load vector" command

By "Assembling the load vector" command the load vector is assembled for all calculation methods. When this command is chosen, "Foundation properties" list box appears as shown in Figure D-4.

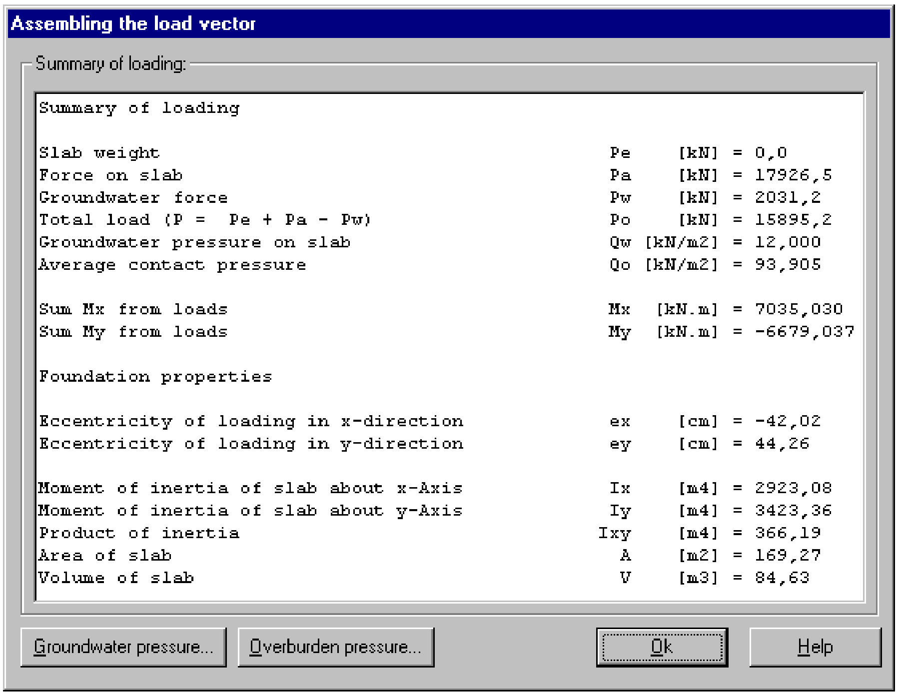


Figure D-4 "Foundation properties" list box

Overburden pressure and groundwater pressure can be also displayed and edited. Clicking "Overburden pressure" and "Groundwater pressure" Buttons can edit these pressures. Figure D-5 shows the Dialog box that appears when "Overburden pressure" Button is clicked. In this Dialog box the overburden pressure can be edited.

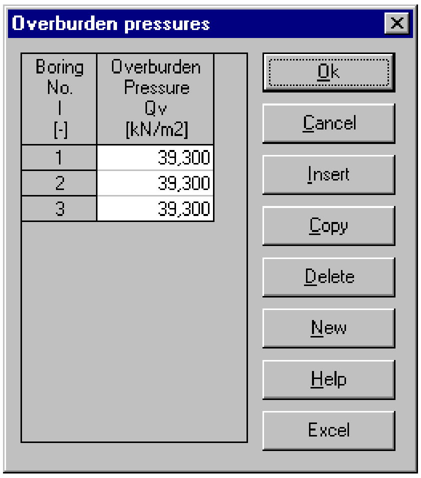


Figure D-5 "Overburden pressures" Dialog box

5.2 Calculation Menu–"Preparing the calculation" command

By "Preparing the calculation" command the optimization of band width is carried out and the load vector is assembled for plane frame and plane stress analyses.

5.3 Calculation Menu–"Determining the ultimate bearing capacity" command

By "Determining the ultimate bearing capacity" command determining the ultimate bearing capacity is carried out. When clicking this option, the program calculates the ultimate bearing capacity and the following menu in Figure D-6 appears. In this menu the main ultimate bearing capacity q_b of each boring is displayed. In the menu of Figure D-6 the ultimate bearing capacities of borings can be redefined, if desired. By clicking the Button "OK", the main ultimate bearing capacities are considered. After that the program calculates the ultimate bearing capacities q_{ul} at all nodes through interpolation or according to the subareas method whenever is applicable (Figure D-7). In the menu of Figure D-7 the ultimate bearing capacities q_{ul} at nodes are displayed and can be redefined, if desired.

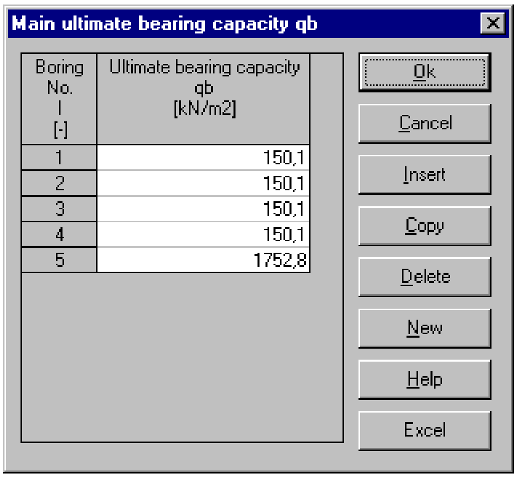


Figure D-6 Main ultimate bearing capacities of borings q_b

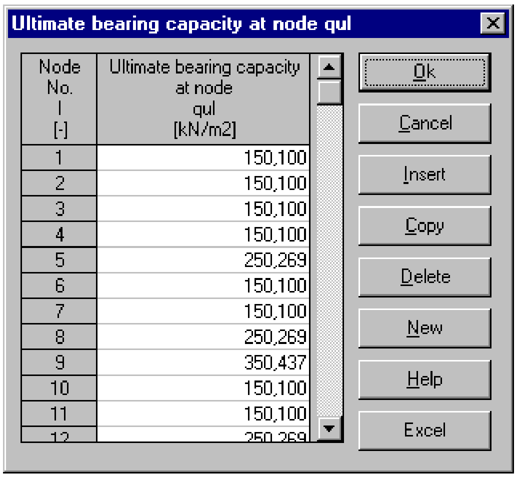


Figure D-7 Ultimate bearing capacities at nodes q_{ul}

5.4 Calculation Menu–"Determining the limit depth" command

By "Determining the limit depth" command the limit depth for the soil layers is determined for layered soil model.

5.5 Calculation Menu–"Determining the modulus of subgrade reaction" command

By "Determining the modulus of subgrade reaction" command determining the modulus of subgrade reaction is carried out for the following calculation methods:

- Constant modulus of subgrade reaction (method 2)
- Variable modulus of subgrade reaction (method 3)

The methods for determining the modulus of subgrade reaction are:

- Modulus is defined by the user
- Modulus is calculated from isotropic elastic half-space soil medium
- Modulus is calculated from layered soil medium

When clicking this option, the program calculates the modulus of subgrade reaction. Then the following menu (Figure D-8) appears. In this menu the average modulus of subgrade reaction k_{sm} of each borings is displayed. In the menu of Figure D-8 the average moduli of subgrade reactions for borings can be redefined, if desired. By clicking the Button "OK", the average moduli of subgrade reactions are considered. After that, the program calculates the moduli of subgrade reactions k_s at all nodes through interpolation or according to the subareas method, Figure D-9. In the menu of Figure D-9 the moduli of subgrade reactions k_s at nodes can be redefined, if desired.

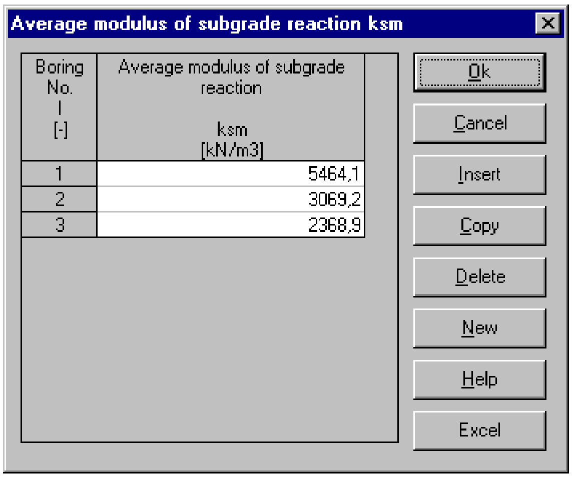


Figure D-8 Average modulus of subgrade reactions of boring k_{sm}

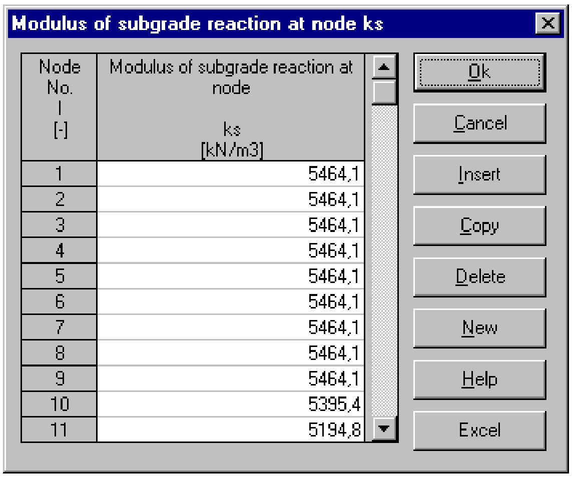


Figure D-9 Modulus of subgrade reactions at node k_s

5.6 Calculation Menu–"Determining flexibility coefficients of piles" command

By this command assembling the flexibility matrix of piles is carried out.

5.7 Calculation Menu–"Determining flexibility coefficients" command

By "Determining flexibility coefficients" command assembling the flexibility matrix is carried out for the following calculation methods:

- Modification of modulus of subgrade reaction by iteration (method 4)
- Modulus of compressibility (half-space, method 5)
- Modulus of compressibility (Iteration, method 6)
- Modulus of compressibility (Elimination, method 7)
- Rigid raft (method 8)
- Flexible foundation (method 9)

5.8 Calculation Menu–"Determining flexibility coefficients for system of rafts" command

By "Determining flexibility coefficients for system of rafts" command assembling the flexibility matrix for system of slab foundations is carried out.

5.9 Calculation Menu–"Assembling the soil stiffness matrix" command

By this command the soil stiffness matrix is carried out for the following calculation methods:

- Modulus of compressibility (half-space, method 5)
- Modulus of compressibility (Iteration, method 6)
- Modulus of compressibility (Elimination, method 7)
- Rigid raft (method 8)

5.10 Calculation Menu–"Influence of neighboring foundations on settlements" command

By "Influence of neighboring foundations on settlements" command determining the settlements due to influence of neighboring foundations is carried out for the following calculation methods:

- Modification of modulus of subgrade reaction by iteration (method 4)
- Modulus of compressibility (half-space, method 5)
- Modulus of compressibility (Iteration, method 6)
- Modulus of compressibility (Elimination, method 7)
- Rigid raft (method 8)
- Flexible foundation (method 9)

5.11 Calculation Menu–"Influence of the temperature change on raft" command

Determining displacements due to temperature change is carried out.

5.12 Calculation Menu–"Influence of additional settlements on raft" command

Determining the influence of additional settlements is carried out.

5.13 Calculation Menu–"Assembling the girder stiffness matrix" command

Assembling the girder stiffness matrix is carried out when girders are in the slab.

5.14 Calculation Menu–"Assembling the slab stiffness matrix" command

Assembling the slab stiffness matrix is carried out for the following calculation methods:

- Linear contact pressure (method 1)
- Constant modulus of subgrade reaction (method 2)
- Variable modulus of subgrade reaction (method 3)
- Modulus of compressibility (half-space, method 5)
- Modulus of compressibility (Elimination, method 7)

5.15 Calculation Menu –"Solving the system of linear equations" command

Solving the system of linear equations is carried out for the following methods:

- Linear contact pressure (method 1)
- Constant modulus of subgrade reaction (method 2)
- Variable modulus of subgrade reaction (method 3)
- Modulus of compressibility (half-space, method 5)
- Modulus of compressibility (Elimination, method 7)

5.16 Calculation Menu–"Iteration process" command

When choosing the command "Iteration process", the iteration process for analyzing the isolated raft (methods 4, 6) or system of rafts is carried out.

Iteration parameters
The iteration process continues until one of the following conditions is met (Figure D-10):

- The accuracy number reaches to the specified tolerance, which means that a sufficient compatibility between the raft deflection and the soil settlement is reached in the slab-soil interface

- The iteration process reaches the specified steps of iterations

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 raft deflection in [m] is considered as an accuracy number.

In the menu of Figure D-10 select the option of the iteration condition. Then click "OK" Button.

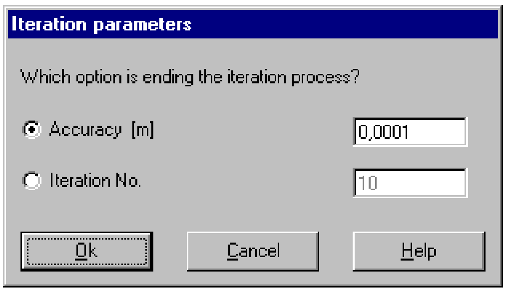


Figure D-10 "Iteration parameters" selection box

Iteration Process
The menu of Figure D-11 displays information about the convergence progress of the solution during the iteration process.

- The iteration processing can be halted at any step by clicking "Stop" Button
- A pause is possible at any step by clicking "Pause" Button. Then "Pause" Button changes to "Continue" Button
- To continue the iteration processing, click "Continue" Button

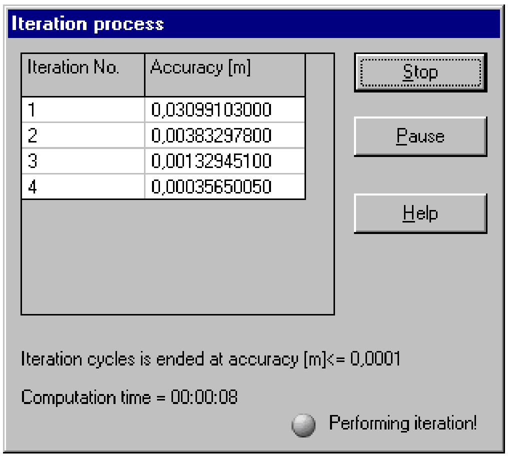


Figure D-11   Menu "Iteration process"

5.17 Calculation Menu–"Analysis of the rigid raft" command

By this command performing the analysis of the rigid raft is carried out (method 8).

5.18 Calculation Menu–"Analysis of the flexible foundation" command

By "Analysis of the flexible foundation" command performing the analysis of the flexible foundation is carried out (method 9).

Negative contact pressures

If a negative value of contact pressure appears (Figure D-12), it indicates tension at the soil raft interface. Since the soil cannot resist tensile forces at the contact surface, a separation occurs between the raft and the soil. An iterative procedure is used to eliminate the negative contact pressures. To eliminate negative contact pressures, click "Yes" Button in the menu of Figure D-12.

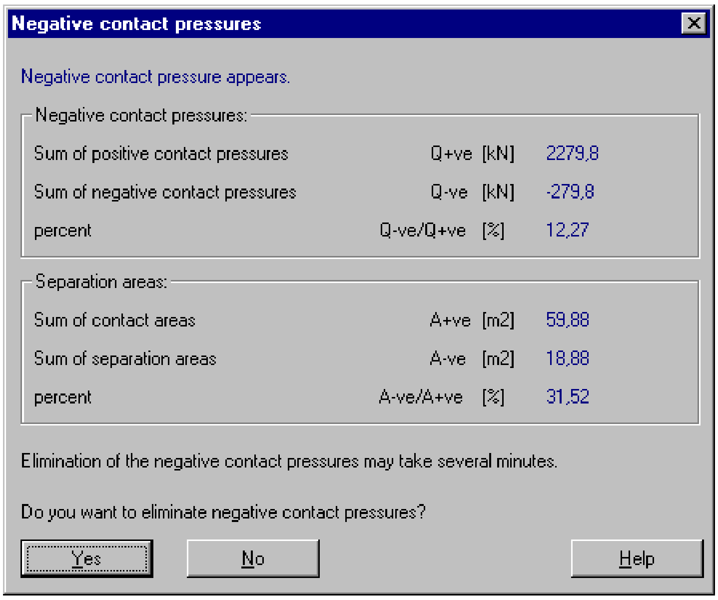


Figure D-12   Menu "Negative contact pressures"

5.19 Calculation Menu–"Performing the nonlinear analysis" command

The nonlinear analysis for methods 2 to 8 is carried out.

5.20 Calculation Menu–"Performing the nonlinear analysis of piled raft foundation" command

Here performing the nonlinear analysis of piled raft foundation is carried out.

5.21 Calculation Menu–"Determining deformation, internal forces, contact pressures" command

When clicking this command, the program calculates the settlements, contact pressures, deformation, rotations, moments and shear forces. By the rigid raft the program calculates only the settlements (= displacement), rotations and contact pressures, while by the flexible foundation, the program calculates only the settlements.

Check of the solution

After determining internal forces and deformation, a check of the solution by comparison between the values of actions and reactions is carried out for all calculation methods. Through this comparative examination, the calculation accuracy is determined, Figure D-13.

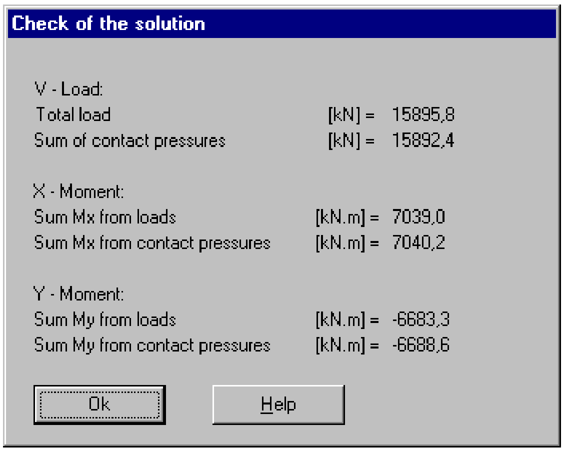


Figure D-13 Menu "Check of the solution"

5.22 Calculation Menu–"Design of the slab" command

By "Design of the slab" command the reinforcement of the slab is determined and a check of punching stress due to column loads, pile reactions and support reactions are carried out for calculation methods 1 to 7.

5.23 Calculation Menu–"Determining displacements, stresses and strains in soil" command

By this command determining displacements, stresses and strains in soil is carried out.

5.24 Calculation Menu–"Analysis of the plane frame" command

By "Analysis of the plane frame" command analyzing plane frame is carried out.

5.25 Calculation Menu–"Analysis of the plane stress" command

By "Analysis of the plane stress" command analyzing plane stress is carried out.

5.26 Calculation Menu–"Computation of all" command

The progress of all computations according to the defined method is carried out.

Different computations are carried out for each method. At the start of the calculation and only by the iteration methods you are asked whether accuracy or iteration No. is to be used for ending the iteration cycles (Figure D-10). Table D-4 shows an overview of the individual calculations for different methods.

Table D-4 Overview of the individual calculations for different methods

Calculation	Numerical calculation method									
	1	2	3	4	5	6	7	8	9	10
Assembling the load vector	x	x	x	x	x	x	x	x	x	x
Determining the ultimate bearing capacity	-	x	x	x	x	x	x	x	-	x
Determining the limit depth	-	x	x	x	-	x	x	x	x	x
Determining the modulus of subgrade	-	x	x	-	-	-	-	-	-	-
Determining flexibility coefficients of piles	-	x	x	x	x	x	x	x	-	-
Determining flexibility coefficients	-	-	-	x	x	x	x	x	x	x
Determining flexibility coefficients for system of rafts	-	-	-	-	-	-	-	-	-	x
Assembling the soil stiffness matrix	-	-	-	*	x	x	x	x	x	x
Influence of neighboring foundations on settlements	-	-	-	x	x	x	x	x	x	x
Influence of the temperature change on raft	-	x	x	x	x	x	x	-	-	x
Influence of additional settlements on raft	-	x	x	x	x	x	x	x	x	x
Assembling the girder stiffness matrix	x	x	x	-	x	-	x	-	-	-
Assembling the slab stiffness matrix	x	x	x	-	x	-	x	-	-	-
Solving system equations (full matrix)	-	-	-	-	x	-	x	-	-	-
Solving system equations (banded matrix)	x	x	x	-	-	-	-	-	-	-
Analysis of the rigid raft	-	-	-	-	-	-	-	x	-	-
Analysis of the flexible foundation	-	-	-	-	-	-	-	-	x	-
Iteration process	-	-	-	x	-	x	-	-	-	x
Performing the nonlinear analysis	-	x	x	x	x	x	x	x	-	x
Performing the nonlinear analysis of piled raft foundation										
Determining def., internal forces and contact pressures	x	x	x	x	x	x	x	x	-	x
Determining deformation and contact pressures	-	-	-	-	-	-	-	-	x	-
Design of the slab	x	x	x	x	x	x	x	-	-	x
Determining displacements, stresses and strains in soil	-	-	-	x	x	x	x	x	x	x
Computation of all	x	x	x	x	x	x	x	x	x	x

* Only by the two especial cases of influence of neighboring foundations on settlements and influence of the temperature change on the raft

The numerical calculation methods in the last 10 cells of Table D-4 are:

a) Analysis of isolated raft

1 Linear contact pressure
2 Constant modulus of subgrade reaction
3 Variable modulus of subgrade reaction
4 Modification of modulus of subgrade reaction by iteration
5 Modulus of compressibility (half-space)
6 Modulus of compressibility (Iteration)
7 Modulus of compressibility (Elimination)
8 Rigid raft
9 Flexible foundation

b) Analysis of system of rafts

10 Analysis of system of many flexible, rigid and elastic rafts

6 View Menu

The View Menu commands are:

- Status bar
- Tool bars

6.1 View Menu–"Status bar" command

"Status bar" command displays a status bar on the screen down. The status bar displays information about the progress of calculation.

6.2 View Menu–"Tool bars" command

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

7 Help Menu

The Help Menu commands are:

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

7.1 Help Menu–"Contents" command

"Contents" command displays a help file in HTML-Format containing the complete ELPLA User's Guide, Figure D-14.

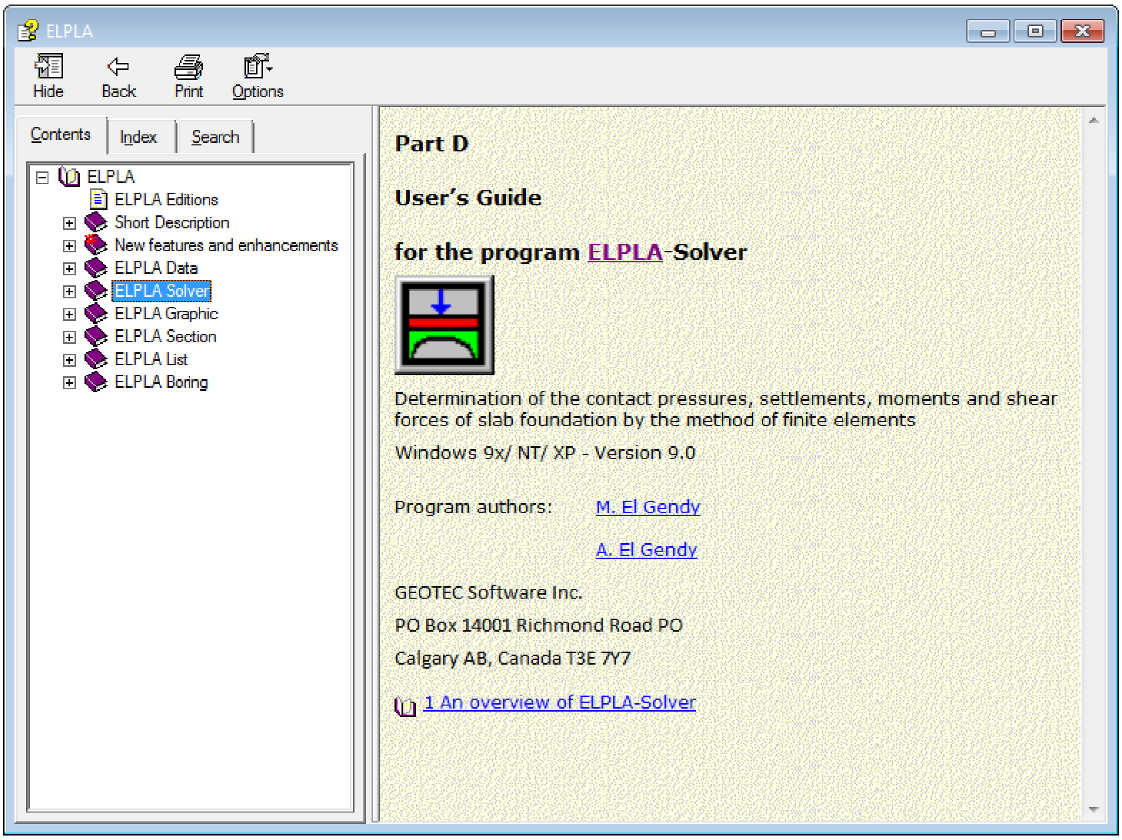


Figure D-14 Menu "Contents"

7.2 Help Menu–"Short description of ELPLA" command

"Short description of ELPLA" command gives a short description of ELPLA package.

7.3 Help Menu–"New in ELPLA" command

"New in ELPLA" command summarizes the new features and enhancements in ELPLA.

7.4 Help Menu–"About ELPLA-Solver" command

Clicking the command "About ELPLA-Solver" displays the information form of ELPLA-Solver as shown in Figure D-15, which gives information about ELPLA-Solver and the calculation method of the loaded project.

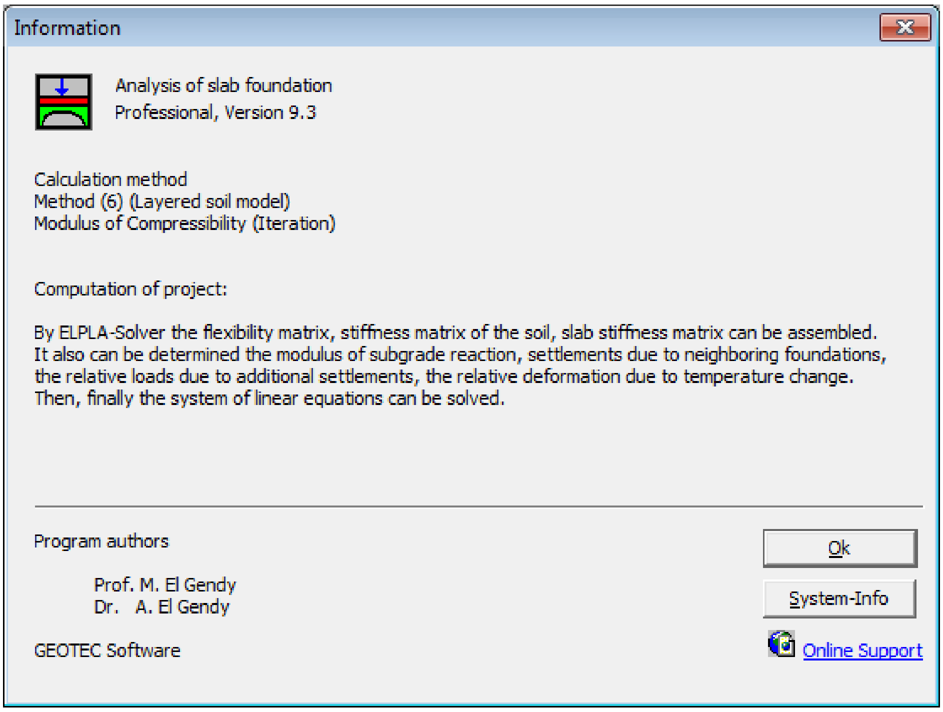


Figure D-15 Information form of ELPLA-Solver

8 Tips and Tricks
8.1 Keyboard

The user can obtain all menu titles and commands also through Shortcut keys. The action of the Shortcut keys is listed in Table D-5 to Table D-9:

Table D-5 Shortcut keys of menu head

Shortcut keys	Action
[Alt+f]	Calling menu head "File"
[Alt+v]	"View"
[Alt+c]	"Calculation"
[Alt+h]	"Help"

Table D-6 Shortcut keys of File-Command

Shortcut keys	Action
[Ctrl +o] or [Alt+f] then [o]	Calling command "Open"
[Alt+f] then [1]	Calling the first project from the last four loaded projects
[Alt+f] then [2]	Calling the second project from the last four loaded projects
[Alt+f] then [3]	Calling the third project from the last four loaded projects
[Alt+f] then [4]	Calling the fourth project from the last four loaded projects
[Ctrl+q] or [Alt+f] then [x]	Calling command "Exit"

Table D-7 Shortcut keys of Calculation-Command

Shortcut keys	Action	
[Alt+c] then [t]	Calling command	"Assembling the load vector"
[Alt+c] then [p]		"Preparing the calculation"
[Alt+c] then [m]		"Determining the ultimate bearing capacity"
[Alt+c] then [z]		"Determining the limit depth"
[Alt+c] then [m]		"Determining the modulus of subgrade reaction"
[Alt+c] then [f]		"Determining flexibility coefficients of piles"
[Alt+c] then [f]		"Determining flexibility coefficients"
[Alt+c] then [x]		"Determining flexibility coefficients for system of rafts"
[Alt+c] then [l]		"Assembling the soil stiffness matrix"
[Alt+c] then [e]		"Influence of neighboring foundations on settlements"
[Alt+c] then [i]		"Influence of the temperature change on raft"
[Alt+c] then [u]		"Influence of additional settlements on raft"
[Alt+c] then [g]		"Assembling the girder stiffness matrix"
[Alt+c] then [b]		"Assembling the slab stiffness matrix"
[Alt+c] then [s]		"Solving the system of linear equations"
[Alt+c] then [s]		"Solving the rigid raft"
[Alt+c] then [s]		"Solving the flexible foundation"
[Alt+c] then [t]		"Iteration process"
[Alt+c] then [n]		"Performing the nonlinear analysis"
[Alt+c] then [n]		"Performing the nonlinear analysis of piled raft foundation"
[Alt+c] then [d]		"Determining deformation, internal forces, contact pressures"
[Alt+c] then [r]		"Design of the slab"
[Alt+c] then [d]		"Determining displacements, stresses and strains in soil"
[Alt+c] then [a]		"Analysis of the plane frame"
[Alt+c] then [a]		"Analysis of the plane stress"
[Alt+c] then [c]		"Computation of all"

Table D-8 Shortcut keys of View-Command

Shortcut keys	Action	
[Alt+v] then [b]	Calling command	"Status bar"
[Alt+v] then [t]		"Tool bars"
[Alt+v] then[t], then [f]		"Tool bars-File"
[Alt+v] then[t], then [c]		"Tool bars-Calculation"
[Alt+v] then[t], then [h]		"Tool bars-Help"
[Alt+v] then[t], then [r]		"Tool bars-Reset Toolbar"

Table D-9 Shortcut keys of Help-Command

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

8.2 Mouse

By clicking the right mouse Button on the screen, the user can also obtain the Popup-Calculation-Menu (Figure D-16).

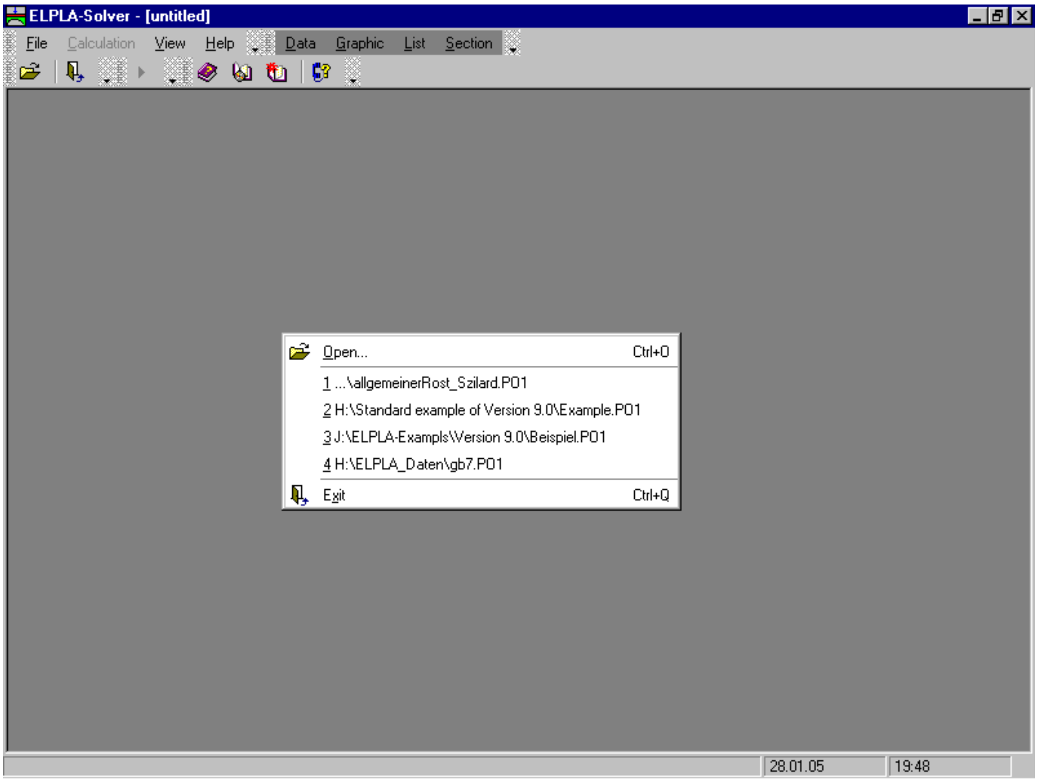


Figure D-16 Menu "Popup-Calculation"

9 Index

Part E

User's Guide
for the program ELPLA-Graphic

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

Version 9.2

Program authors: M. El Gendy
A. El Gendy

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

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

Contents Page

Page

1 An overview of ELPLA-Graphic

ELPLA-Graphic is used to display graphically the calculated results and input project data. The drawing of the results and data can be displayed first on the screen, and then can be sent to the printer or plotter. The calculated results and input project data can be presented graphically as follows:

- Results in isometric view
- Results as contour lines
- Result values in the plan
- Distribution curves of results in the plan
- Results as circular diagrams
- Deformation
- Principal moments as streaks
- Support reactions as arrows
- Soil deformation as vectors
- Soil deformation as deformed mesh
- Principal soil stresses as streaks
- Principal soil strains as streaks
- Data in isometric view
- Data in the plan
- Boring locations
- Boring logs/ Limit depth
- Punching shear
- Beam results

The drawings, if desired, can be saved as WMF-format files, in which they can be exported to other Windows applications to prepare reports, slide presentations or add further information.

2 Description of ELPLA-Graphic

ELPLA-Graphic is a 32-bit, graphical software product that operates under Microsoft Windows 9x/ NT/ ME/ XP. The common "what you see is what you get" of Windows applications makes it easy to learn how to use ELPLA-Graphic, especially if you are already familiar with the Windows environment. The program package ELPLA consists of 7 separate programs. These programs can run independently. The name and short description of the seven separate programs are given in Table E-1.

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

Table E-1 Names and descriptions of the 7 separate programs

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

In order to use ELPLA-Graphic, first the user must define the project data by ELPLA-Data, and then must analyze the problem by ELPLA-Solver. Table E-2 gives a list of files, which are read or created by ELPLA-Graphic. The files can be classified in four groups.

Table E-2 Names of file groups

Group	Saved from the program
A Main data files	ELPLA-Data
B Graphic files (*.GRA)	ELPLA-Graphic
C Project data files	ELPLA-Data
D Final result files	ELPLA-Solver

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

Table E-3 Names and contents of files

A Main data files

Filename	Contents
FIRMA	Firm header
STEU	Default directory for files that are saved by ELPLA
NOFORMAT	Number formats
UNITS	System of units

B Graphic files

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

C Project data files

Filename	Contents
*. AUF	3 lines text to identify the project
* .PO1	System data (calculation method-isolated slab)
* .PO2	System data (calculation method-system of slabs)
*. P23	Reinforcement data
*. P33	Data of supports/ boundary conditions
*. P35	Data of spring supports
*. P81	Data of spring supports for plane frame
*. P91	Data of spring supports for plane stress
*. P61	Data of supports/ boundary conditions for plane frame
*. P71	Data of supports/ boundary conditions for plane stress
*. P31	Data of supports/ boundary conditions for slab and grid
* .BAU	Soil data
*. PC1	Load data
*. PCF	Load data for plane frame
*. PCW	Load data for plane stress
*. PL6	FE-Net data
*. PL4	Area around nodes
*. GL1	Girder data
*. P21	Data of material/ slab thickness/ levels/ coordinates
*. P41	File of boring fields
*. PP1	File of external foundations
*. PV1	Data of additional soil settlements
*. DSS	FE-Net in z-direction
*. PIL	Pile properties

D Final result files

Filename	Contents
*. GH1	Internal forces of girders
*. PT3	Displacements due to temperature change (s_t)
*. PP3	Settlements due to neighboring foundations (s_e)
*. QUN	Ultimate bearing capacities at nodes (q_{ul})
*. PW1	Modulus of subgrade reaction (k_s)
*. PH1	Settlements (s)
*. PH2	Contact pressures (q)
*. PH3	Moments (m_x)
*. PH4	Moments (m_y)
*. PH5	Moments (m_{xy})
*. PH6	Shear forces (Q_x)
*. PH7	Shear forces (Q_y)
*. H10	Support reactions (V)
*. H11	Support reactions (M_y)
*. H12	Support reactions (M_x)
*. H13	Reinforcement of the slab (A_{sx1})
*. H14	Reinforcement of the slab (A_{sx2})
*. H15	Reinforcement of the slab (A_{sy1})
*. H16	Reinforcement of the slab (A_{sy2})
*.U_X	X-Displacements in soil (u)
*.V_Y	Y-Displacements in soil (v)
*.W_Z	Z-Displacements in soil (w=s)
*.S_X	X-Stresses in soil (σ_x)
*.S_Y	Y-Stresses in soil (σ_y)
*.S_Z	Z-Stresses in soil (σ_z)
*.TXY	XY-Shear stresses in soil (τ_{xy})
*.TXZ	XZ-Shear stresses in soil (τ_{xz})
*.TYZ	YZ-Shear stresses in soil (τ_{yz})
*.VAX	X-Strains in soil (ε_x)
*.VAY	Y-Strains in soil (ε_y)
*.VAZ	Z-Strains in soil (ε_z)
*.VXY	XY-Shear strains in soil (γ_{xy})
*.VXZ	XZ-Shear strains in soil (γ_{xz})
*.VYZ	YZ-Shear strains in soil (γ_{yz})
*. PPU	Punching results

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

Next paragraphs describe the purpose and function of each ELPLA-Graphic command.

3 Starting ELPLA-Graphic

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

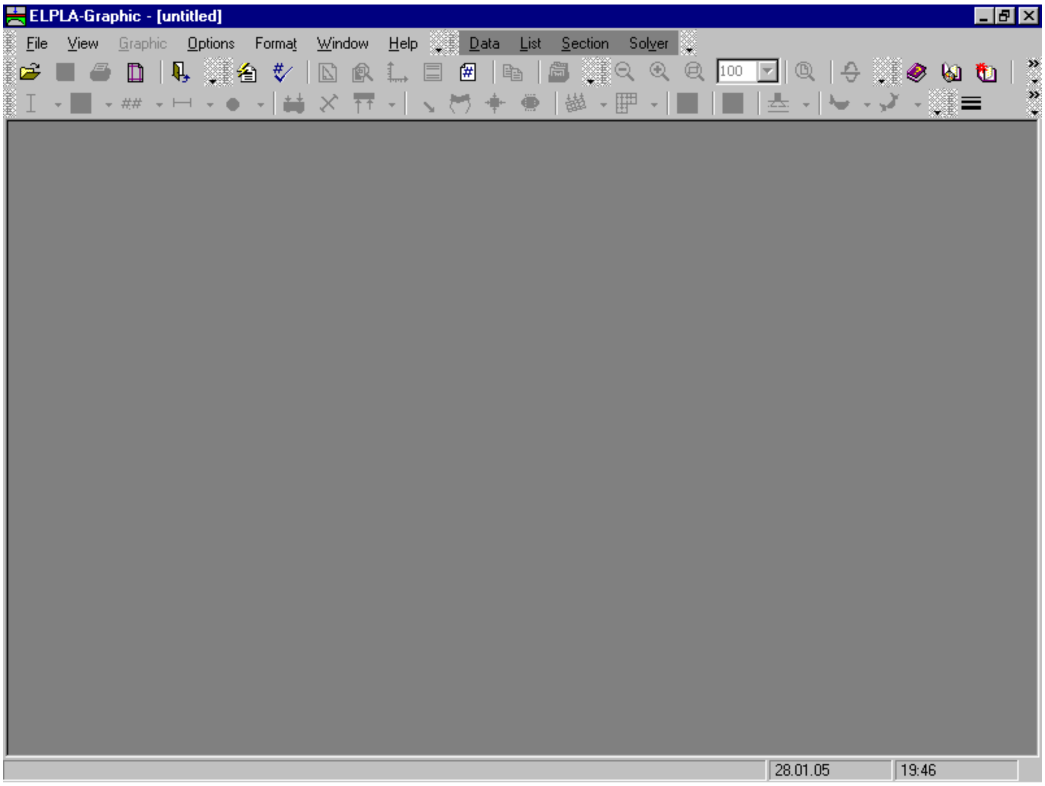


Figure E-1 Introduction screen of ELPLA-Graphic

The menu head of Figure E-1 contains the following seven commands:

- File
- View
- Graphic
- Options
- Format
- Window
- Help

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

4 File Menu

The File Menu commands are:

- Open
- Make WMF-File
- Send to ELPLA-Section
- Print
- Page setup
- Files 1, 2, 3, 4
- Exit

4.1 File Menu–"Open" command

By "Open" command the current project is closed, if one is loaded, and an existing project is opened. Figure E-2 shows "Open" Dialog box used to open a specified project.

ELPLA is used to analyze not only isolated slab foundation but also system of rafts. Therefore the program can read two types of file names. One has the extension of PO1, which represents the isolated raft and the other has the extension of PO2, which represents the system of rafts.

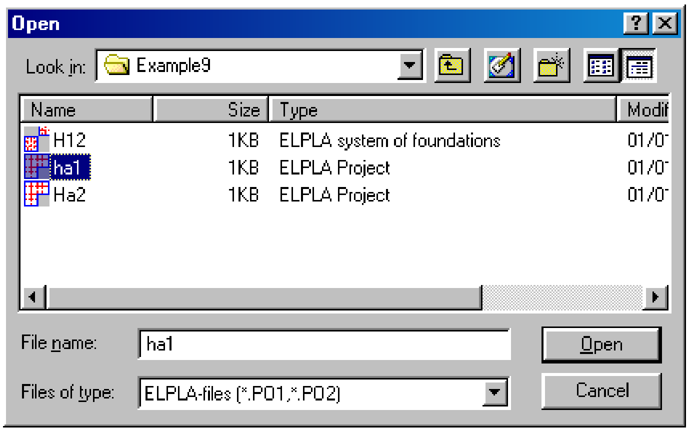


Figure E-2 "Open project" Dialog box

4.2 File Menu–"Make WMF-File" command

The drawing can be saved in a format that can be read by other programs. This feature allows you to include your drawing in reports and presentations and to enhance the drawing using other drawing or CAD software packages. The drawing can be exported in the Windows Metafile format (WMF). Figure E-3 shows "Save as" Dialog box used to export the drawing with the filename you wish to give the exported file, including extension and the directory in which to save the file.

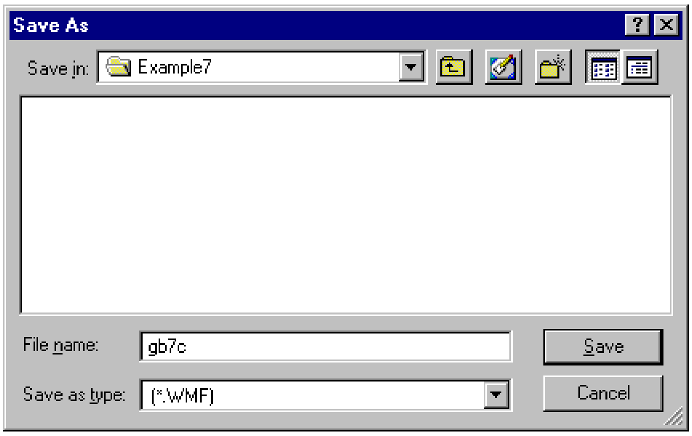


Figure E-3 "Save as" Dialog box

4.3 File Menu–"Send to ELPLA-Section" command

By "Send to ELPLA-Section" command it is possible, to define a diagram in ELPLA-Graphic by Mouse and end it to ELPLA-Section, Figure E-4.

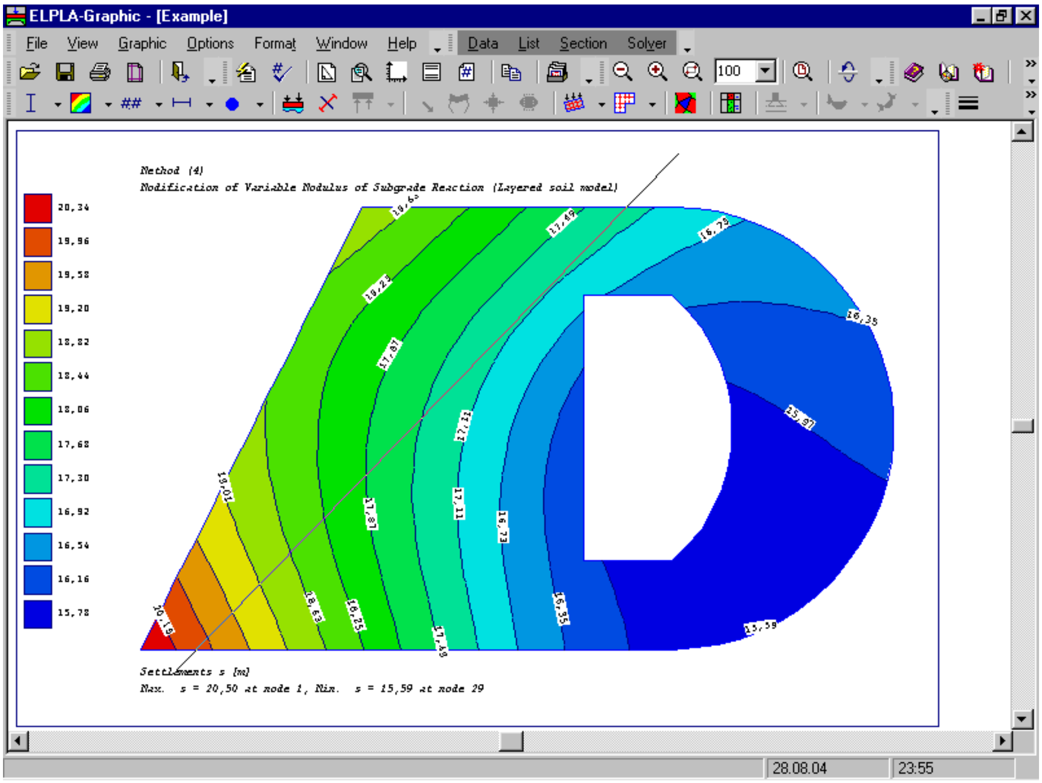


Figure E-4 Defining a diagram in ELPLA-Graphic

4.4 File Menu–"Print" command

By "Print" command data and results can be graphically plotted or printed. Only the objects currently displayed on the drawing are printed. Figure E-5 shows "Print" Dialog box. The printer group box contains controls for selecting the printer and changing its properties. Use "Name" Combo box to select the printer and use "Properties" Button to set printer settings. The number of printing copies can be defined in "Copies" Input box, Figure E-5.

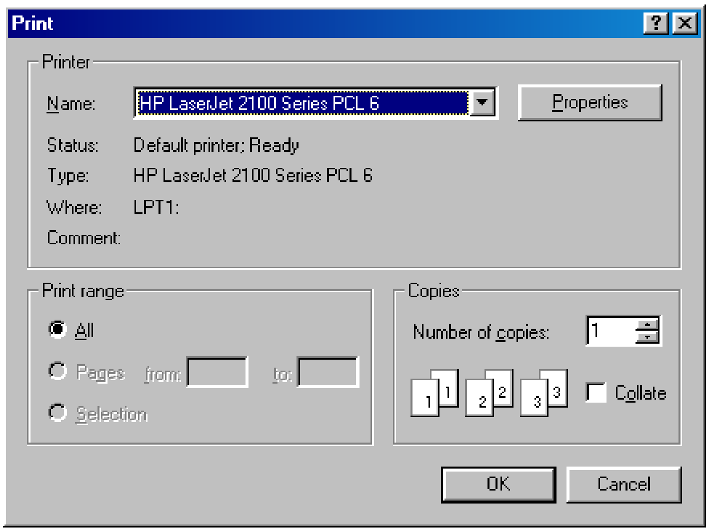


Figure E-5 "Print" Dialog box

4.5 File Menu–"Page setup" command

By "Page setup" command the standard "Page setup" Dialog box can be displayed with options to specify the printer, page orientation, paper size, and paper source, as well as other printing options. Figure E-6 shows "Page setup" Dialog box.

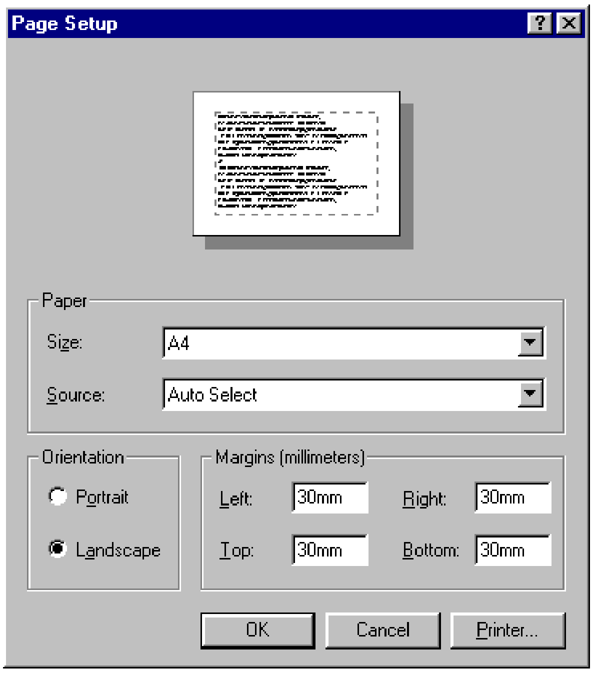


Figure E-6 "Page setup" Dialog box

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

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

4.7 File Menu–"Exit" command

By "Exit" command the current project is closed and ELPLA-Graphic is quitted, Figure E-7.

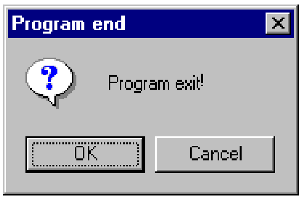


Figure E-7 "Exit" Message box

5 View Menu

The View Menu commands are:

- Status bar
- Tool bars

5.1 View Menu–"Status bar" command

The status bar displays information about the progress of the current operation.

5.2 View Menu–"Tool bars" command

Tool bars located just below the menu head are displayed with icons of program menus.

6 Graphic Menu

The graphic menu is the main menu, used to display the specified drawing with the setting given by options, format and window menus. The Graphic Menu commands are:

- Results in isometric view
- Results as contour lines
- Result values in the plan
- Distribution curves of results in the plan
- Results as circular diagrams
- Deformation
- Principal moments as streaks
- Support reactions as arrows
- Soil deformation as vectors
- Soil deformation as deformed mesh
- Principal soil stresses as streaks
- Principal soil strains as streaks
- Data in isometric view
- Data in the plan
- Boring locations
- Boring logs/ Limit depth
- Punching shear
- Beam results

6.1 Graphic Menu–"Results in isometric view" command

Calculated results can be displayed on the screen and plotted in isometric shape. In the Dialog box of Figure E-8 select an item to draw and click "OK". Figure E-9 shows the contact pressures in isometric view.

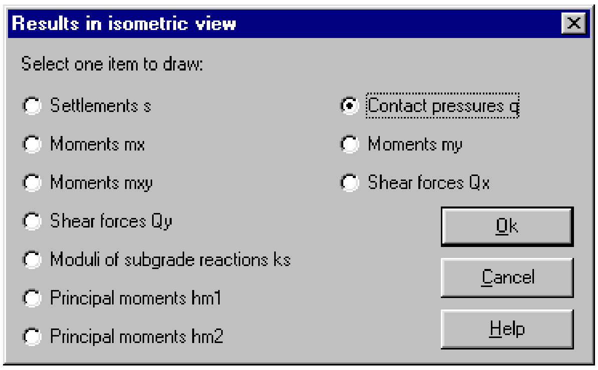


Figure E-8 "Results in isometric view" items

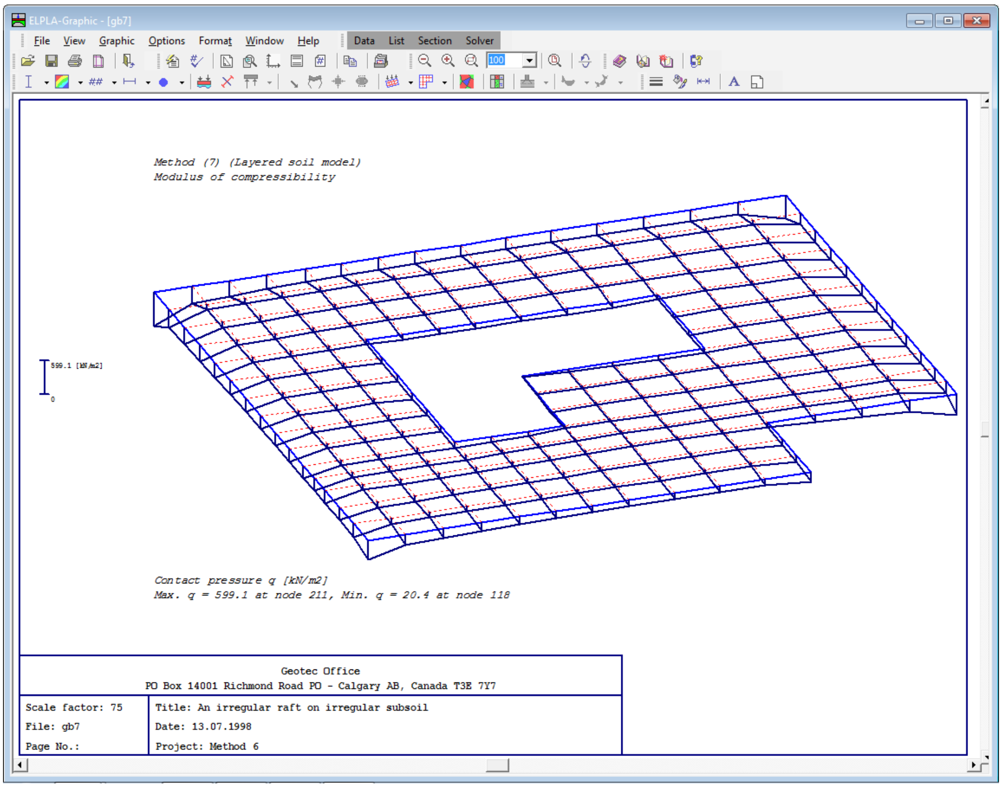


Figure E-9 Contact pressures in isometric view

6.2 Graphic Menu–"Results as contour lines"

By this command the calculated results can be contoured and shaded to reflect the variation of results. When "Results as contour lines" command is chosen, the selection Dialog box shown in Figure E-10 appears. In this dialog box select an item to draw and then click "OK" Button.

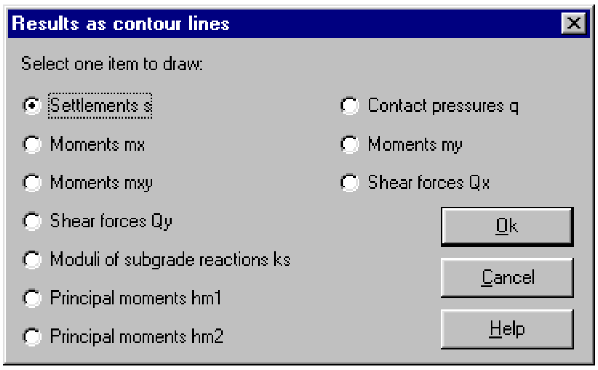


Figure E-10 "Results as contour lines" items

Contours with a constant interval

Contours can be displayed with a constant interval by checking "Contours with a constant interval" check box in "Plot parameters" form, Figure E-11. For drawing the contours, the minimum and maximum contour values, and also the contour interval are required. As an example, Figure E-12 shows the settlements as contours with a constant interval.

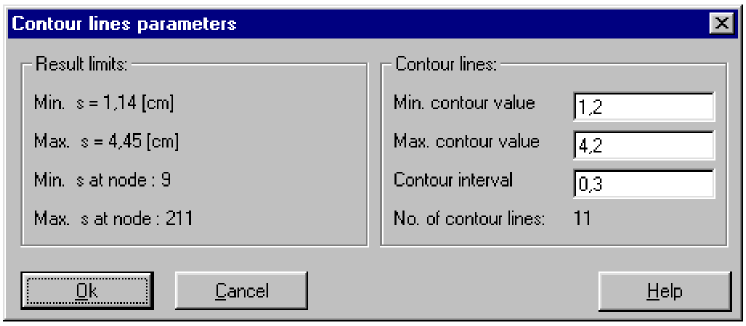


Figure E-11 "Contour lines parameters" Dialog box

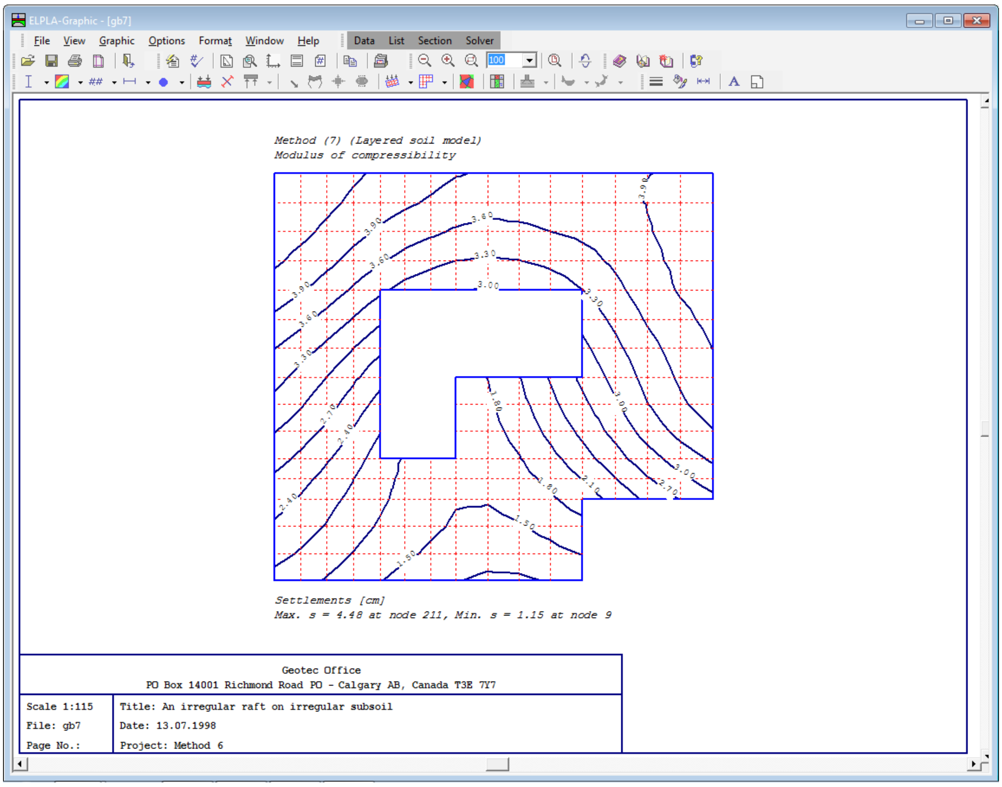


Figure E-12 Settlements as contour lines with a constant interval

Contours with a variable interval

Contours can be displayed with a variable interval by making "Contours with a constant interval" check box unchecked in "plot parameters" form, Figure E-41. For drawing the contour lines with a variable interval, the contour values are required to define in the form "Contour lines", Figure E-13.

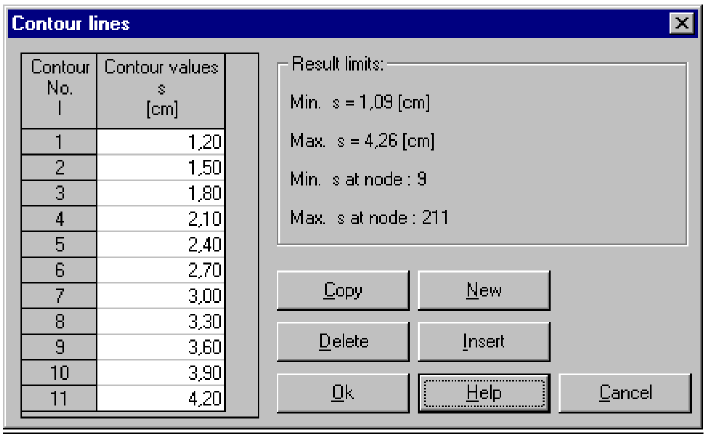


Figure E-13 "Contours with a variable interval" Dialog box

Color contours

Contours can be colored and shaded by checking "Color contours" check box in "Plot parameters" form, Figure E-41. The number of shaded colors is 13, start at color blue, which represents the minimum value and end at color red, which represents the maximum value. The lines between shaded contours will be displayed, if "Display lines" check box in "Plot parameters" form is checked. As an example, Figure E-14 shows the settlements in color contours.

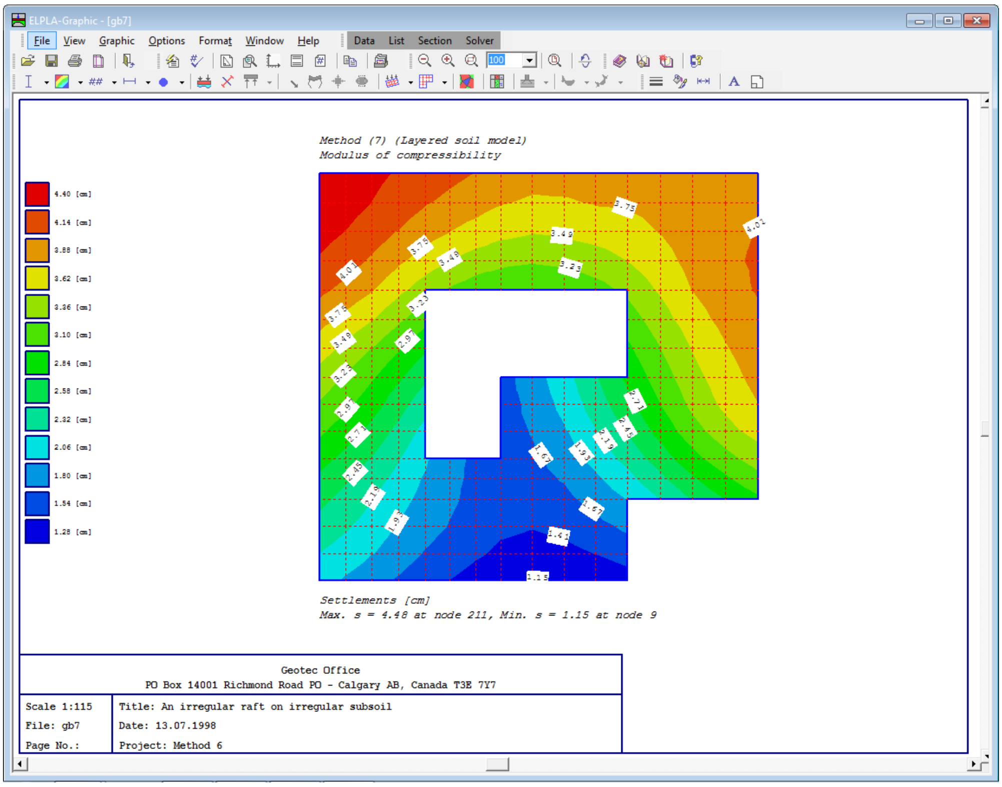


Figure E-14 Color contours for settlements

6.3 Graphic Menu–"Result values in the plan" command

The calculated results can be tabulated on the mesh. When this command is chosen, the selection Dialog box shown in Figure E-15 appears. Select an item to draw and then click "OK" Button. As an example, Figure E-16 shows the values of top reinforcement in *x*-direction on the mesh.

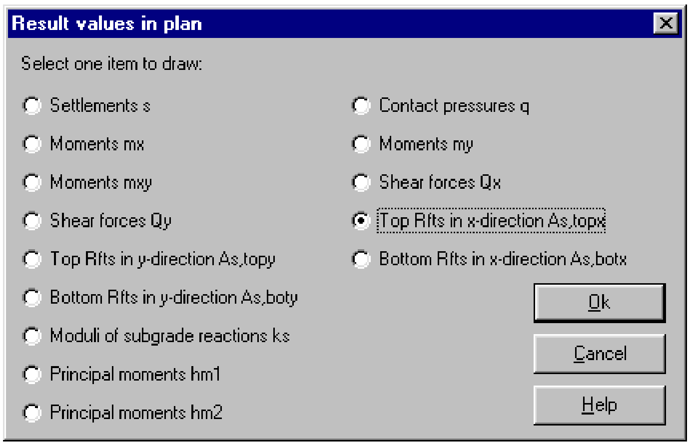


Figure E-15 "Result values in the plan" items

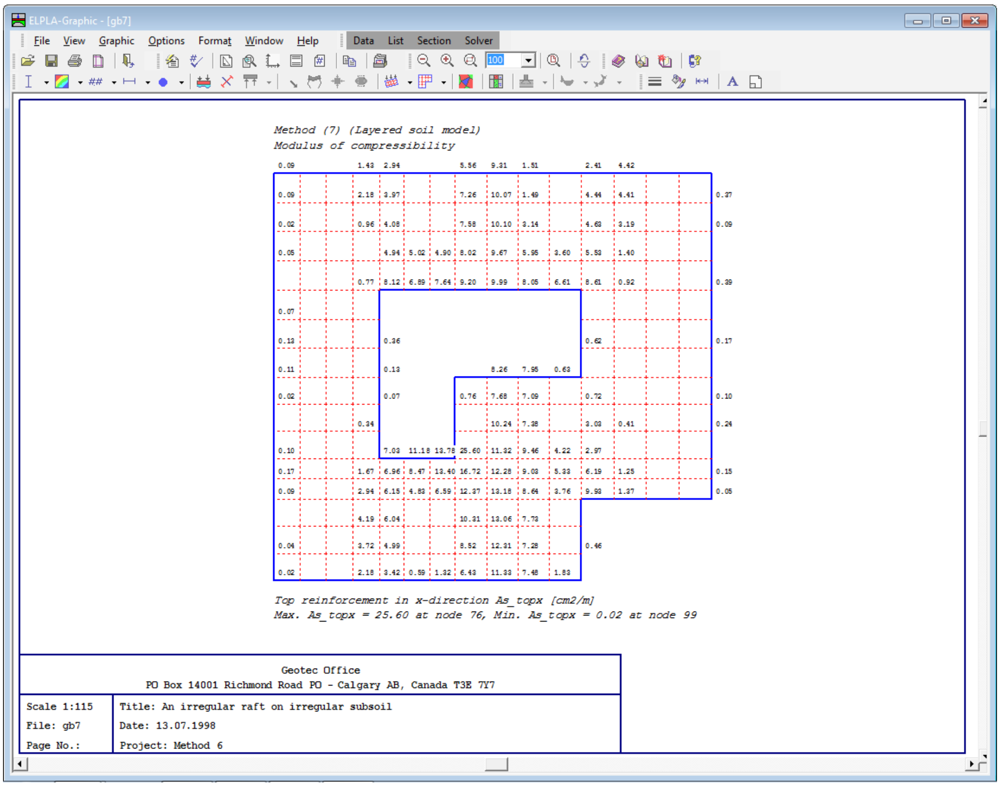


Figure E-16 Tabulation of the top reinforcement in x-direction on the mesh

6.4 Graphic Menu–"Distribution curves of results in the plan" command

By "Distribution curves of results in the plan" command the calculated results can be plotted as distribution curves on the mesh. When "Distribution curves of results in the plan" command is chosen, the selection Dialog box shown in Figure E-17 appears. In this Dialog box select an item to draw and then click "OK" Button. As an example, Figure E-18 shows the distribution curves of moment m_x on the slab.

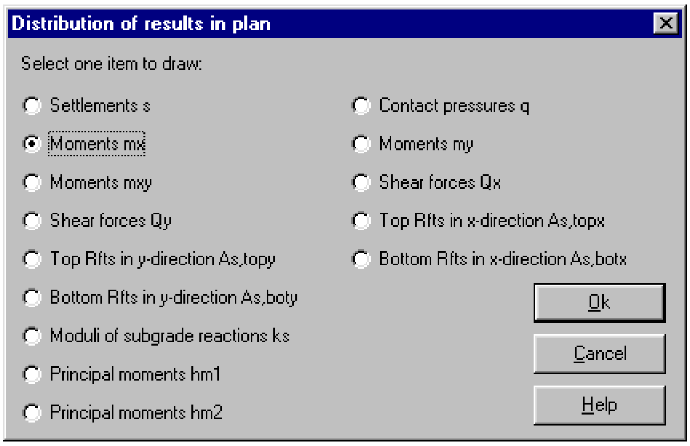


Figure E-17 "Distribution curves of results in the plan" items

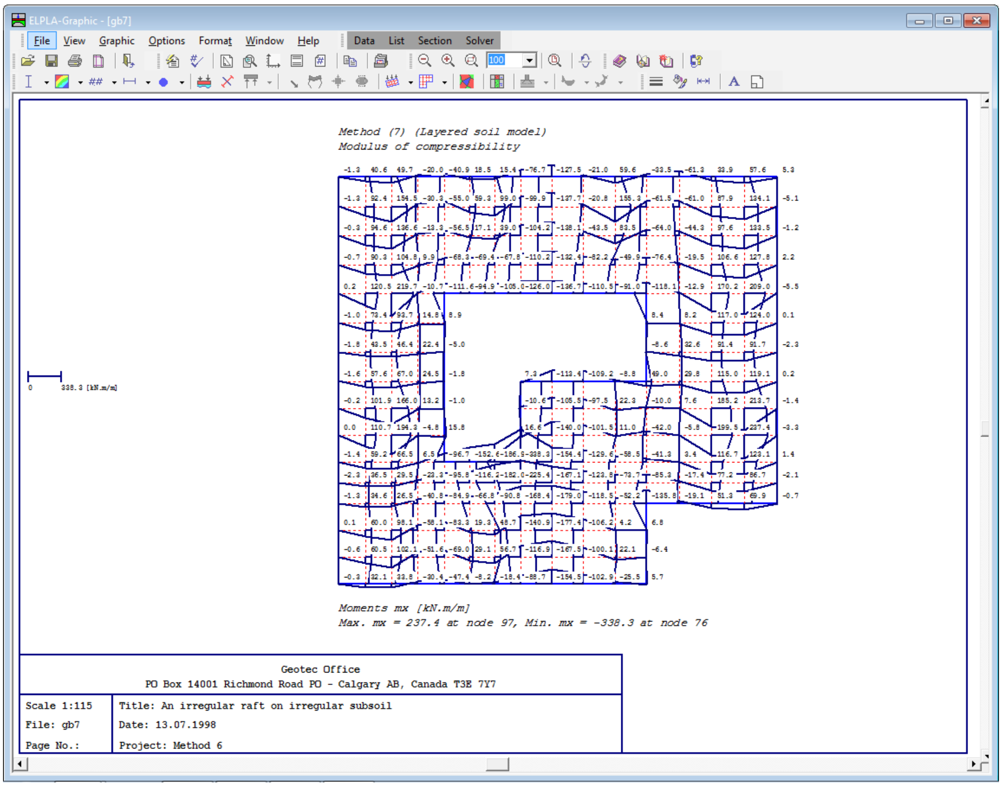


Figure E-18 Distribution curves of moment m_x on the slab

6.5 Graphic Menu–"Results as circular diagrams" command

By this command the calculated results can be plotted as circular diagrams on the mesh. When "Results as circular diagrams" command is chosen, the selection Dialog box shown in Figure E-19 appears. In this Dialog box select an item to draw, then click "OK" Button. As an example, Figure E-20 shows the circular diagrams for shear forces Q_x.

Color circles

Circular diagrams can be colored and shaded by checking "Color circles" check box in "Plot parameters" form, Figure E-41.

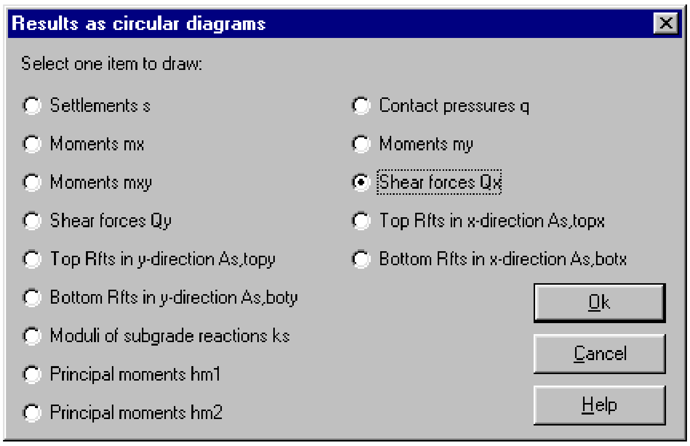


Figure E-19 "Results as circular diagrams" items

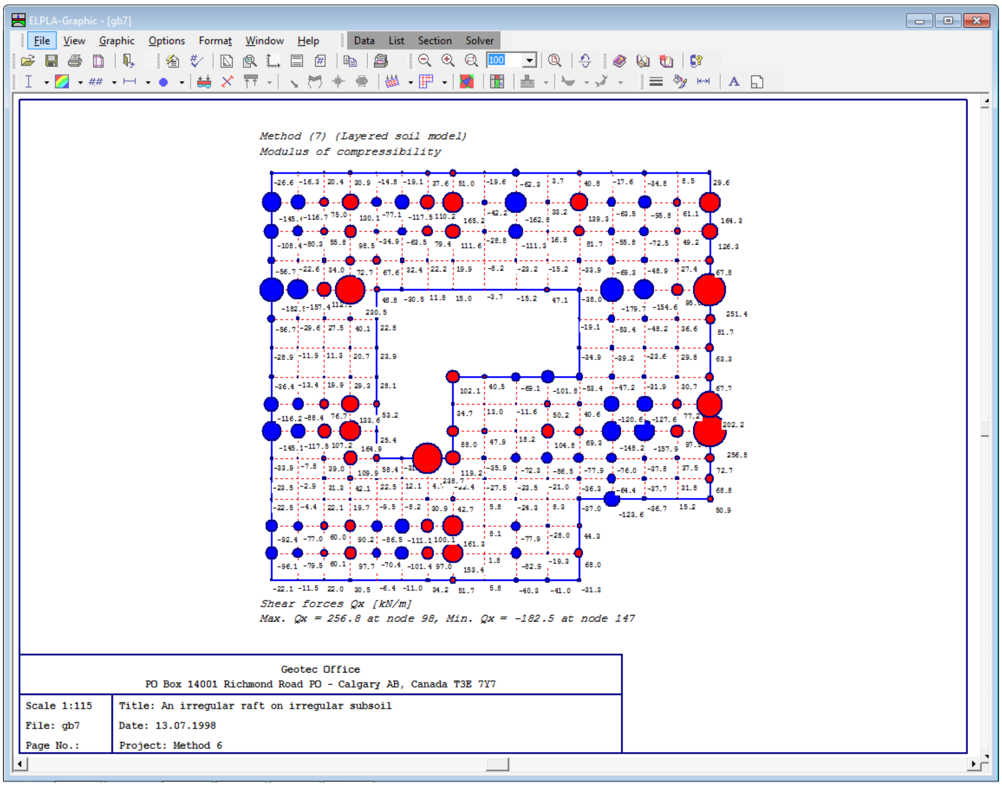


Figure E-20 Circular diagrams for shear forces Q_x

6.6 Graphic Menu–"Deformation" command

By "Deformation" command the slab deformation can be displayed as a deformed mesh plot as shown in Figure E-21.

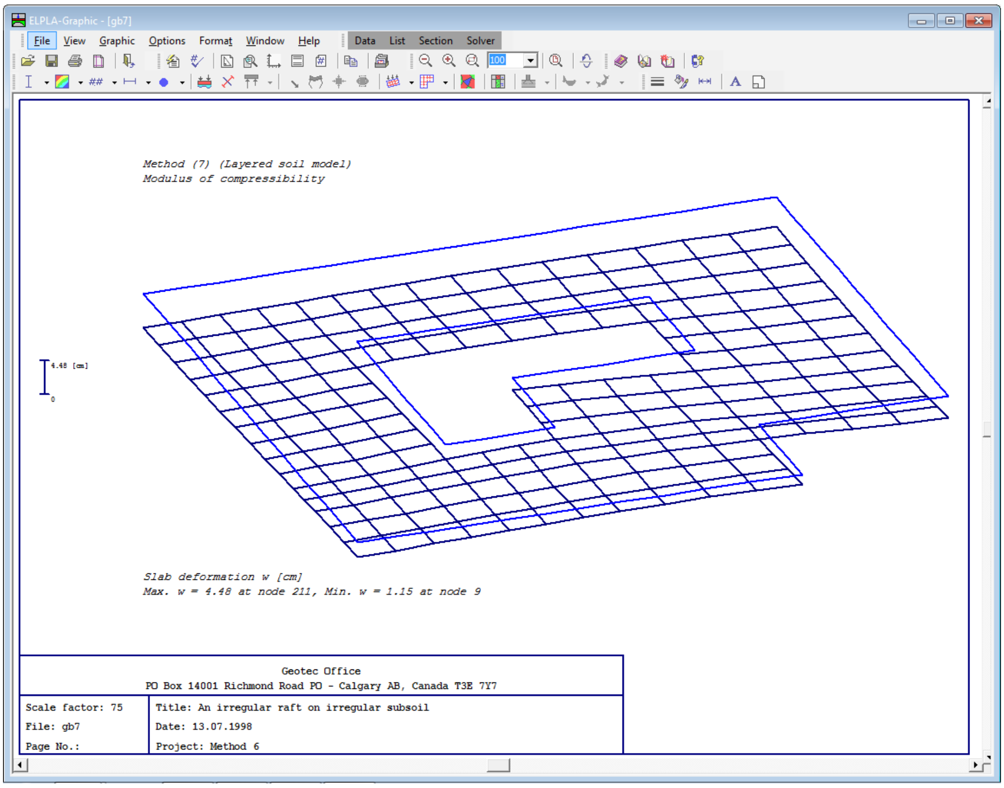


Figure E-21 Slab deformation

6.7 Graphic Menu–"Principal moments as streaks" command

By "Principal moments as streaks" command the principal moments h_{m1} and h_{m2} can be displayed as streaks as shown in Figure E-22.

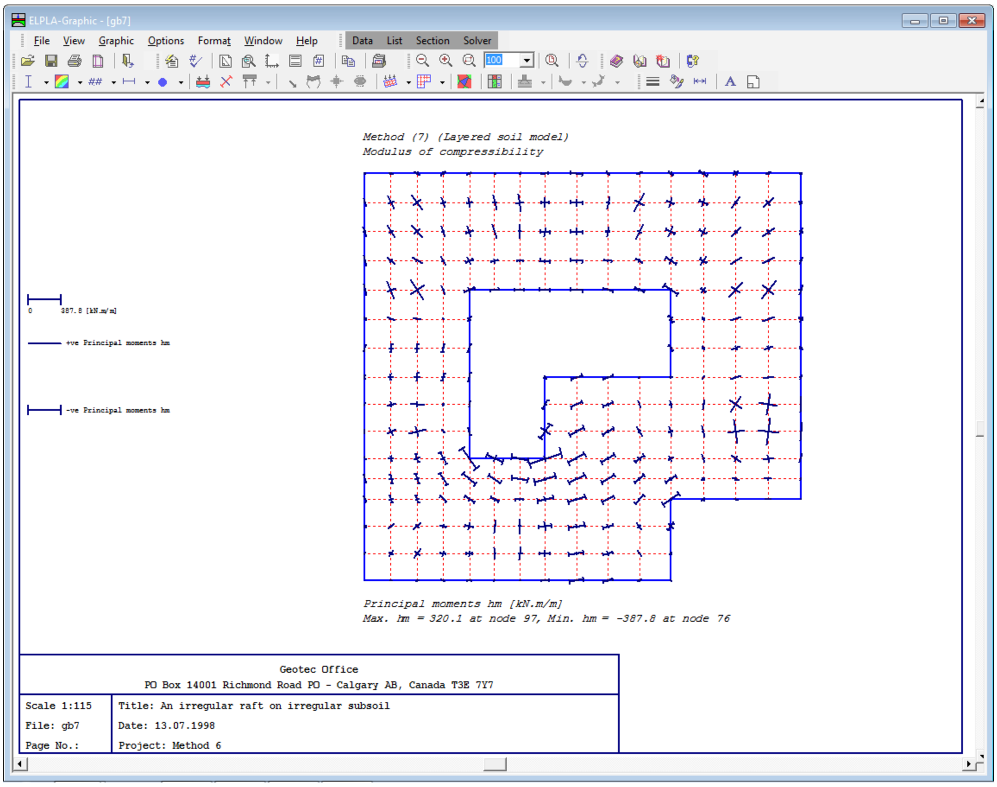


Figure E-22 Principal moments as streaks

6.8 Graphic Menu–"Support reactions as arrows" command

By "Support reactions as arrows" command the support reactions V, M_x and M_y can be displayed in isometric shape as arrows. When this command is chosen, the selection Dialog box shown in Figure E-23 appears. In this Dialog box select an item to draw, then click "OK" Button. As an example, Figure E-24 shows the support reactions V as arrows.

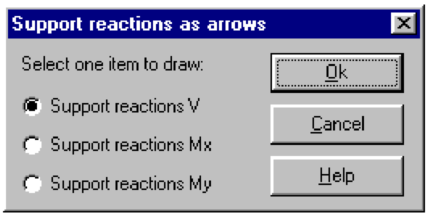


Figure E-23 "Support reactions as arrows" items

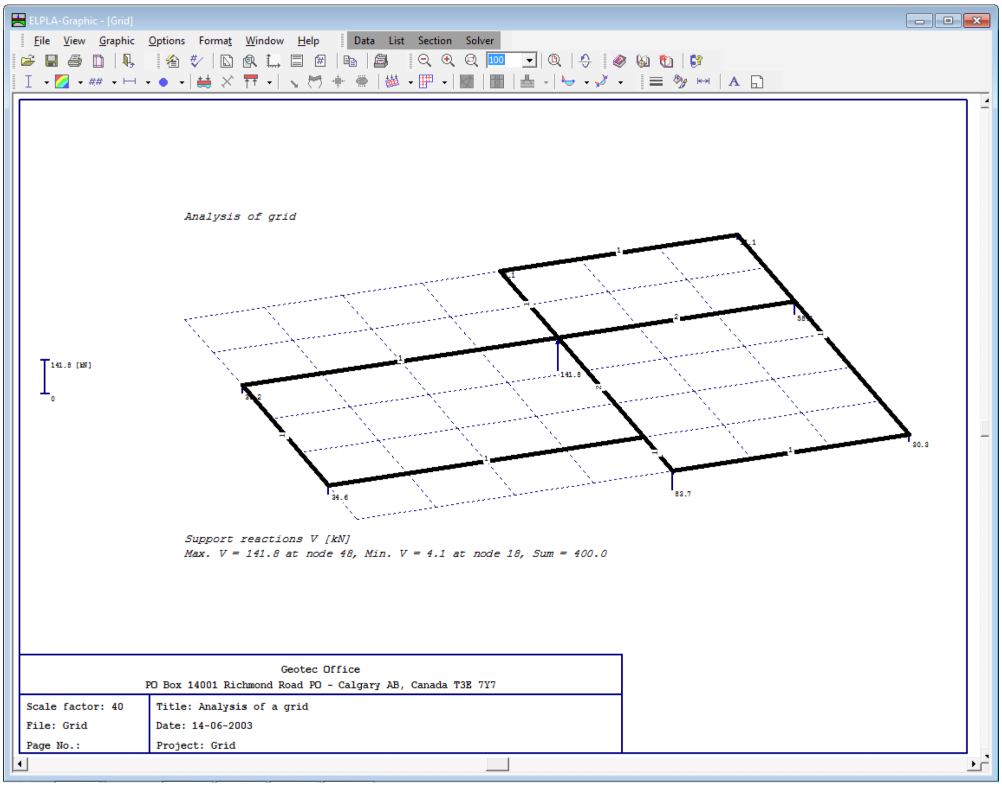


Figure E-24 Support reactions V as arrows

6.9 Graphic Menu–"Soil deformation as vectors" command

By "Soil deformation as vectors" command the soil deformation can be displayed as vectors. When this command is chosen, the selection Dialog box shown in Figure E-25 appears. In this Dialog box select an item to draw, then click "OK" Button. As an example, Figure E-26 shows the soil deformation as vectors.

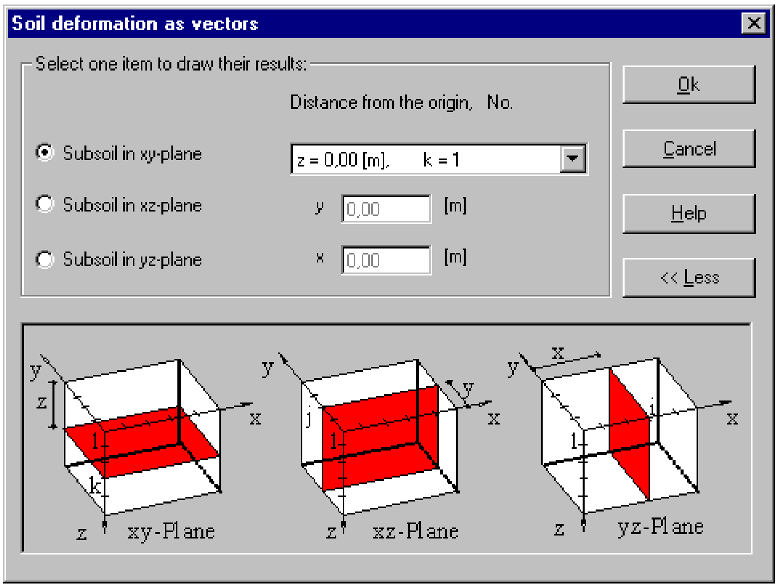


Figure E-25 "Soil deformation as vectors" items

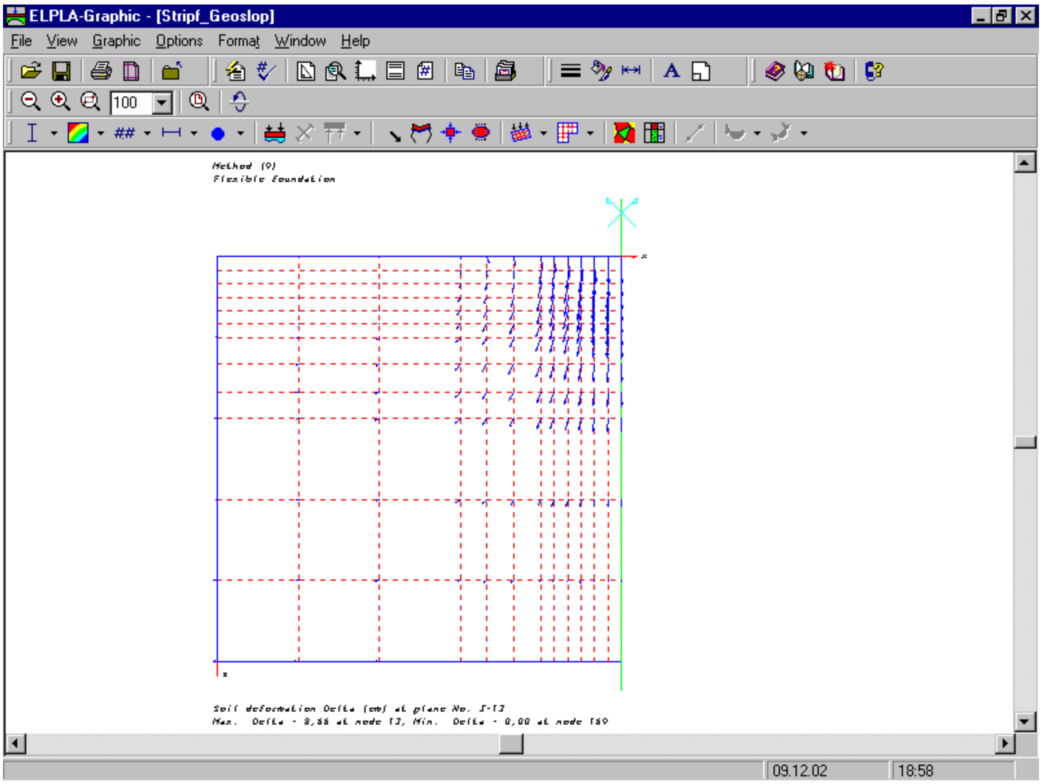


Figure E-26 Soil deformation as vectors

6.10 Graphic Menu–"Soil deformation as deformed mesh" command

By this command the soil deformation can be displayed as deformed mesh. As an example, Figure E-27 shows the soil deformation as deformed mesh.

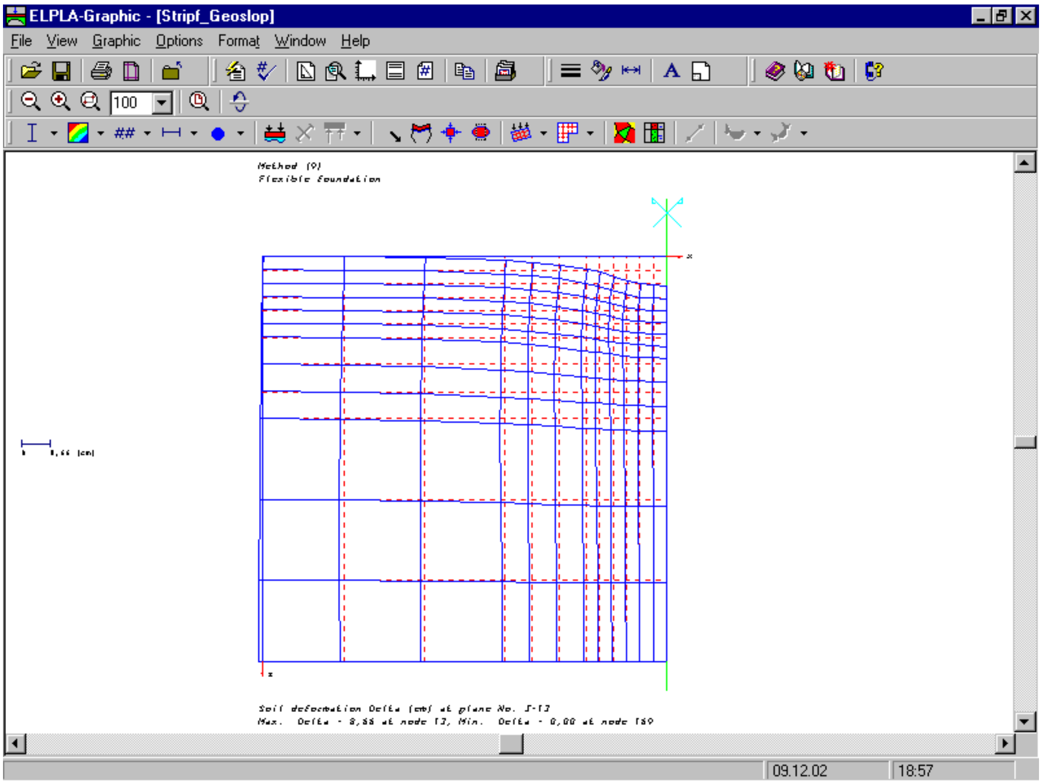


Figure E-27 Soil deformation as deformed mesh

6.11 Graphic Menu–"Principal soil stresses as streaks" command

Here the principal soil stresses can be displayed as streaks. As an example, Figure E-28 shows the principal soil stresses as streaks.

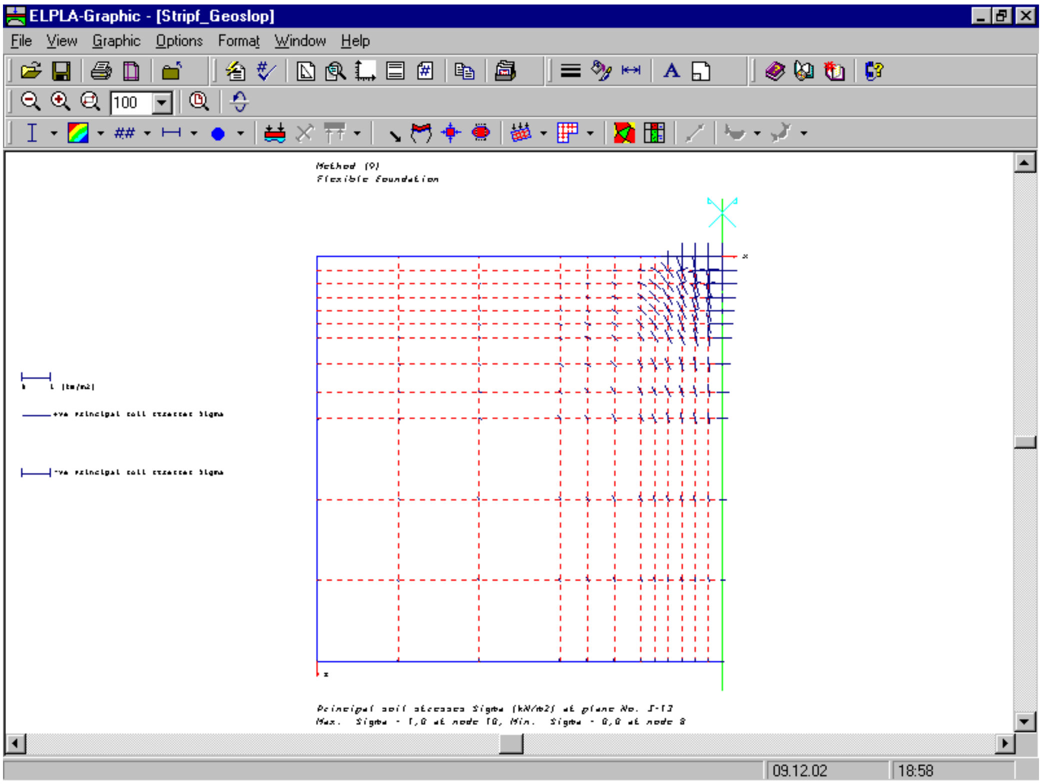


Figure E-28 Principal soil stresses as streaks

6.12 Graphic Menu–"Principal soil strains as streaks" command

By this command the Principal soil strains can be displayed as streaks. As an example, Figure E-29 shows the principal soil strains as streaks.

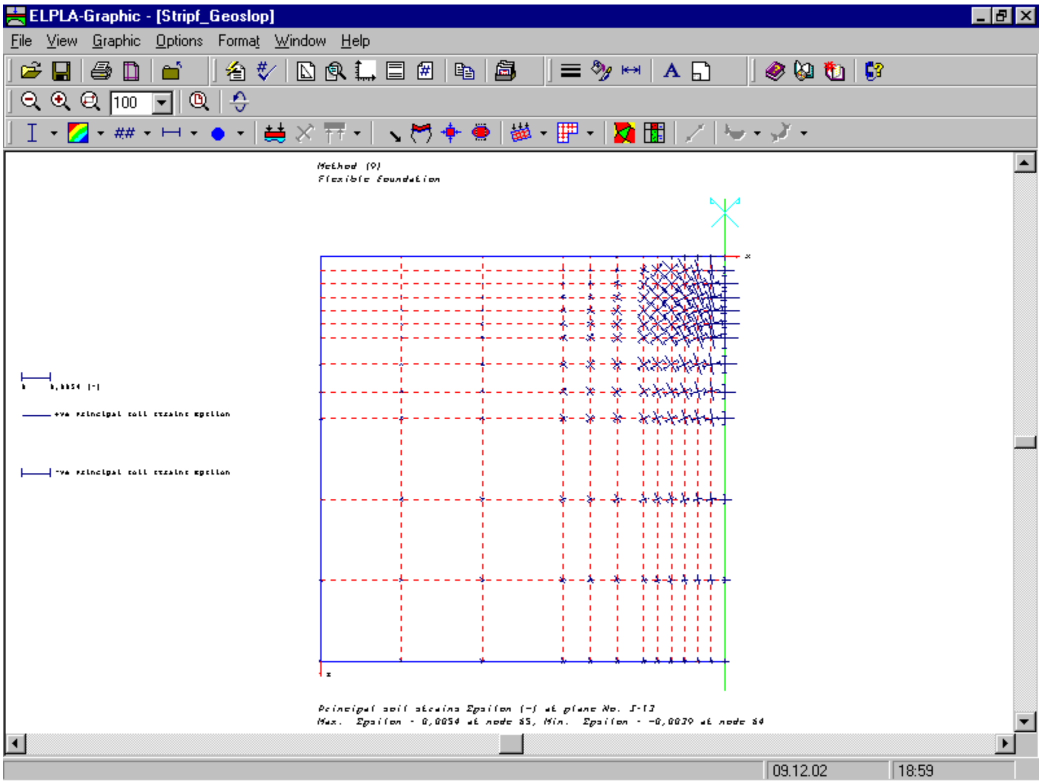


Figure E-29 Principal soil strains as streaks

6.13 Graphic Menu–"Data in isometric view" command

By "Data in isometric view" command the input data can be displayed on the screen and plotted in isometric shape. When "Data in isometric view" command is chosen, the selection Dialog box shown in Figure E-30 appears. In this Dialog box select an item to draw, then click "OK" Button. As an example, Figure E-31 shows the loading in isometric view.

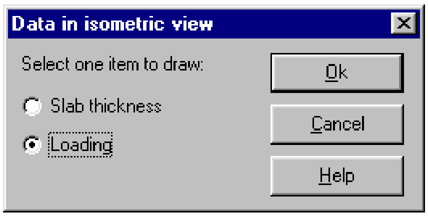


Figure E-30 "Data in isometric view" items

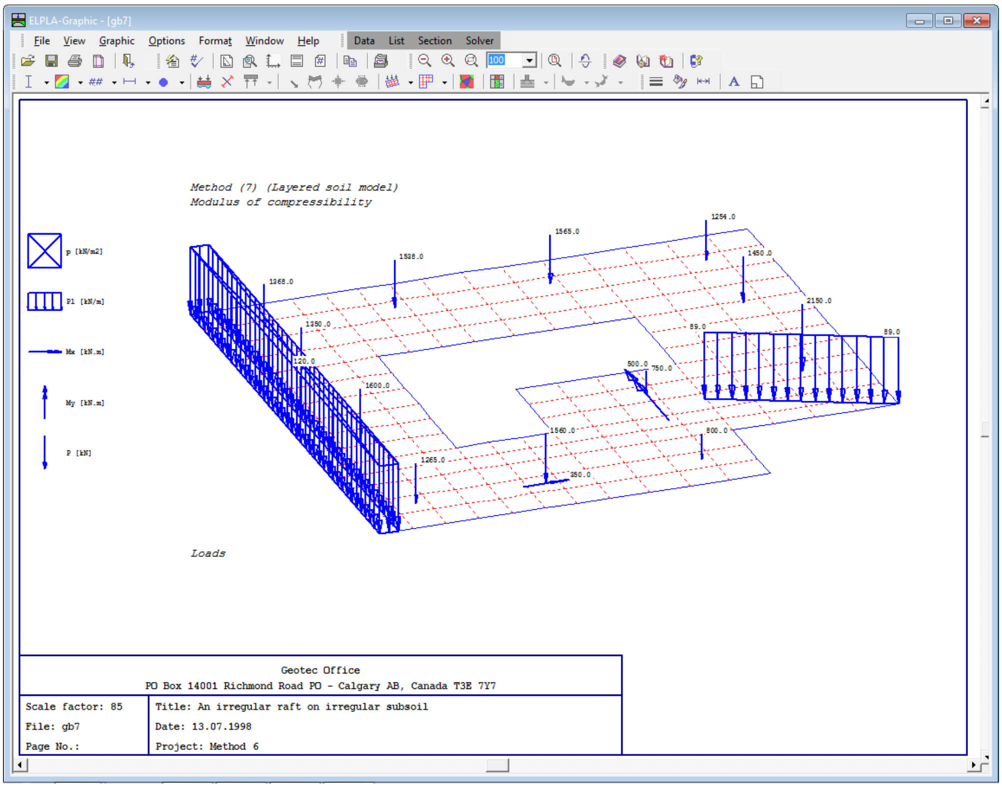


Figure E-31 Loading in isometric view

6.14 Graphic Menu–"Data in the plan" command

By "Data in the plan" command the input data can be displayed in the plan view. When "Data in the plan" command is chosen, the selection Dialog box shown in Figure E-32 appears. In this Dialog box select an item to draw and click "OK" Button. As an example, Figure E-33 shows the loading in the plan.

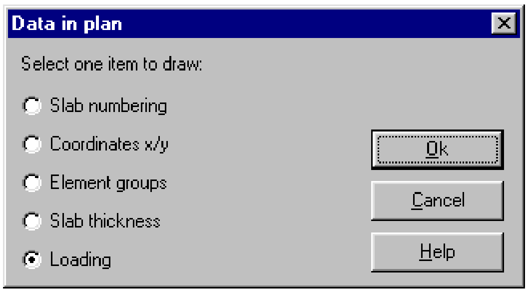


Figure E-32 "Data in the plane" items

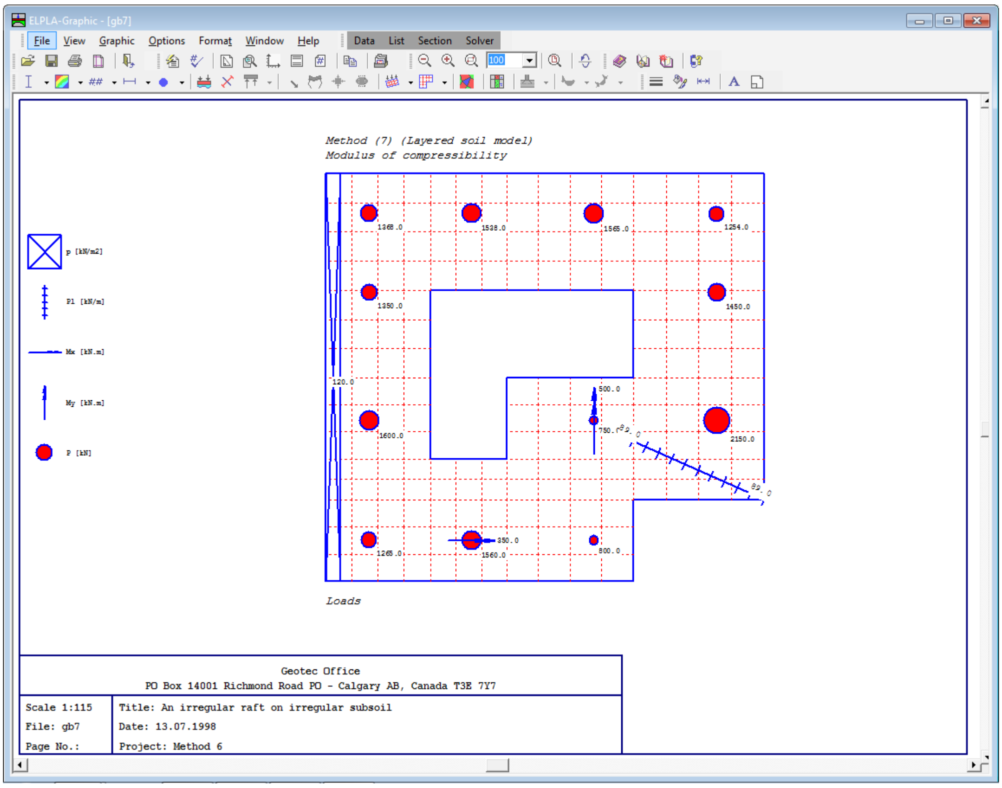


Figure E-33 Loading in the plan

6.15 Graphic Menu–"Boring locations" command

By "Boring locations" command the boring locations in global coordinates can be displayed with the slab geometry as shown in Figure E-34.

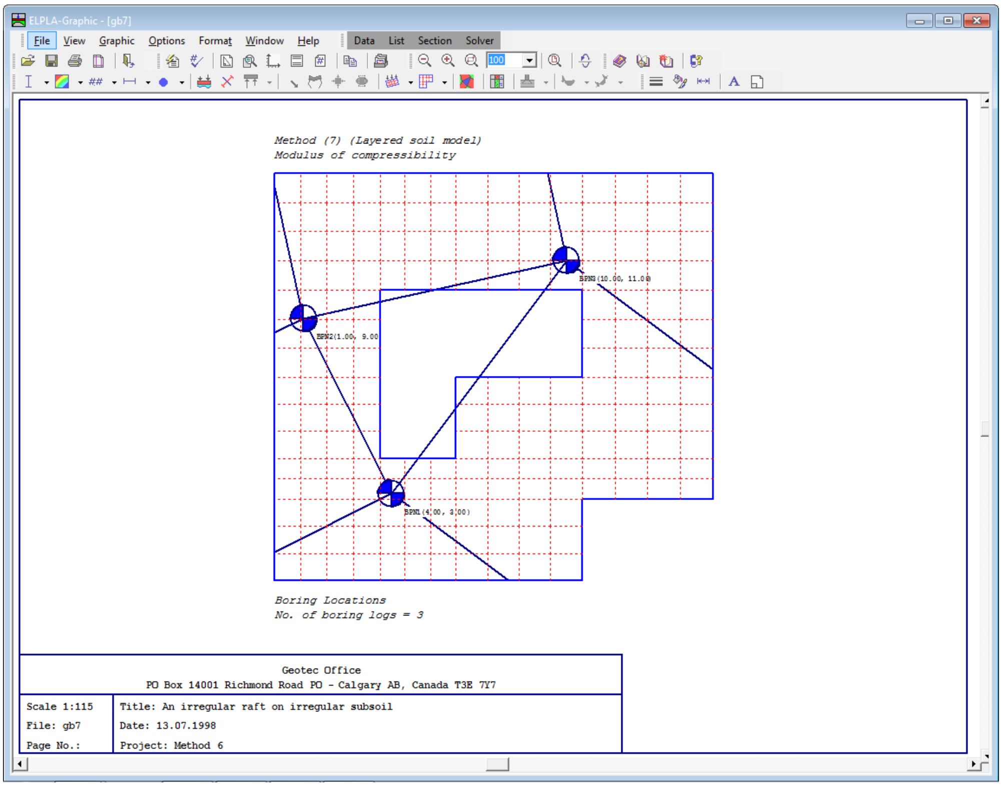


Figure E-34 Location of boring with zone types

6.16 Graphic Menu–"Boring logs/ Limit depth" command

Here the boring logs (boring layers, soil material, water table and limit depth) can be displayed through ELPLA-Boring as shown in Figure E-35. The stress on soil under a specified point on the foundation with effective stress can be also displayed beside the corresponding boring log through ELPLA-Boring as shown in Figure E-36. The stress on soil is used to determine the limit depth of the soil layers. To get information about ELPLA-Boring, see the User’s Guide of ELPLA-Boring.

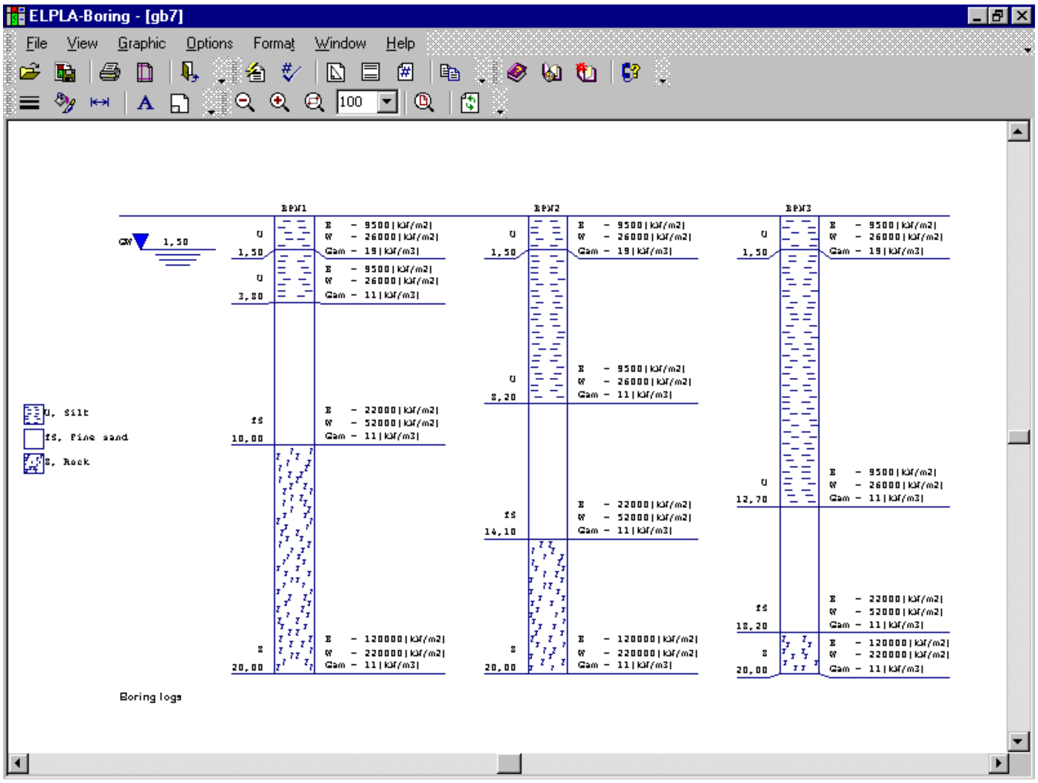


Figure E-35 Boring logs with multi-layers with different soil material

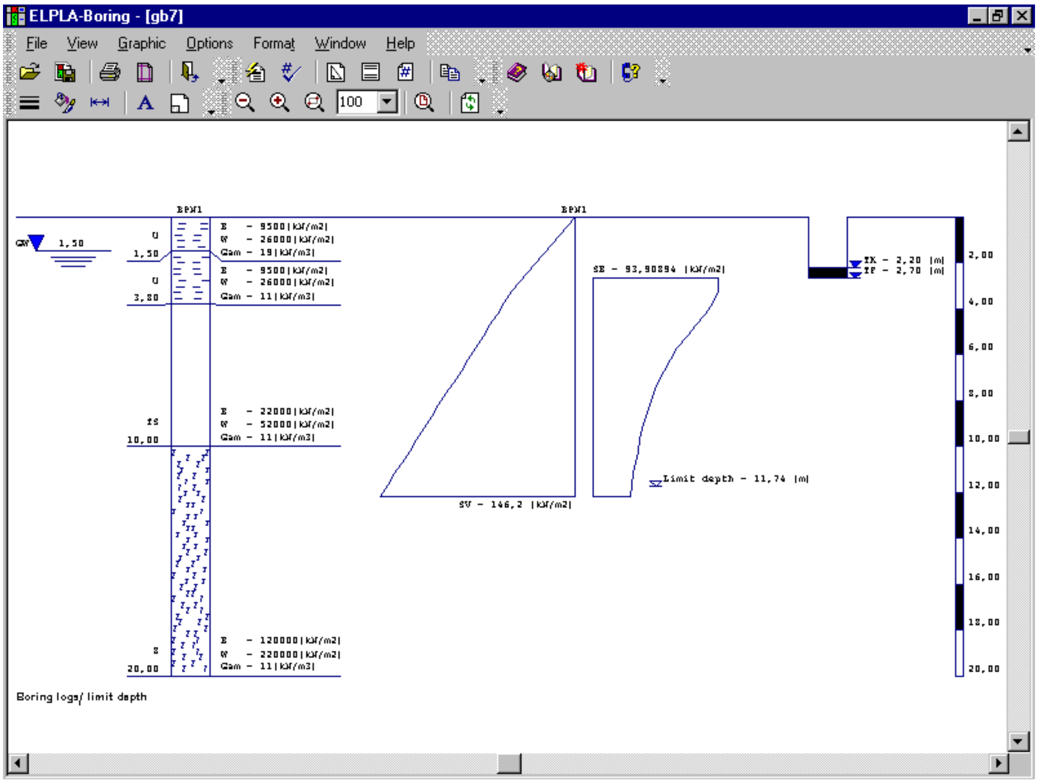


Figure E-36 Limit depth of the soil layers

6.17 Graphic Menu–"Beam results" command

In ELPLA the finite element method was used to analyze both of the raft and girders, in which the raft was represented by plate bending elements while the girders were represented by grid elements. Furthermore, a combination between plate elements and grid elements may be used to represent ribbed rafts. If girders are considered in the project, "Beam results" command will appear in the graphic menu.

"Beam results" command has the following options (commands):

- Distribution of internal forces in the plan
- Internal forces in isometric view

"Distribution of internal forces in the plan"–Option

This option displays the beam internal forces (torsion moments, bending moments and shear forces) with the slab geometry in the plan. The selection Dialog box shown in Figure E-37 appears. In this Dialog box select an item to draw and click "OK" Button. As an example, Figure E-38 shows the distribution of bending moments in the plan.

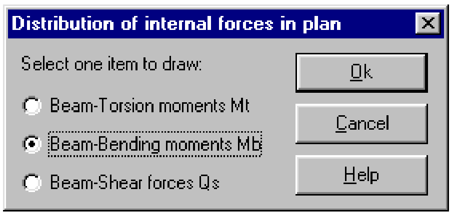


Figure E-37 "Distribution of internal forces in the plan" items

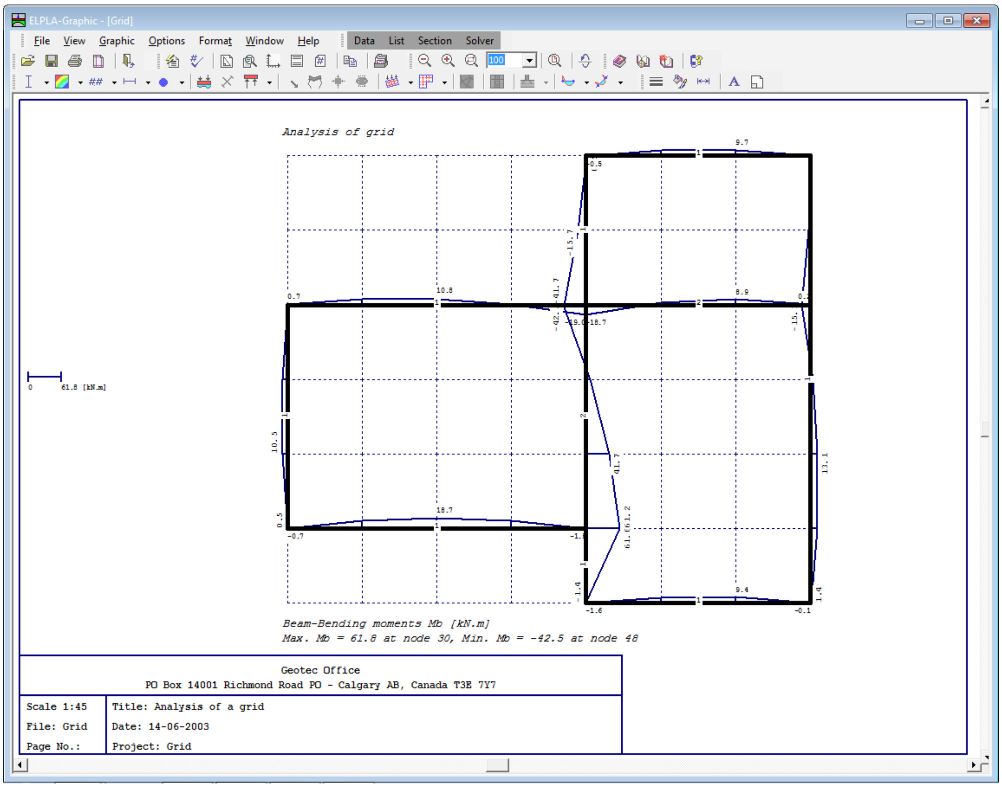


Figure E-38 Distribution of bending moments in the plan

"Internal forces in isometric view"–Option

This option displays the beam internal forces (torsion moments, bending moments and shear forces) with the slab geometry in isometric view. When "Distribution of internal forces isometric view"-Option is chosen, the selection Dialog box shown in Figure E-39 appears. In this Dialog box select an item to draw, then click "OK" Button. As an example, Figure E-40 shows the distribution of shear forces in isometric view.

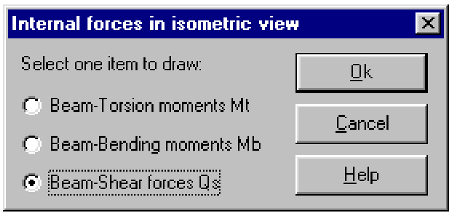


Figure E-39 "Distribution of internal forces in isometric view" items

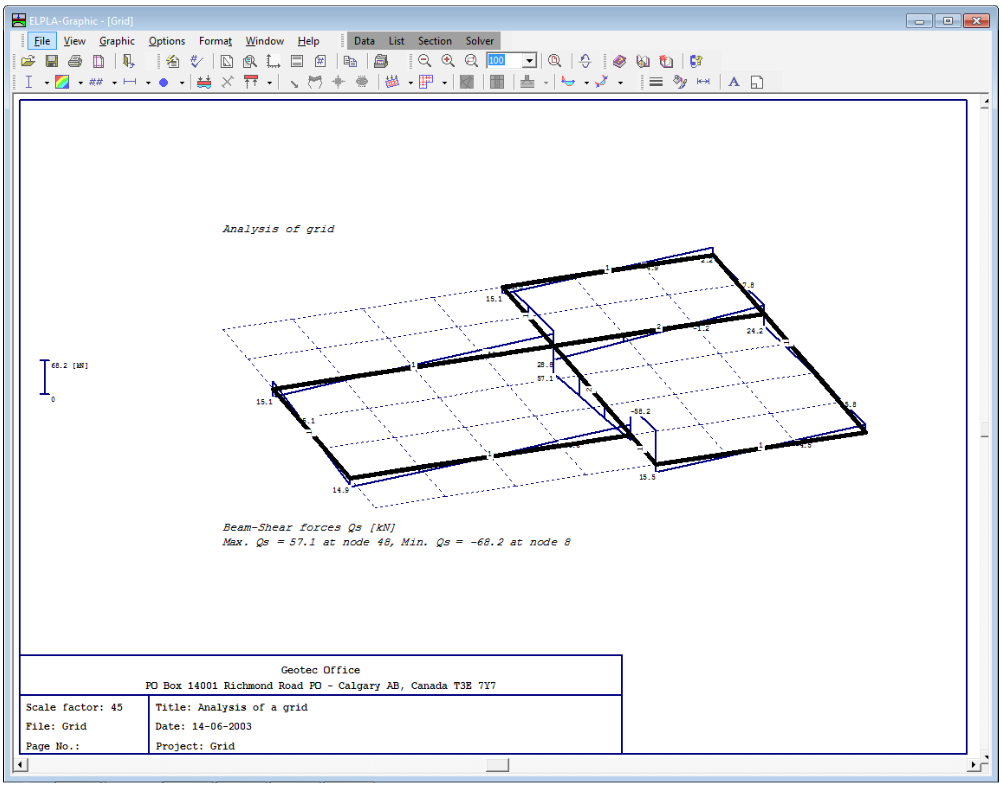


Figure E-40 Distribution of shear forces in isometric view

7 Options Menu

The Options menu has the following commands:

- Plot parameters
- Display values
- Scale
- Set range
- Axes
- Title
- Page No.
- Copy
- View grouping

7.1 Options Menu–"Plot parameters" command

Plot parameters may be set as default values by the program or be fully specified by the user. By "Plot parameters" command the following plot parameters can be specified (Figure E-41):

- Display FE-Net, Axes, slab dimensions, FE-Net in separated elements, reference points and lines, axis arrows
- Pages with or without a frame
- Color element groups and slab thickness
- Color section of punching shear
- Color girders

- Draw girder thickness
- Color circles of circular diagrams. If "color circles" is checked, the positive circles are red and the negative circles are blue
- Color contours with or without display contour lines. If "color contours" is checked, the contours will have 13 constant intervals with 13 colors. Colors start at color blue, which represents the minimum value and end at color red representing the maximum value
- Writing factor. If "Writing factor" is checked, the intensity of numbers on the contour lines will be defined
- Display contours with a constant interval
- Display boring subareas boundary
- Color boring subareas
- Mark boring subareas
- Draw neighboring foundations
- Draw block element
- Draw element border
- Color solid element
- Display FE-Net in separated elements
- Display reference points/ lines

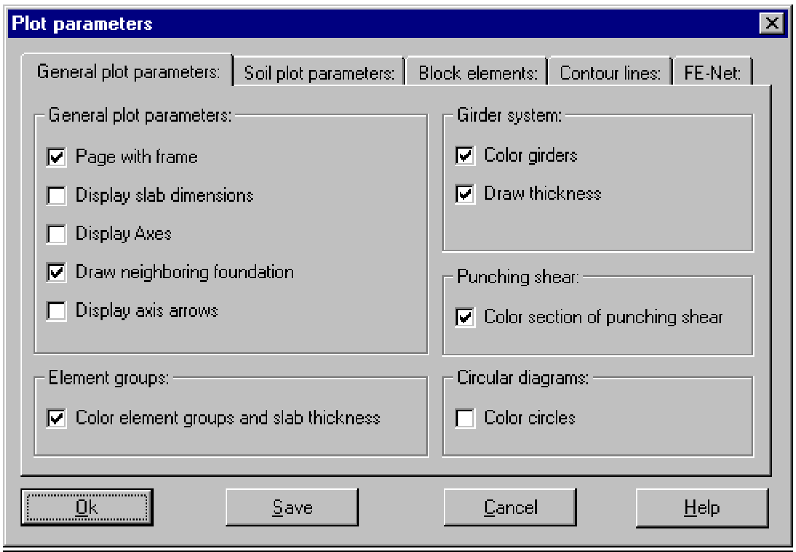


Figure E-41 "Plot parameters" Dialog box

7.2 Options Menu–"Display values" command

By "Display values" command the values of the following items can be displayed, if desired, on the drawing (Figure E-42):

- Loads
- Boundary conditions
- Spring supports
- Element groups
- Girder system
- Piles
- Internal forces of girders
- Axes

- Support reactions as arrows
- Slab thickness
- Isometric views
- Contour lines
- Circular diagrams
- Distribution curves of results
- Soil deformation as vectors
- Soil deformation as deformed mesh
- Punching shear
- Column types

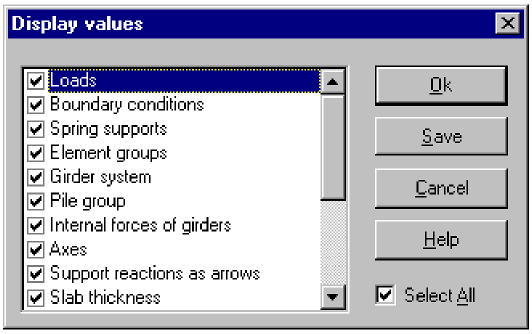


Figure E-42 "Display values" Dialog box

7.3 Options Menu–"Scale" command

By "Scale" command the scale factor of the drawing can be defined as shown in Figure E-43. The defaulted value for the scale factor is chosen to pass the active printer-paper format. By graphical drawing in the plan the scale is changed to scale factor, "scale factor" = "scale 1: ".

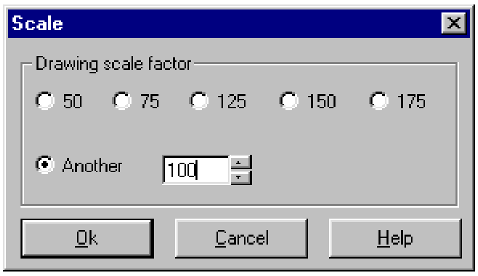


Figure E-43 "Scale" Dialog box

7.4 Options Menu–"Set range" command

By "Set range" command the area of the slab, that will be drawn, can be defined. In the Dialog box shown in Figure E-44 define ranges, which bound this slab area.

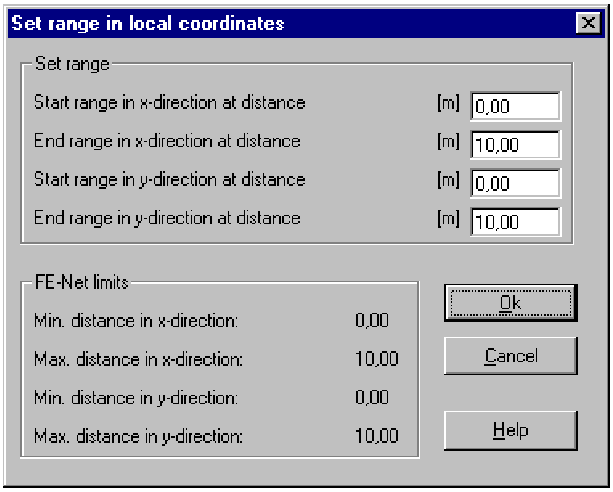


Figure E-44 "Set range" Dialog box

7.5 Options Menu–"Axes" command

By "Axes" command scaled axes can be defined on the drawing, Figure E-45.

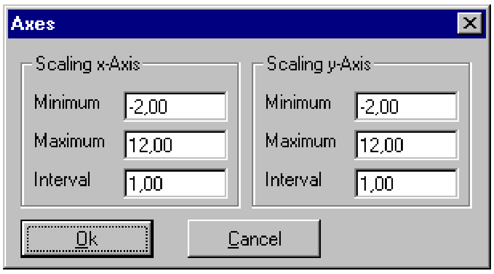


Figure E-45 "Axes" Dialog box

7.6 Options Menu–"Title" command

By "Title" command the text data (two lines text above the drawing and two lines text under the drawing) can be defined, Figure E-46.

Default texts are:

- A-Title1: Method No.
- A-Title2: Name of method
- U-Title1: Name of drawing
- U-Title2: No. of nod columns and nod rows that define the drawing

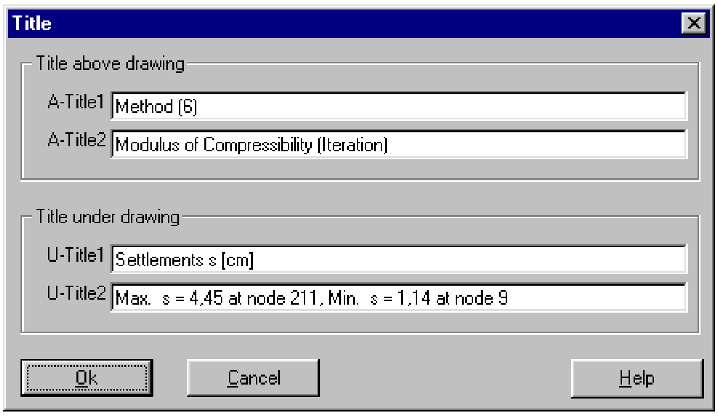


Figure E-46 "Title" Dialog box

7.7 Options Menu–"Page No." command

By "Page No." command the page No. can be defined, Figure E-47.

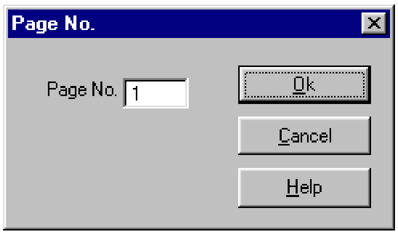


Figure E-47 "Page No." Dialog box

7.8 Options Menu–"Copy" command

By "Copy" command the current drawing can be copied in Metafile-Format to Clipboard. Then it can be inserted directly to other Windows programs such as Word and AutoCAD, Figure E-48.

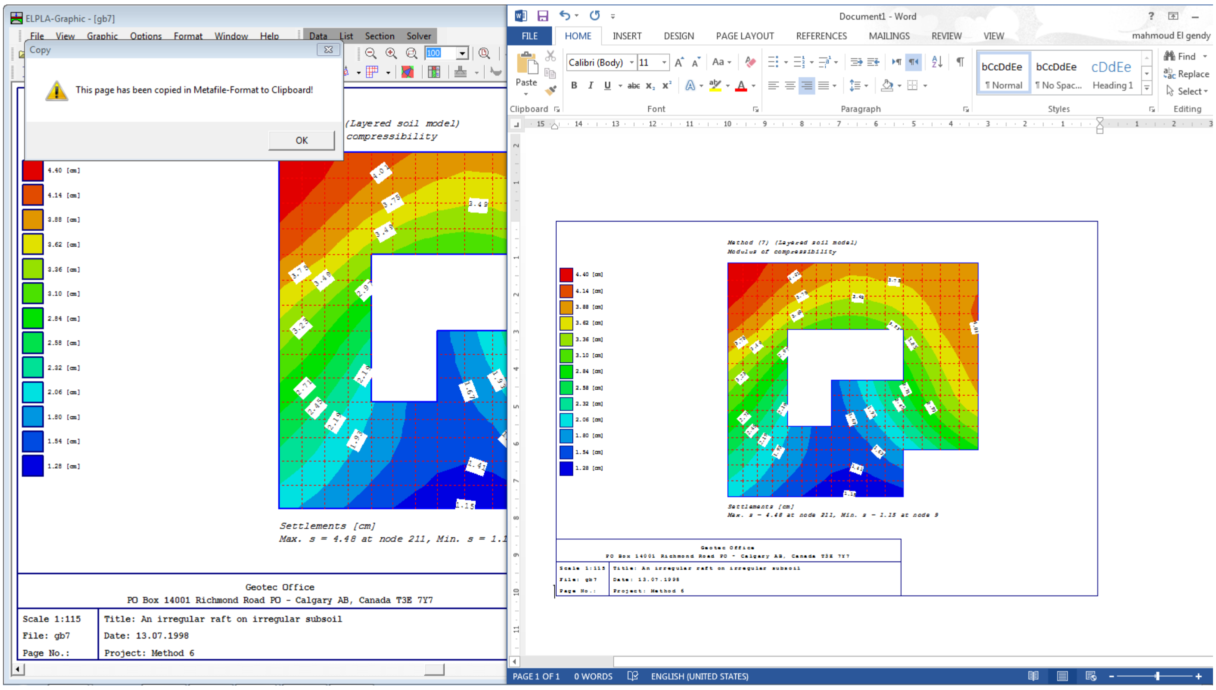


Figure E-48 Drawing can be copied to other Windows programs

7.9 Options Menu–"View grouping" command

Draw a group of data with results or a group of data together in one presentation (contour lines of settlements with loads and slab thickness, or loads with boring locations), Figure E-49.

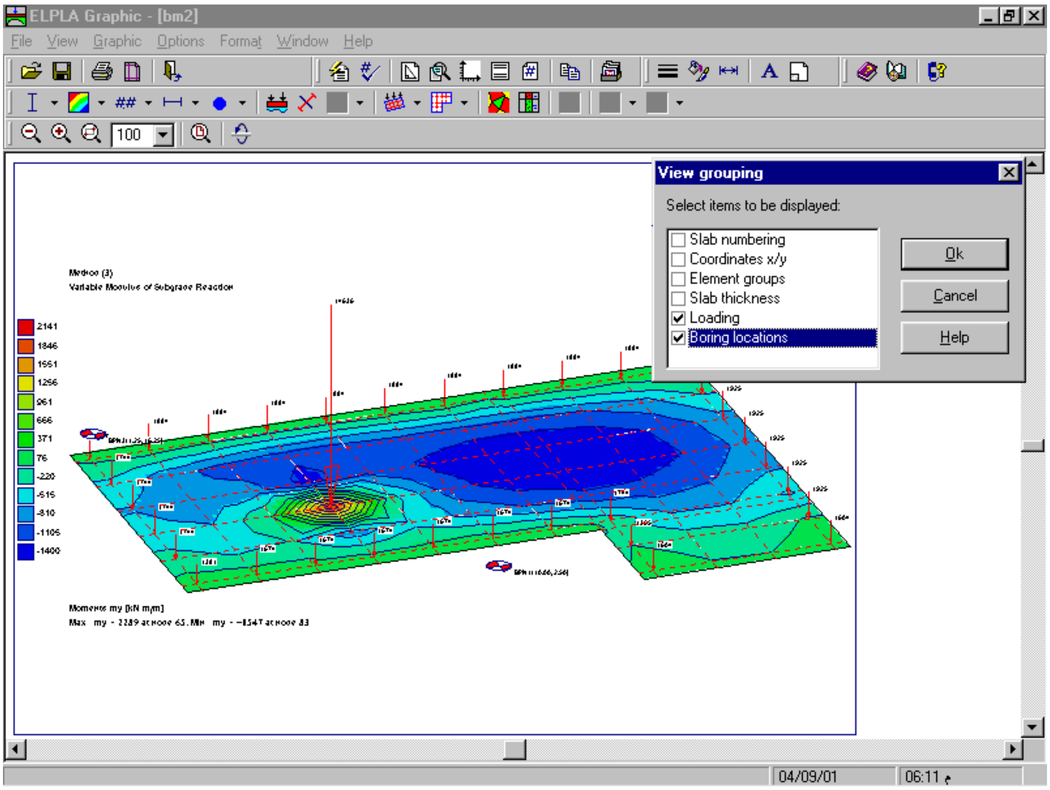


Figure E-49 Presentation of a group of data with results

8 Format Menu

The Format menu has the following commands:

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

8.1 Format Menu–"Line formats" command

By "Line formats" command the color, style and thickness of drawing lines can be defined, Figure E-50. The way a line is drawn depends on the setting of the color and style properties. There are available 15 different colors and 5 styles for line formats. The following list shows the available lines, which can be formatted:

- Slab boundary
- Elements
- Isometric view
- Distribution curves of results
- Contour lines

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

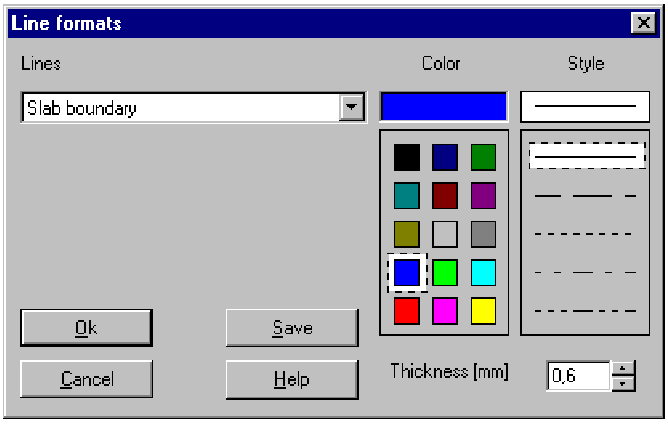


Figure E-50 "Line formats" Dialog box

8.2 Format Menu–"Fill color" command

By "Fill color" command the fill color of drawing can be defined, Figure E-51. The following list shows the available items, which can be filled with a specified color:

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

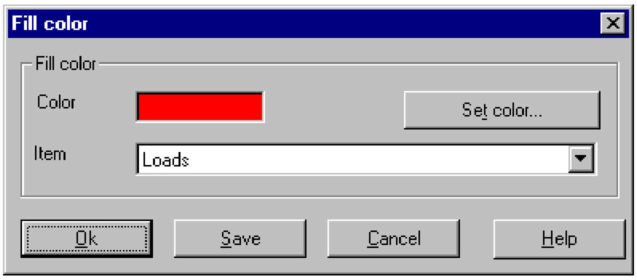


Figure E-51 "Fill color" Dialog box

8.3 Format Menu–"Max. ordinate" command

By "Max. ordinate" the max. ordinate, max. diameter, max. side, max. width and max. length for the drawing can be defined, Figure E-52.

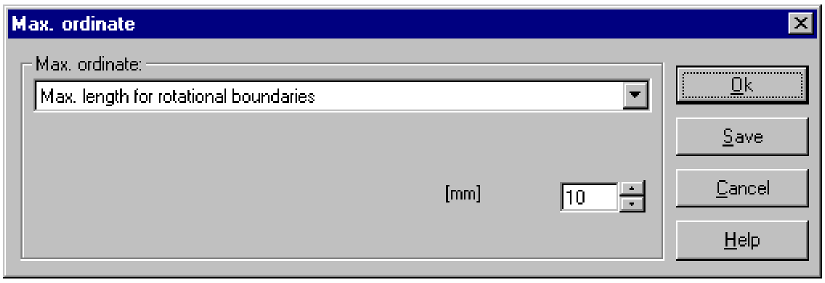


Figure E-52 "Max. ordinate" Dialog box

8.4 Format Menu–"Font" command

By "Font" command the font size (Figure E-53) and font type (Figure E-54) can be defined.

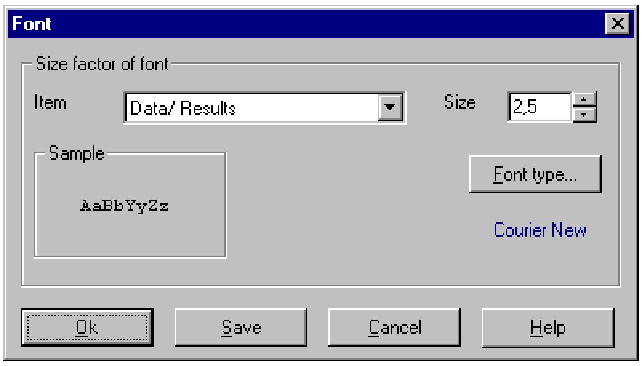


Figure E-53 "Font size" Dialog box

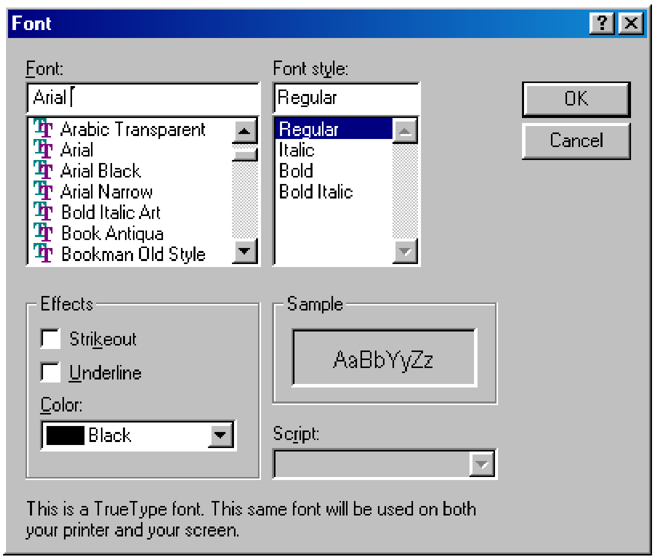


Figure E-54 "Font type" Dialog box

8.5 Format Menu–"Legend" command

By this command the height and width of legends can be specified as shown in Figure E-55.

Display legend

In "Display legend" Group box check the legend that you wish to display.

Legend size

In "Legend size" Group box the height and width of the legend can be defined.

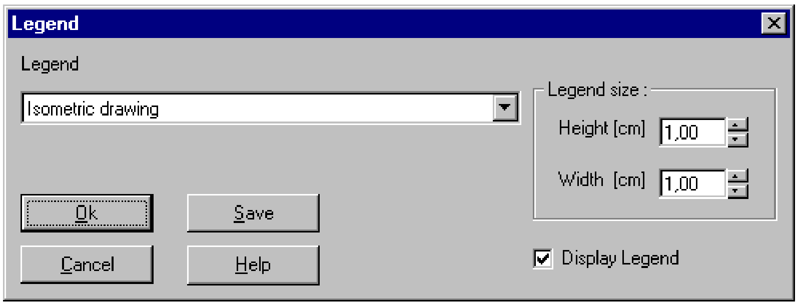


Figure E-55 "Legend" Dialog box

9 Window Menu

The Window menu has the following commands:

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

9.1 Window Menu–"Zoom in" command

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

9.2 Window Menu–"Zoom out" command

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

9.3 Window Menu–"Zoom window" command

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

9.4 Window Menu–"Zoom %" command

When you choose "Zoom %" command, the following Dialog box in Figure E-56 appears.

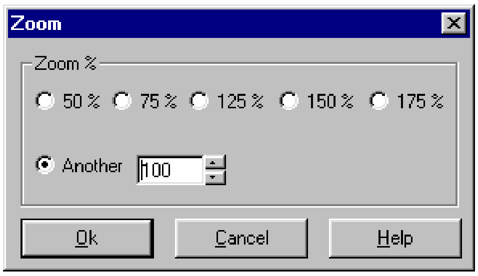


Figure E-56 "Zoom %" Dialog box

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

9.5 Window Menu–"Original size" command

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

9.6 Window Menu–"Viewing angle" command

The perspective drawing in a three dimensional view can be defined, Figure E-57.

The default viewing angle for isometric drawing, Figure E-58:

Viewing angle about x-Axis = 295
Viewing angle about y-Axis = 0
Viewing angle about z-Axis = 20

The default viewing angle for drawing in the plans:

Viewing angle about x-Axis = 0
Viewing angle about y-Axis = 0
Viewing angle about z-Axis = 0

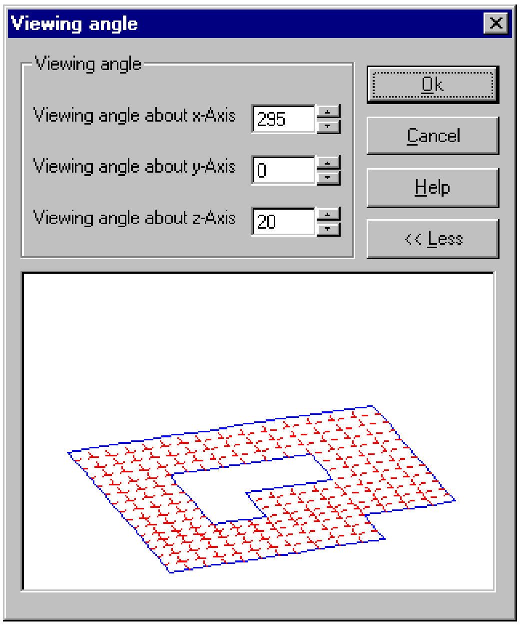


Figure E-57 "Viewing angle" Dialog box

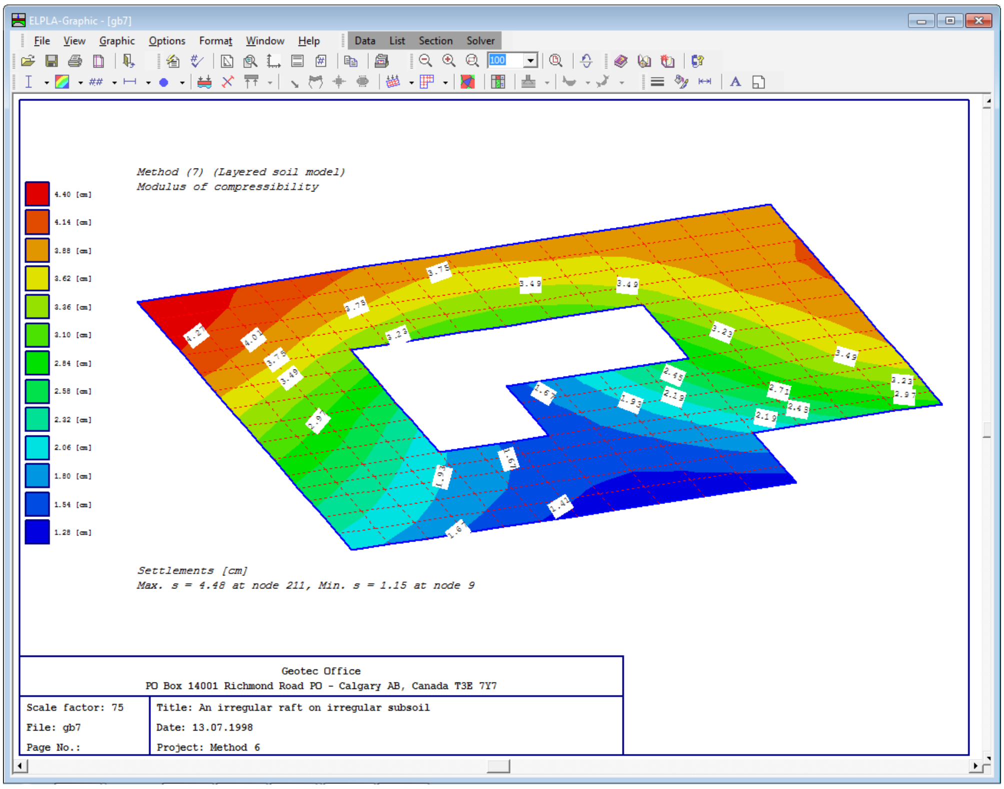


Figure E-58 Contour lines of settlements with viewing angles 295, 0, and 20 about the x-, y-, and z-axes, respectively

10 Help Menu

The Help Menu commands are:

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

10.1 Help Menu–"Contents" command

"Contents" command displays a help file in HTML-Format containing the complete ELPLA User’s Guide, Figure E-59.

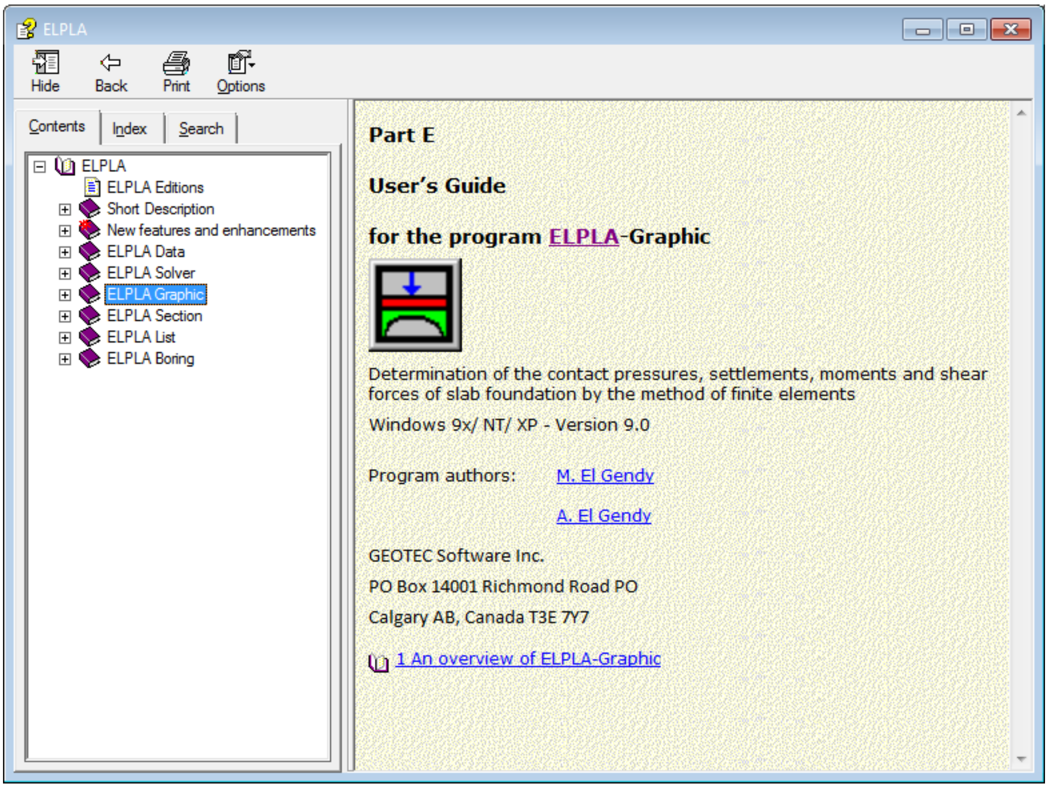


Figure E-59 Menu "Contents"

10.2 Help Menu–"Short description of ELPLA" command

"Short description of ELPLA" command gives a short description of ELPLA package.

10.3 Help Menu–"New in ELPLA" command

"New in ELPLA" command summarizes the new features and enhancements in ELPLA.

10.4 Help Menu–"About ELPLA-Graphic" command

Clicking the command "About ELPLA-Graphic" displays the information form of ELPLA-Graphic as shown in Figure E-60, which gives information about ELPLA-Graphic and the calculation method of the loaded project.

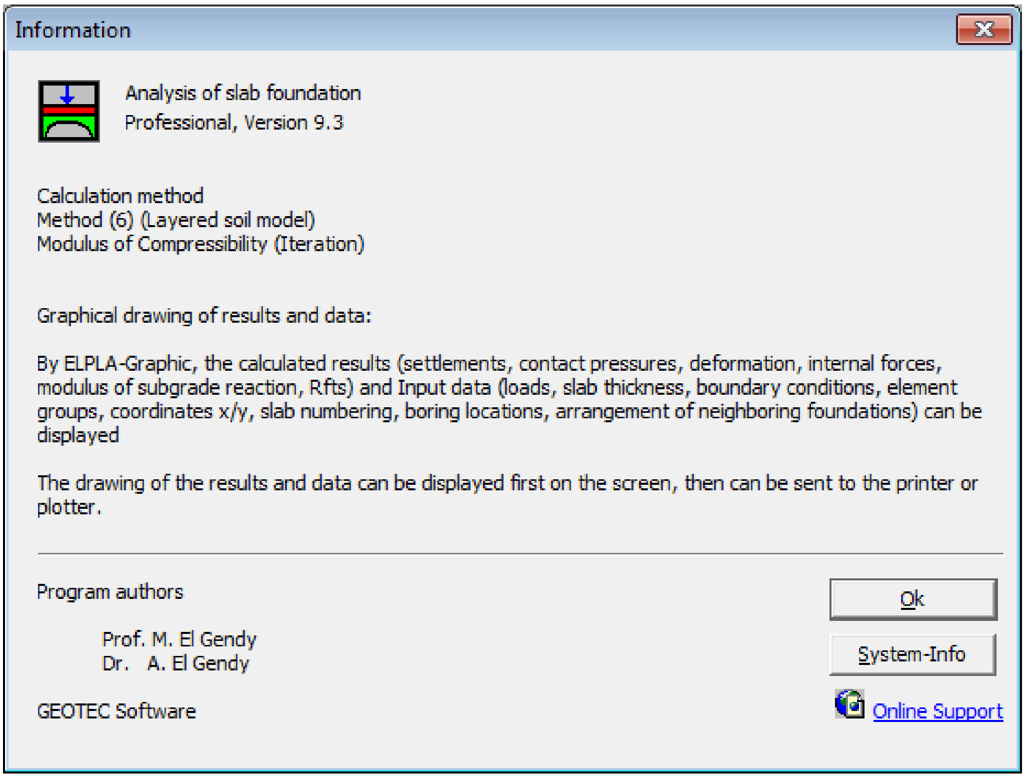


Figure E-60 Information form of program ELPLA-Graphic

11 Tips and Tricks

11.1 Keyboard

The user can obtain all menu titles and commands also through Shortcut keys. The action of the Shortcut keys is listed in Table E-4 to Table E-11:

Table E-4 Shortcut keys of menu head

Shortcut keys	Action
[Alt+f]	Calling menu head "File"
[Alt+v]	"View"
[Alt+g]	"Graphic"
[Alt+o]	"Options"
[Alt+t]	"Format"
[Alt+w]	"Window"
[Alt+h]	"Help"

Table E-5 Shortcut keys of File-Command

Shortcut keys	Action
[Ctrl +o] or [Alt+f] then [o]	Calling command "Open"
[Alt+f] then [w]	"Make WMF-File"
[Alt+f] then [s]	"Send to ELPLA-Section"
[Ctrl +p] or [Alt+f] then [p]	"Print"
[Alt+f] then [u]	"Page setup"
[Alt+f] then [1]	Calling the first project from the last four loaded projects
[Alt+f] then [2]	Calling the second project from the last four loaded projects
[Alt+f] then [3]	Calling the third project from the last four loaded projects
[Alt+f] then [4]	Calling the fourth project from the last four loaded projects
[Ctrl+q] or [Alt+f] then [x]	Calling command "Exit"

Table E-6 Shortcut keys of View-Command

Shortcut keys	Action
[Alt+v] then [b]	Calling command "Status bar"
[Alt+v] then [t]	"Tool bars"
[Alt+v] then [t], then [f]	"Tool bars-File"
[Alt+v] then [t], then [g]	"Tool bars-Graphic"
[Alt+v] then [t], then [o]	"Tool bars-Options"
[Alt+v] then [t], then [t]	"Tool bars-Format"
[Alt+v] then [t], then [w]	"Tool bars-Window"
[Alt+v] then [t], then [h]	"Tool bars-Help"
[Alt+v] then [t], then [r]	"Reset Toolbar"

Table E-7 Shortcut keys of Graphic-Command

Shortcut keys	Action	
[Alt+g] then [r]	Calling command	"Results in isometric view"
[Alt+g] then [e]		"Results as contour lines"
[Alt+g] then [v]		"Result values in the plan"
[Alt+g] then [d]		"Distribution curves of results in the plan"
[Alt+g] then [f]		"Deformation"
[Alt+g] then [p]		"Principal moments as streaks"
[Alt+g] then [s]		"Support reactions as arrows"
[Alt+g] then [v]		"Soil deformation as vectors"
[Alt+g] then [m]		"Soil deformation as deformed mesh"
[Alt+g] then [s]		"Principal soil stresses as streaks"
[Alt+g] then [p]		"Principal soil strains as streaks"
[Alt+g] then [a]		"Data in isometric view"
[Alt+g] then [n]		"Data in the plan"
[Alt+g] then [b]		"Boring locations"
[Alt+g] then [o]		"Boring logs/ Limit depth"
[Alt+g] then [g]		"Beam results"

Table E-8 Shortcut keys of Options-Command

Shortcut keys	Action	
[Alt+o] then [l]	Calling command	"Plot parameters"
[Alt+o] then [d]		"Display values"
[Alt+o] then [c]		"Scale"
[Alt+o] then [s]		"Set range"
[Alt+o] then [a]		"Axes"
[Alt+o] then [t]		"Title"
[Alt+o] then [p]		"Page No."
[Alt+o] then [c]		"Copy"
[Alt+o] then [v]		"View grouping"

Table E-9 Shortcut keys of Format-Command

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

Table E-10 Shortcut keys of Window-Command

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

Table E-11 Shortcut keys of Help-Command

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

11.2 Mouse

By double-clicking the left mouse Button on a specified screen position the user can obtain almost menus of the program.

- By double-clicking on *legend*, *firm header*, *title* or *project identification* the corresponding menu appears
- By double-clicking on *scale* in the identification box "Scale"-Menu appears
- By double-clicking on *file name* in the identification box "Open"-Menu appears
- By double-clicking on *page No.* in the identification box "Page No."-Menu appears
- By clicking the right mouse Button at any position on the screen the user can also obtain the Popup-Options-Menu, Figure E-61
- By double-clicking on a specified node on the FE-Net the corresponding node information appears, Figure E-62

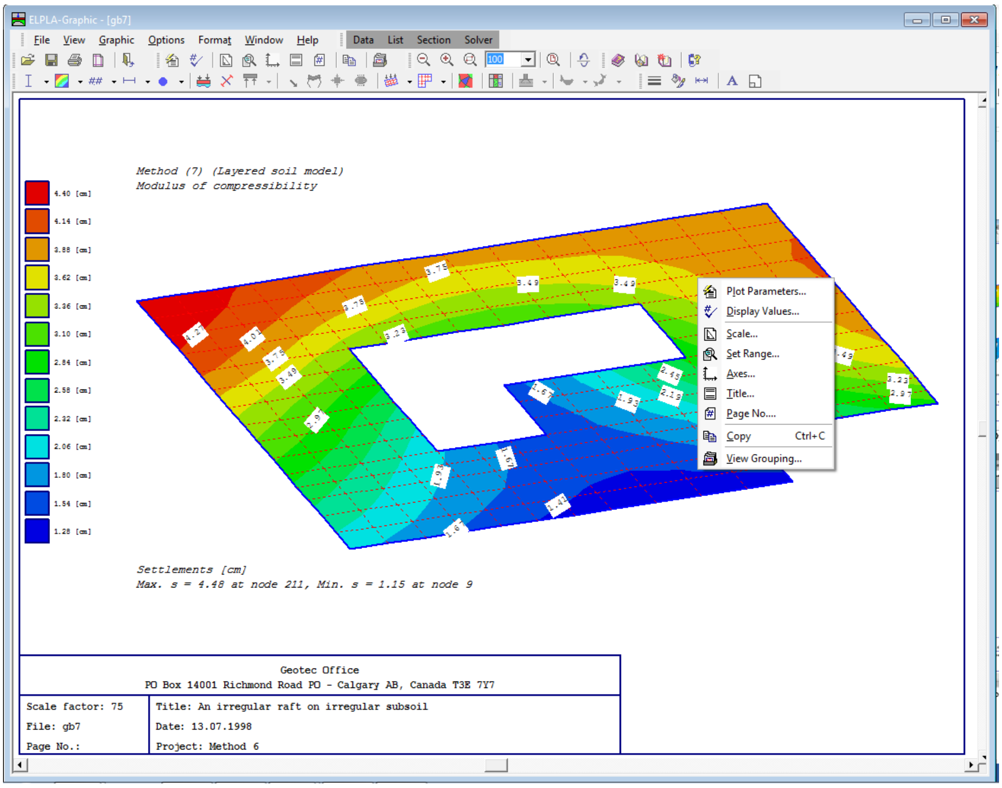


Figure E-61 Menu "Popup-Options"

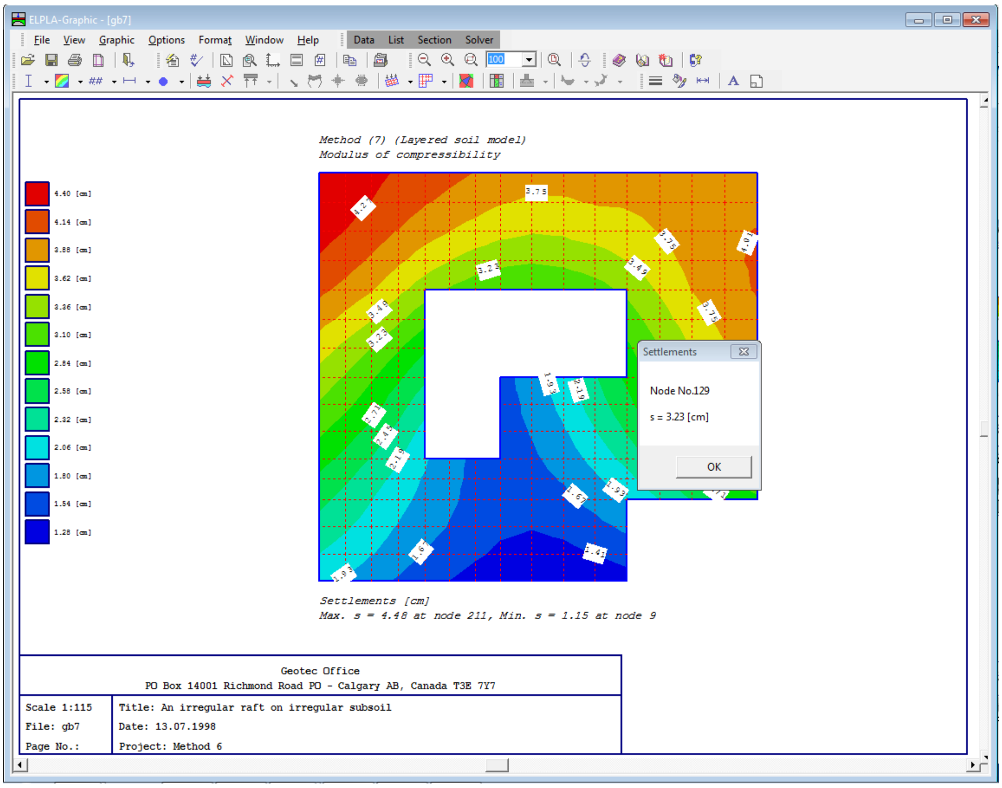


Figure E-62 Node information

12 Samples for graphical drawings using ELPLA-Graphic

ELPLA-Graphic gives the ability to present more than 100 different graphical drawings for data and results. In this paragraph some graphical drawings for data and results of example problems "gb7" (an irregular raft on irregular subsoil) and "rib" (a ribbed raft) are presented.

The pages P1 to P10 show some graphical presentations of data and results for example problem "gb7", while pages P11 and P12 show the graphical presentation of girders for example problem "rib".

12.1 Graphical drawings of data

		Page	
Data in isometric view	Loading		P1
Data in the plan	Geometry of slab with loading		P2
	Location of borings		P3

12.2 Graphical drawings of results

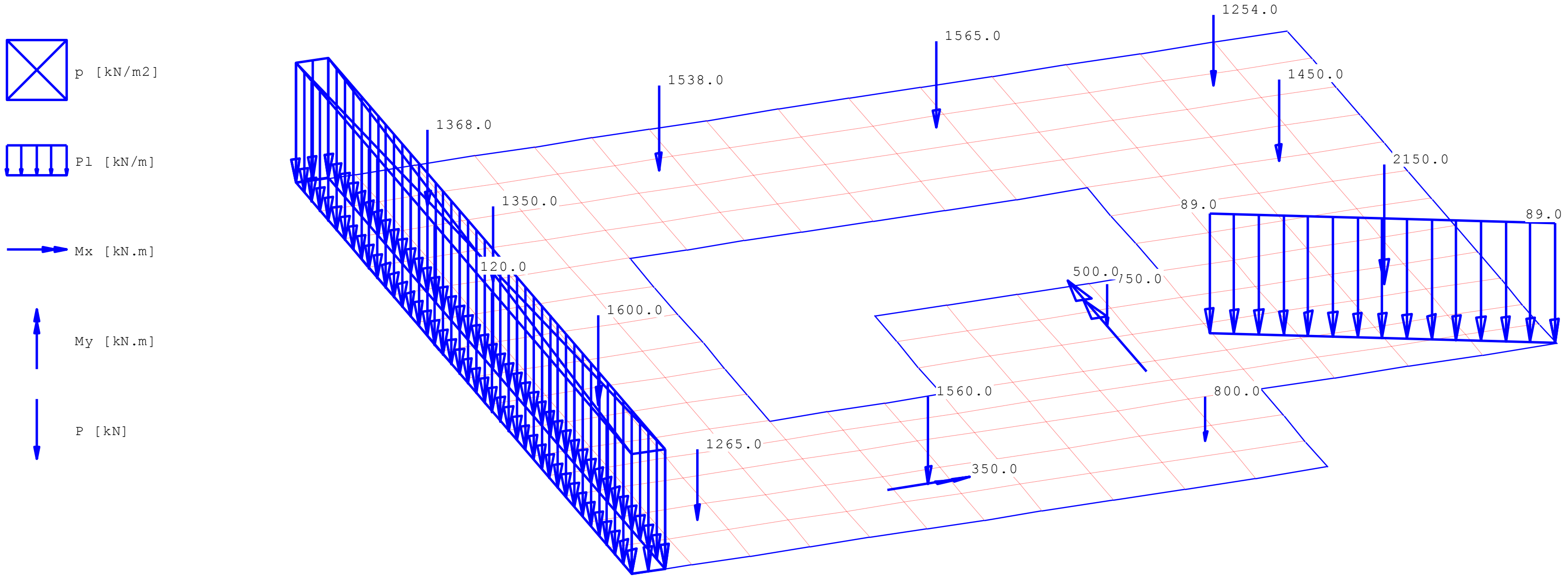
Results in isometric view	Contact pressures q	P4
Deformation	Deformation w	P5
Results as contour lines	Settlements s	P6
Result values in the plan	Bottom Rafts y-Direction A_{sy2}	P7
Distribution curves of results in the plan	Moments m_y	P8
Results as circular diagrams	Moments m_x	P9
Principal moments as streaks	Principal moments	P10
Girders presentation in the plan	Beam-Bending moments M_b	P11
Girders presentation in isometric view	Beam-Bending moments M_b	P12

13 Index

W

Z

Method (7) (Layered soil model)
Modulus of compressibility

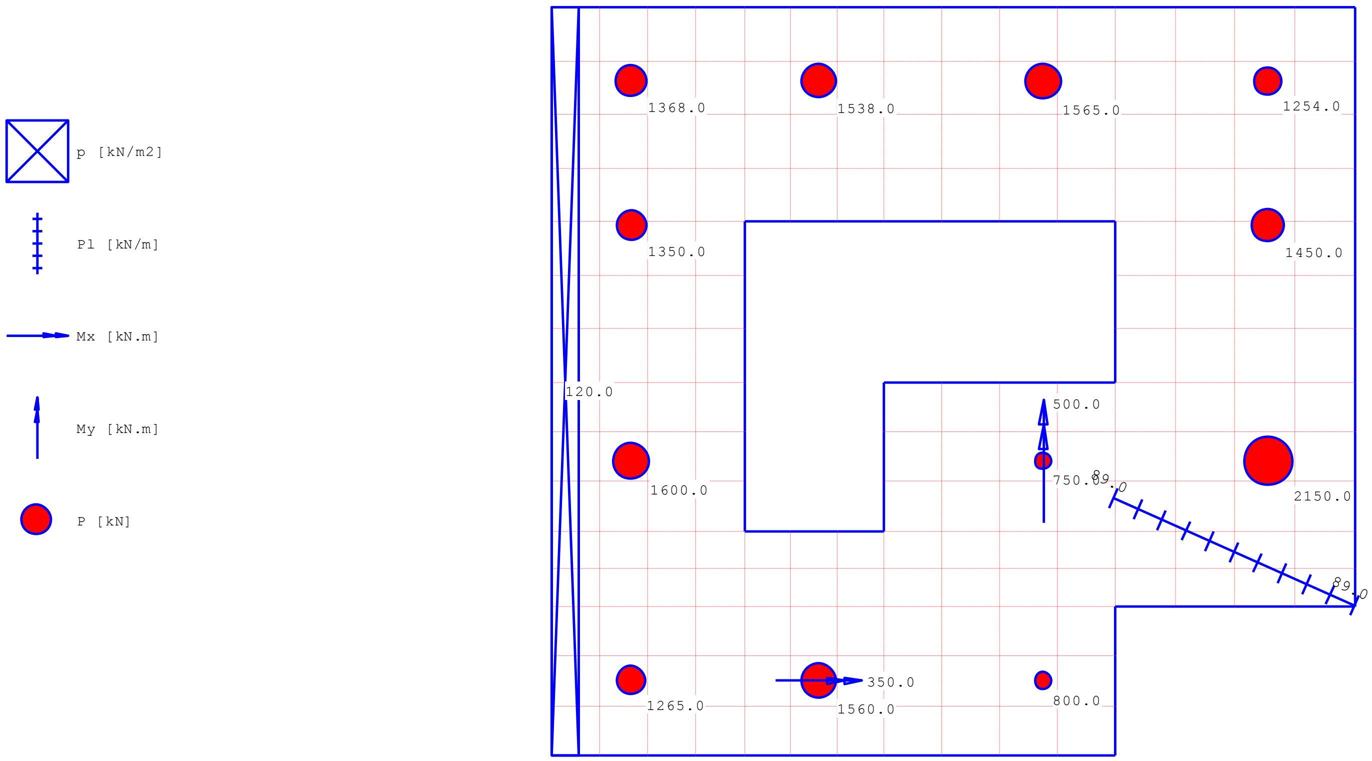


Loads

Geotec Office
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale factor: 85	**Title: An irregular raft on irregular subsoil**
File: gb7	**Date: 13.07.1998**
Page No.:	**Project: Method 6**

Method (7) (Layered soil model)
Modulus of compressibility

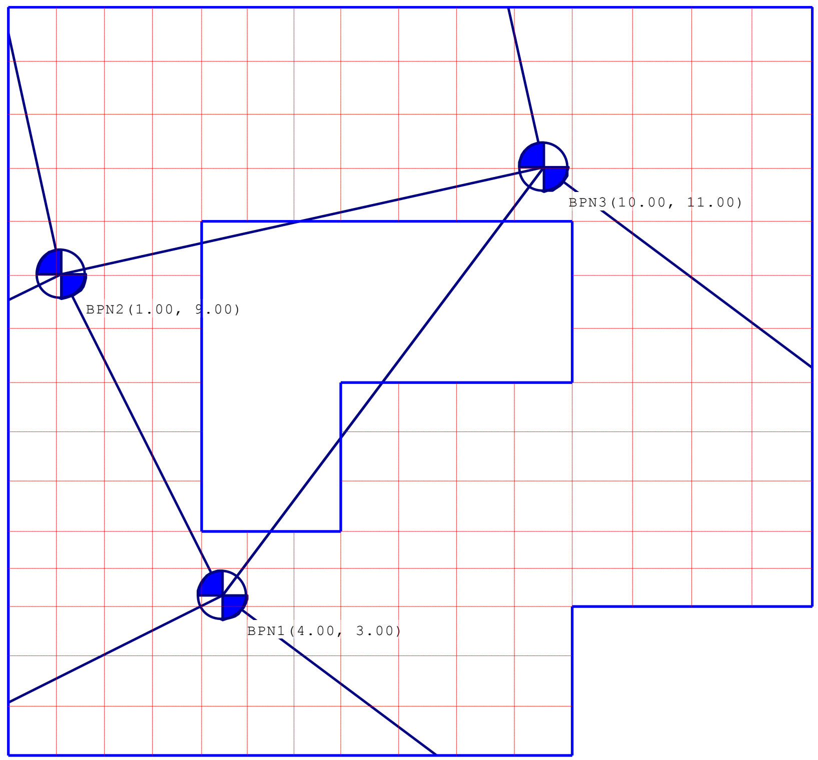


Loads

Geotec Office
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:115	**Title: An irregular raft on irregular subsoil**
File: gb7	**Date: 13.07.1998**
Page No.:	**Project: Method 6**

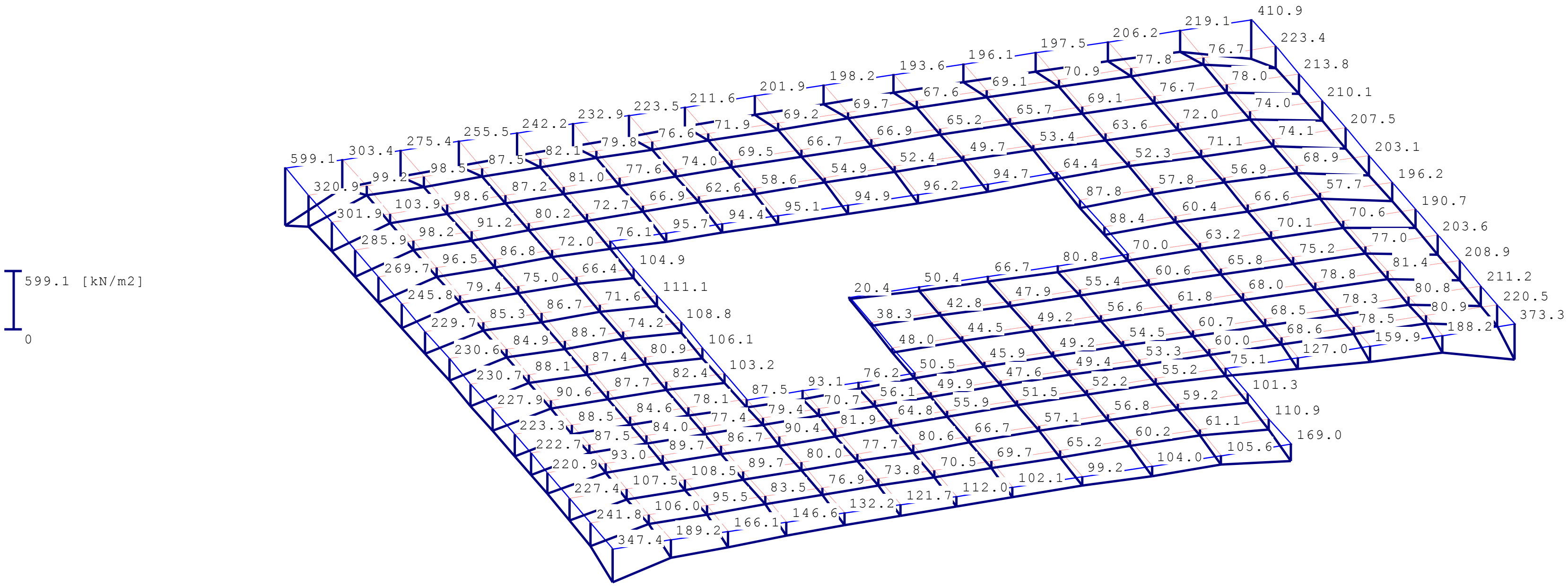
Method (7) (Layered soil model)
Modulus of compressibility



Boring Locations
No. of boring logs = 3

Geotec Office **PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7**	
Scale 1:115	**Title: An irregular raft on irregular subsoil**
File: gb7	**Date: 13.07.1998**
Page No.:	**Project: Method 6**

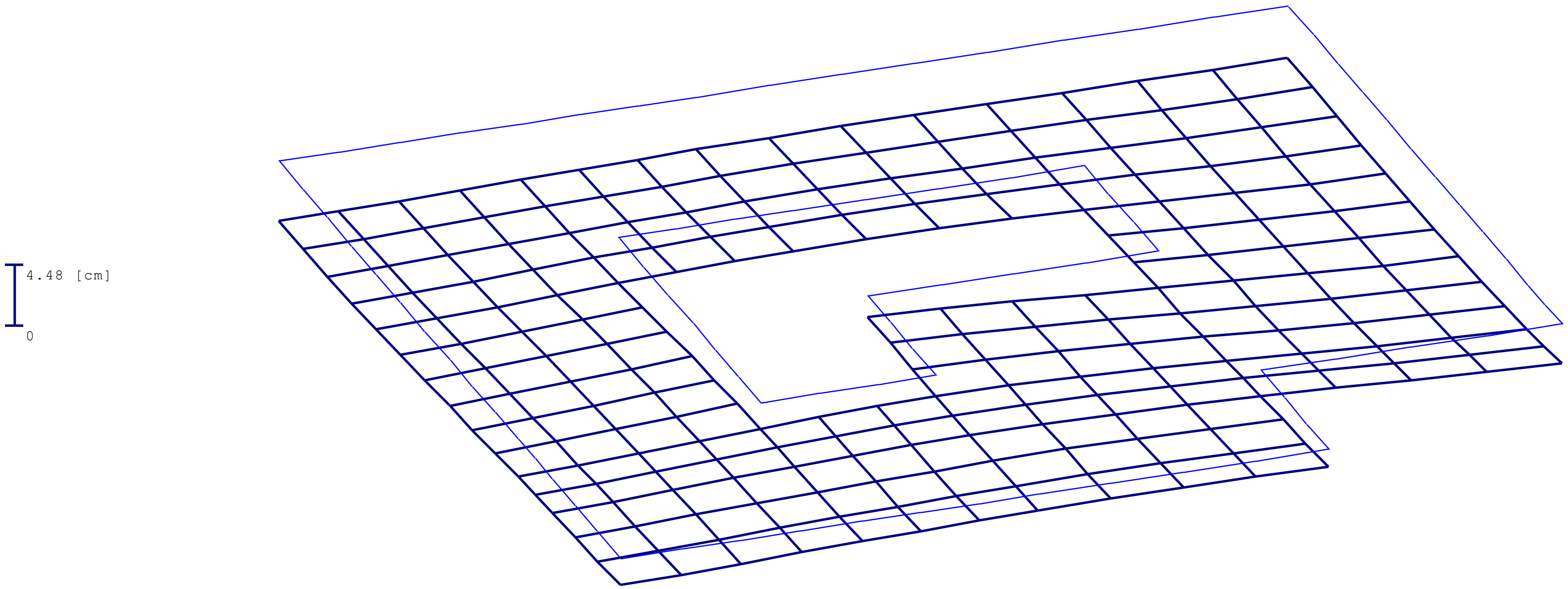
Method (7) (Layered soil model)
Modulus of compressibility



Contact pressure q [kN/m2]
Max. q = 599.1 at node 211, Min. q = 20.4 at node 118

Geotec Office PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7	
Scale factor: 85	**Title: An irregular raft on irregular subsoil**
File: gb7	**Date: 13.07.1998**
Page No.:	**Project: Method 6**

Method (7) (Layered soil model)
Modulus of compressibility

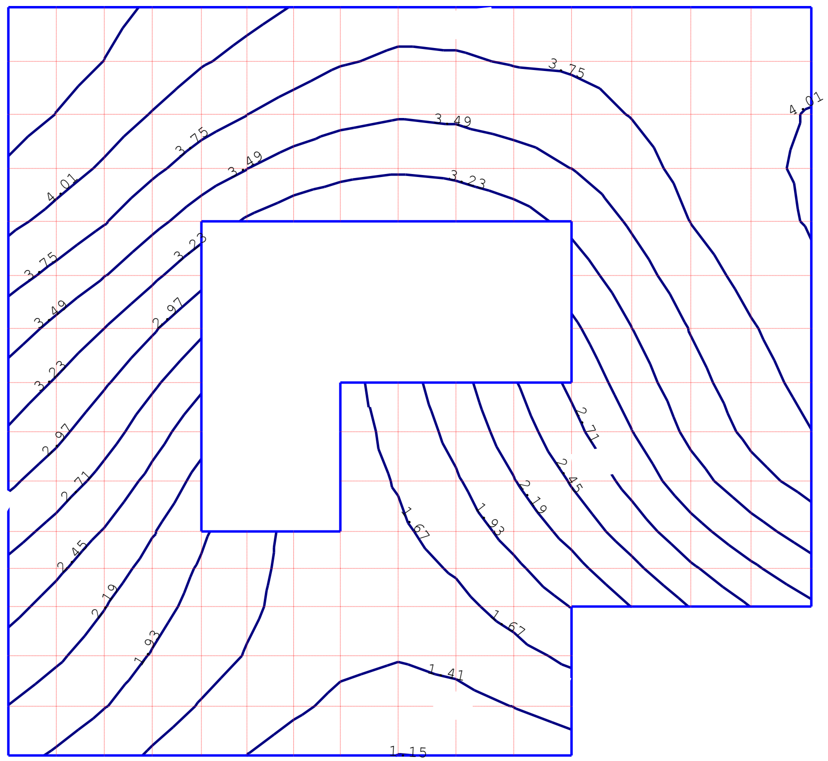


Slab deformation w [cm]
Max. w = 4.48 at node 211, Min. w = 1.15 at node 9

Geotec Office
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale factor: 85	**Title: An irregular raft on irregular subsoil**
File: gb7	**Date: 13.07.1998**
Page No.:	**Project: Method 6**

Method (7) (Layered soil model)
Modulus of compressibility

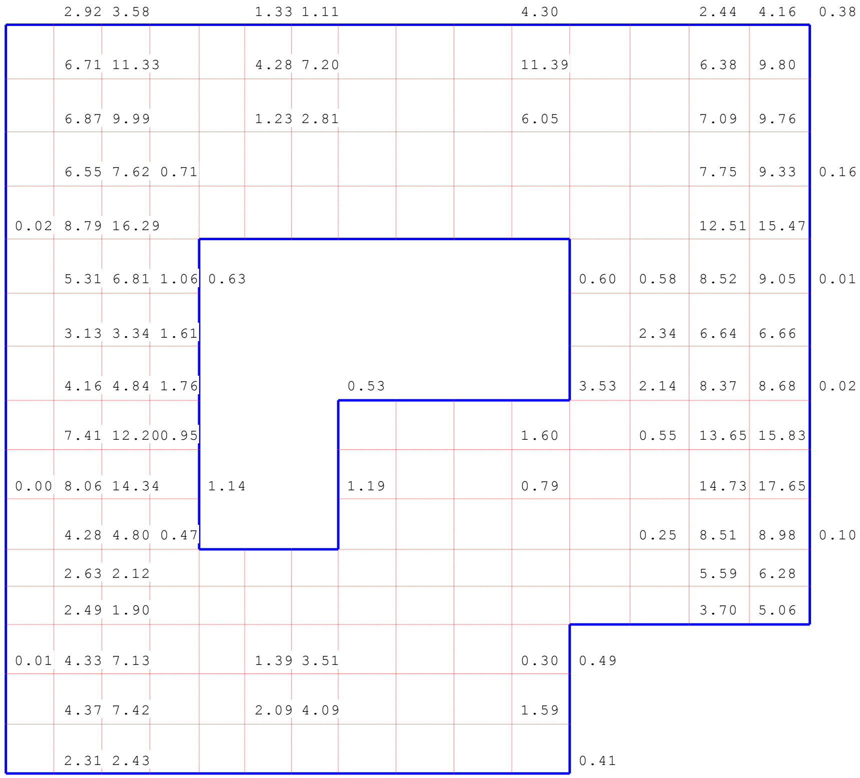


Settlements [cm]
Max. s = 4.48 at node 211, Min. s = 1.15 at node 9

Geotec Office
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:115	**Title: An irregular raft on irregular subsoil**
File: gb7	**Date: 13.07.1998**
Page No.:	**Project: Method 6**

Method (7) (Layered soil model)
Modulus of compressibility



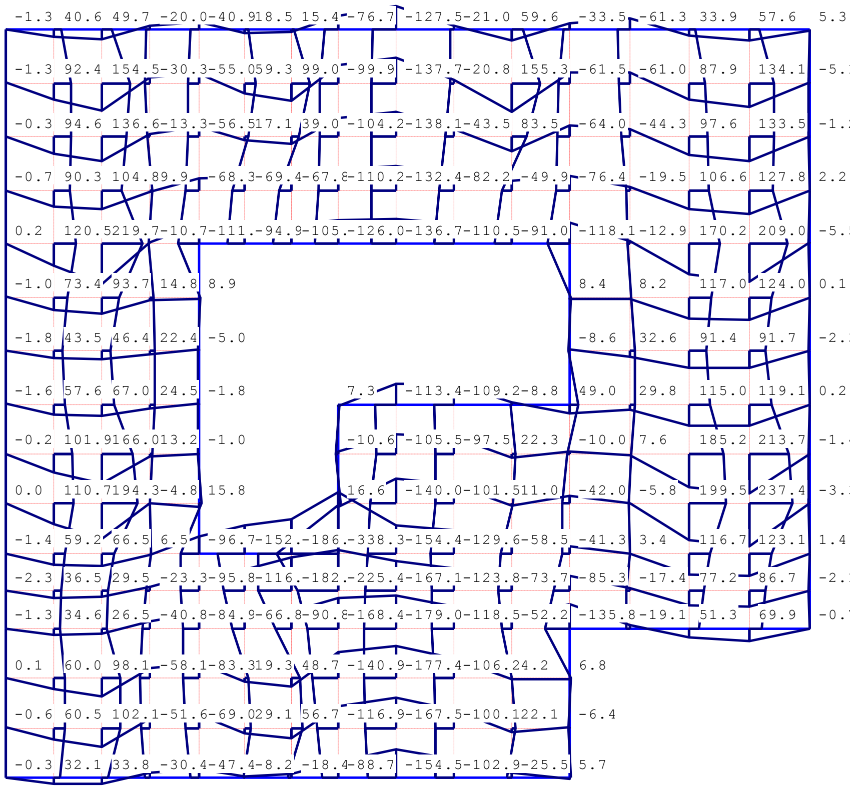
Bottom reinforcement in x-direction As_botx [cm2/m]
Max. As_botx = 17.65 at node 97, Min. As_botx = 0.00 at node 85

Geotec Office
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:115	**Title: An irregular raft on irregular subsoil**
File: gb7	**Date: 13.07.1998**
Page No.:	**Project: Method 6**

Method (7) (Layered soil model)
Modulus of compressibility

0 338.3 [kN.m/m]

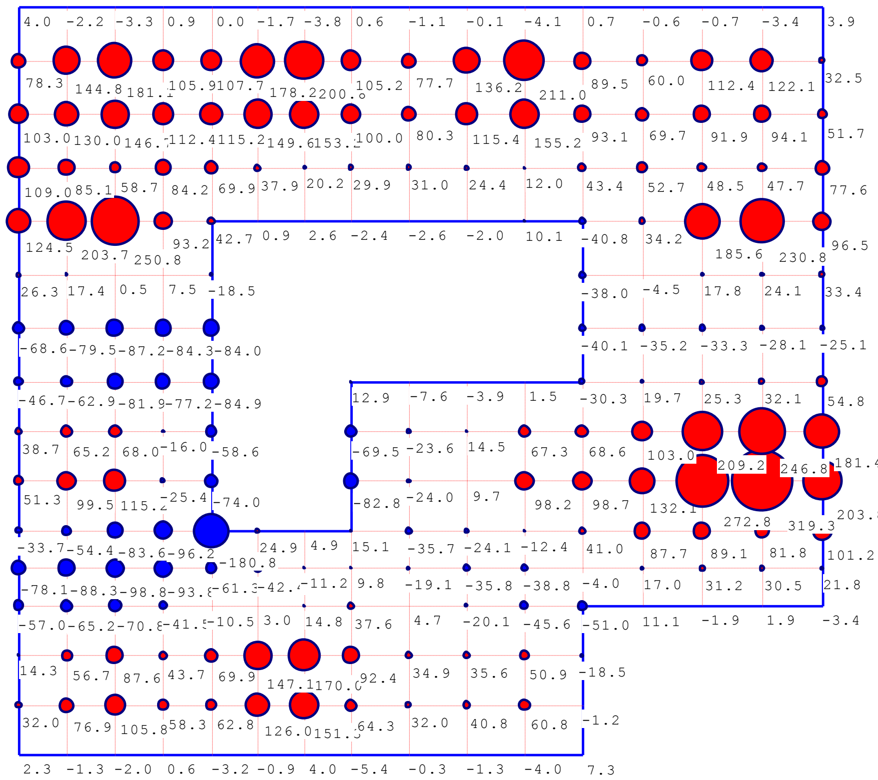


Moments mx [kN.m/m]
Max. mx = 237.4 at node 97, Min. mx = -338.3 at node 76

Geotec Office
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:115	**Title: An irregular raft on irregular subsoil**
File: gb7	**Date: 13.07.1998**
Page No.:	**Project: Method 6**

Method (7) (Layered soil model)
Modulus of compressibility

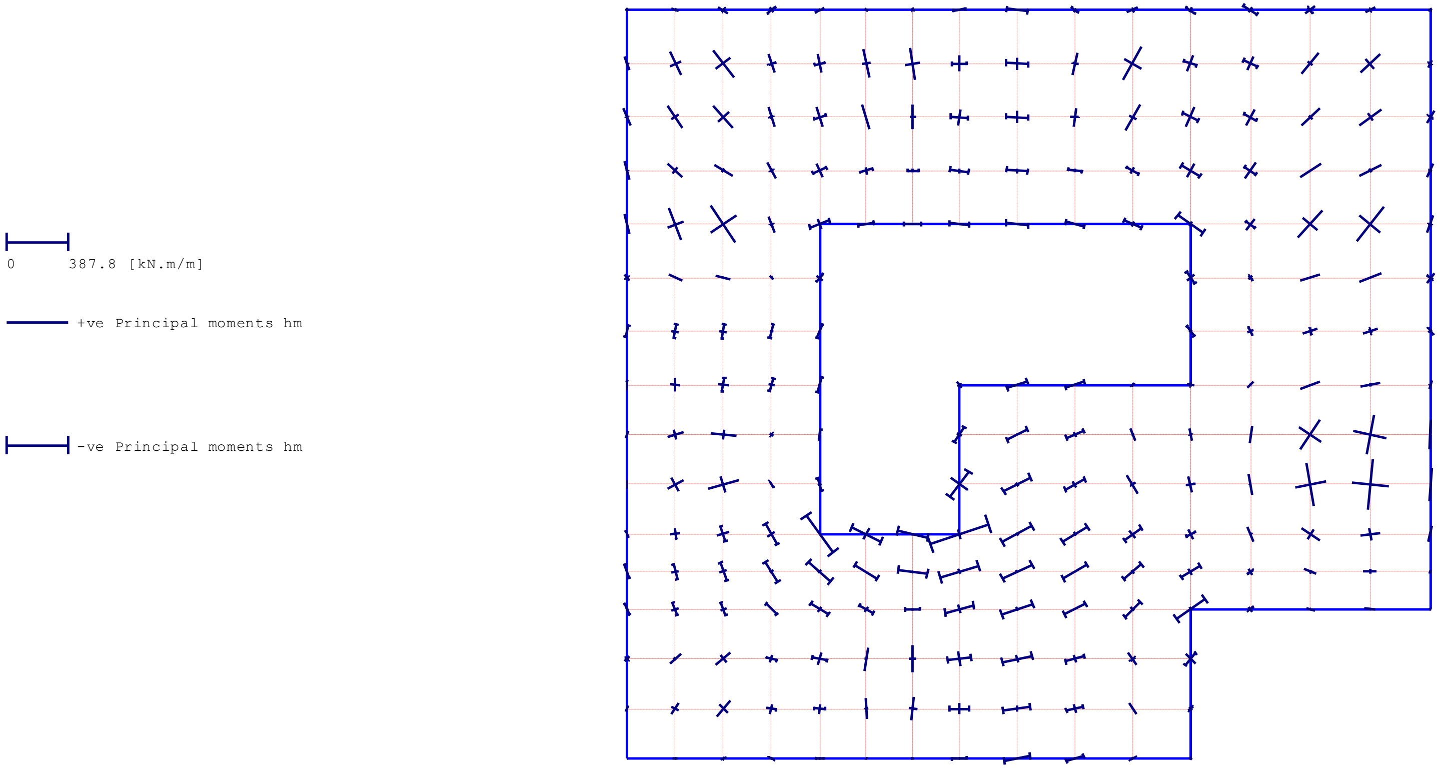


Moments my [kN.m/m]
Max. my = 319.3 at node 97, Min. my = -180.8 at node 73

Geotec Office
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:115	Title: An irregular raft on irregular subsoil
File: gb7	Date: 13.07.1998
Page No.:	Project: Method 6

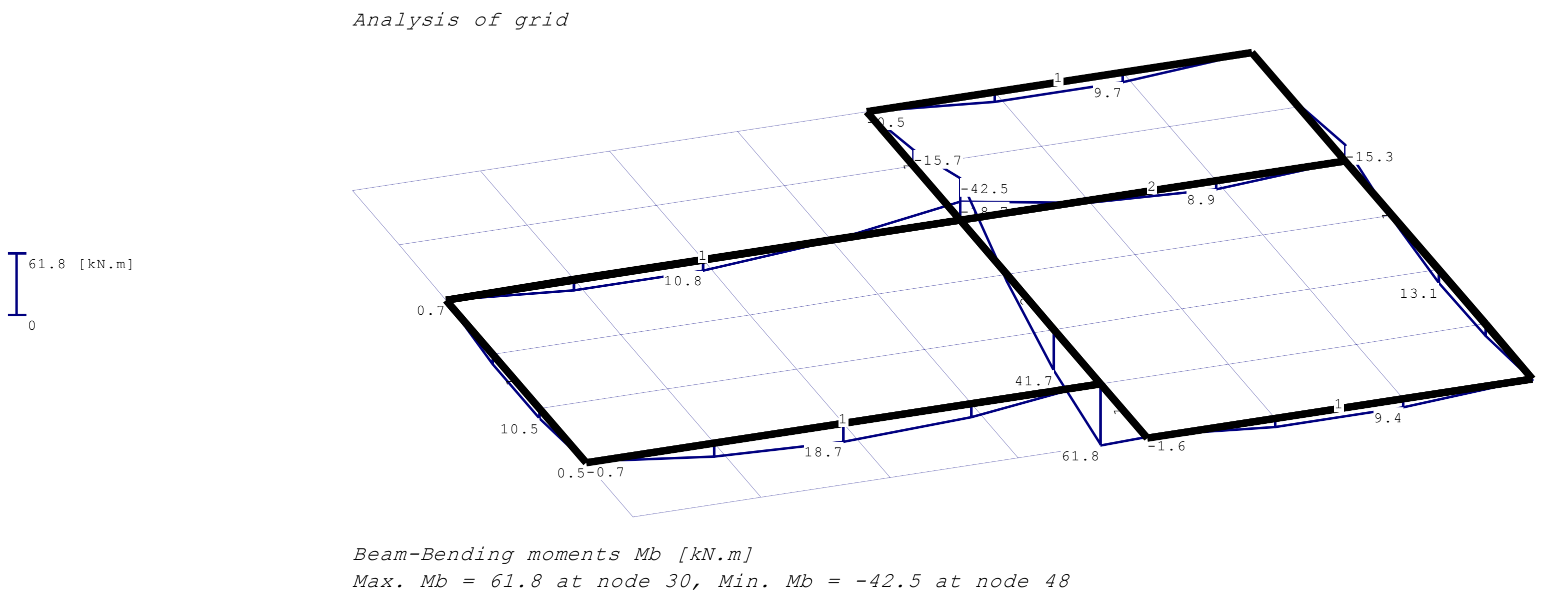
Method (7) (Layered soil model)
Modulus of compressibility



Principal moments hm [kN.m/m]
Max. hm = 320.1 at node 97, Min. hm = -387.8 at node 76

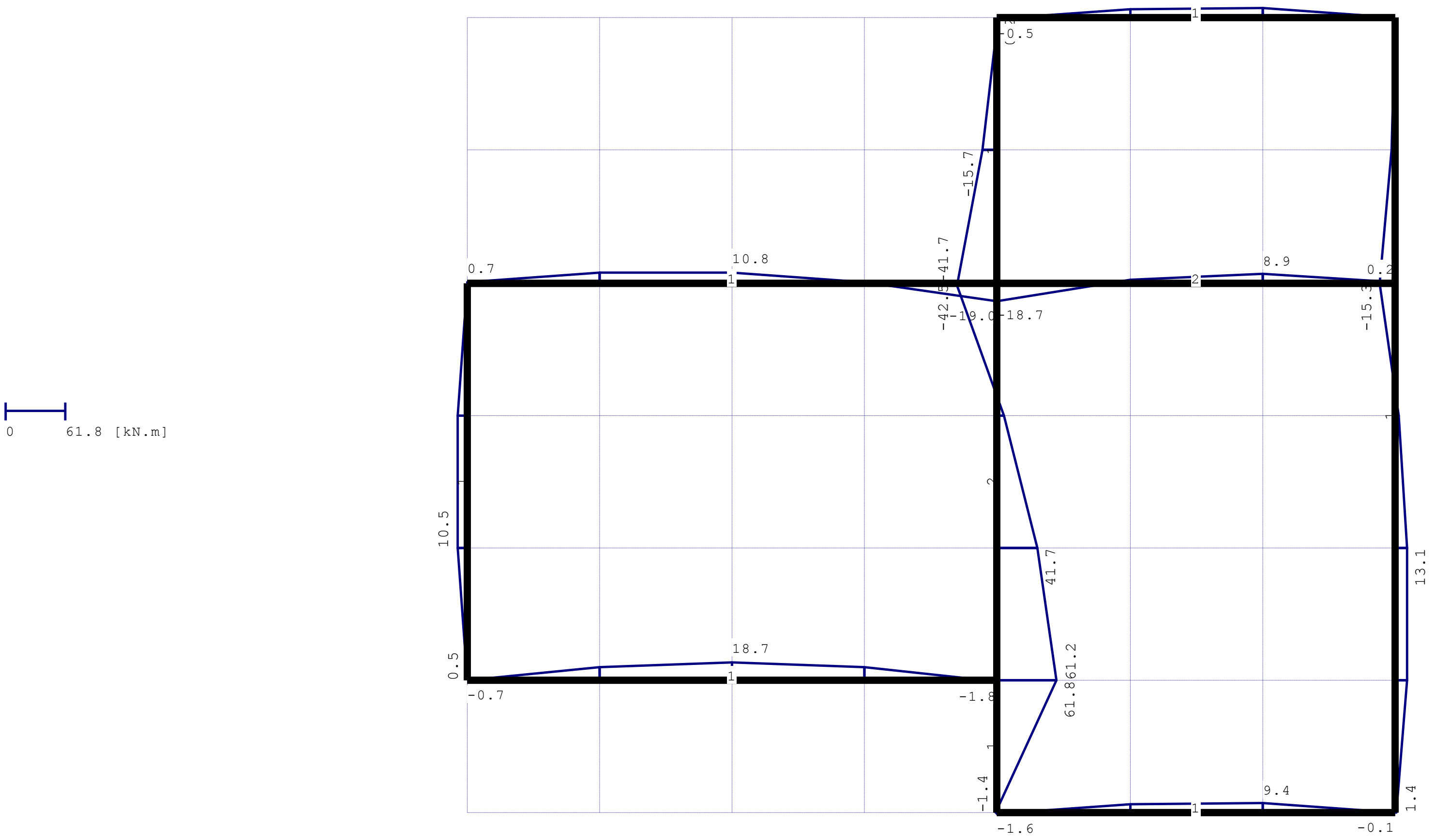
Geotec Office
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:115	**Title: An irregular raft on irregular subsoil**
File: gb7	**Date: 13.07.1998**
Page No.:	**Project: Method 6**



Geotec Office PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7	
Scale factor: 45	**Title: Analysis of a grid**
File: Grid	**Date: 14-06-2003**
Page No.:	**Project: Grid**

Analysis of grid



Beam-Bending moments Mb [kN.m]

Max. Mb = 61.8 at node 30, Min. Mb = -42.5 at node 48

Geotec Office

PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:45	**Title: Analysis of a grid**
File: Grid	**Date: 14-06-2003**
Page No.:	**Project: Grid**

Part F

User’s Guide
for the program ELPLA-Section

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

Version 9.2

Program authors: M. El Gendy
A. El Gendy

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

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

Contents Page

1 An overview of ELPLA-Section

ELPLA-Section is used to display graphically the calculated results (settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement) and input project data (additional settlements) at specified sections. The drawing of the results and data can be displayed first on the screen, and then can be sent to the printer or plotter. Also, all drawing sections can be exported to MS Excel.

The calculated results and input project data can be presented graphically as follows:

1 Section in x-direction
2 Max./ Min. values in x-direction
3 Overlapping in x-direction
4 Section in y-direction
5 Max./ Min. values in y-direction
6 Overlapping in y-direction
7 Arbitrary section

The drawings, if desired, can be saved as WMF-Format files, in which they can be exported to other Windows applications such as Word and AutoCAD to prepare reports, slide presentations or add further information.

2 Description of ELPLA-Section

ELPLA-Section is a 32-bit analysis and design software product that operates under Microsoft Windows 9x/ NT/ ME/ XP. The common "what you see is what you get" of Windows applications makes it easy to learn how to use ELPLA-Section, especially if you are already familiar with the Windows environment.

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

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

In order to use ELPLA-Section, first the user must enter the project data by ELPLA-Data, and then must analyze the problem by ELPLA-Solver. Table F-2 gives a list of files, which are read or created by ELPLA-Section. The files can be classified in four groups.

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

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

Table F-2 Names of file groups

Group	Saved from the program
A Main data files	ELPLA-Data
B Section files (*.SEC)	ELPLA-Section
C Project data files	ELPLA-Data
D Final result files	ELPLA-Solver

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

Table F-3 Names and contents of files

A Main data files

Filename	Contents
FIRMA	Firm header
STEU	Default directory for files that are saved by ELPLA
NOFORMAT	Number formats
UNITS	System of units

B Section files

Filename	Contents
PLOTPAR.SEC	Plot parameters
LEGENDE.SEC	Legend data
ORDINATE. SEC	Max. ordinate data
LINEFORM.SEC	Line formats
FONT.SEC	Font data

C Project data files

Filename	Contents
*. AUF	3 line texts to identify the project
*. BAU	Soil properties
* .PO1	System data (calculation method)
*. PL6	FE-Net data
*. P33	Data of supports/ boundary conditions
*. PV1	Data of additional soil settlements

D Final result files

Filename	Contents
*. PT3	Displacements due to temperature change (s_t)
*. PP3	Settlements due to neighboring foundations (s_e)
*. QUN	Ultimate bearing capacities at nodes (q_{ul})
*. PW1	Modulus of subgrade reaction (k_s)
*. PH1	Settlements (s)
*. PH2	Contact pressures (q)
*. PH3	Moments (m_x)
*. PH4	Moments (m_y)
*. PH5	Moments (m_{xy})
*. PH6	Shear forces (Q_x)
*. PH7	Shear forces (Q_y)
*. PH8	Settlements due to reloading (s_w)
*. PH9	Overburden pressures (Q_u)
*. H10	Support reactions (V)
*. H11	Support reactions (M_y)
*. H12	Support reactions (M_x)
*. H13	Reinforcement of the slab (A_{sx1})
*. H14	Reinforcement of the slab (A_{sx2})
*. H15	Reinforcement of the slab (A_{sy1})
*. H16	Reinforcement of the slab (A_{sy2})

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

Next paragraphs describe the purpose and function of each ELPLA-Section command.

3 Starting ELPLA-Section

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

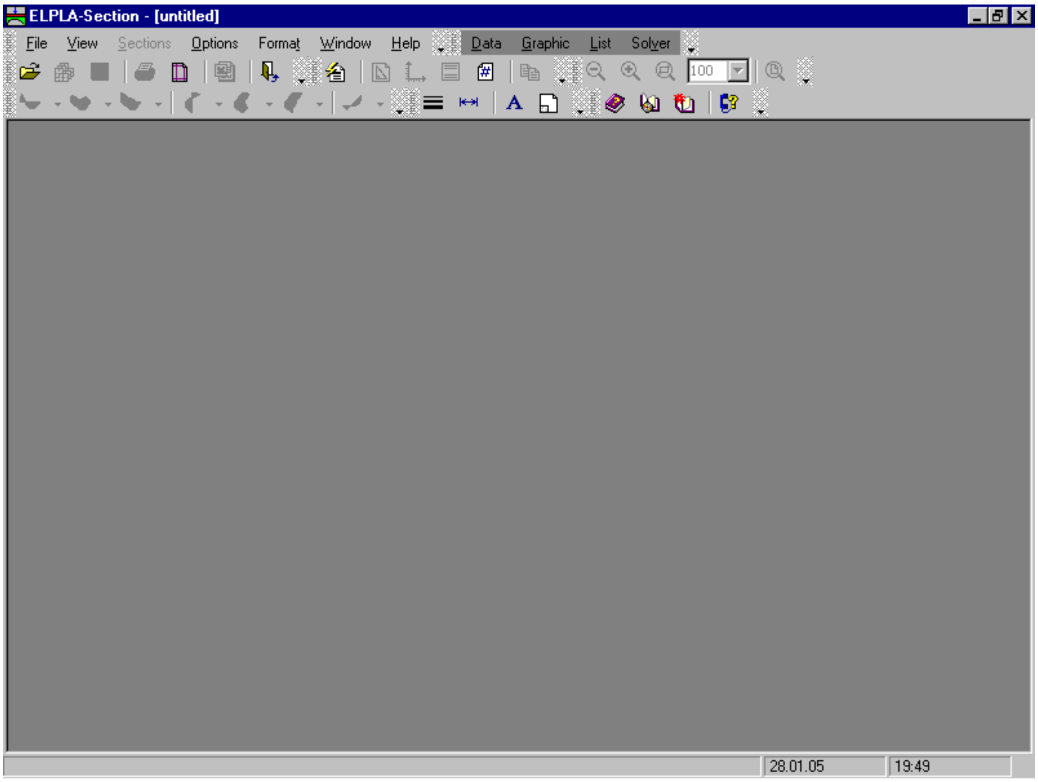


Figure F-1 Introduction screen of ELPLA-Section

The menu head of Figure F-1 contains the following seven commands:

- File
- View
- Sections
- Options
- Format
- Window
- Help

After clicking one of the seven menu commands, another sub-commands or options become available. The seven menu commands and their sub-commands are presented and described in the following paragraphs 4 to 11.

4 File Menu

The File Menu commands are:

- Open
- Combination of many projects
- Make WMF-File
- Print
- Page setup
- Send to Excel
- Files 1, 2, 3, 4
- Exit

4.1 File Menu–"Open" command

By "Open" command the current project is closed, if one is loaded, and an existing project is opened. Figure F-2 shows "Open" Dialog box used to open a specified project.

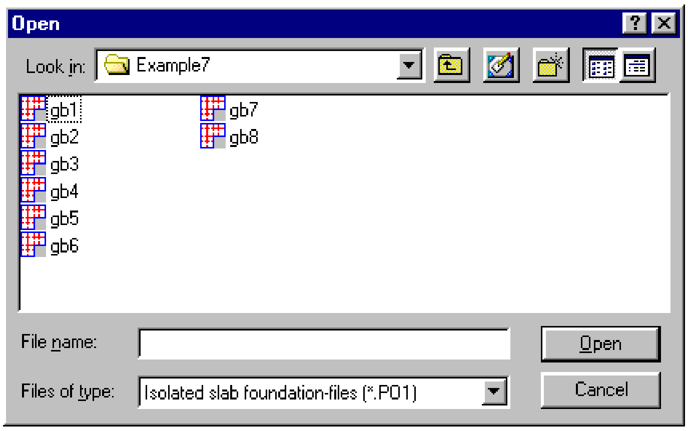


Figure F-2 "Open project" Dialog box

4.2 File Menu–"Combination of many projects" command

The extreme values of calculated results (settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement) of many projects can be drawn.

For determining results as extreme values or drawing results of many projects in a diagram, project names and their comments are required.

When the command is chosen, the input Dialog box shown in Figure F-3 appears. In this input Dialog box the project name and its identification can be defined.

4.3 File Menu–"Make WMF-File" command

By "Make WMF-File" command the drawing can be saved in a format that can be read by other programs. This feature allows you to include your drawing in reports and presentations and to enhance the drawing using other drawing or CAD software packages. The drawing can be exported in the Windows Metafile (WMF) format.

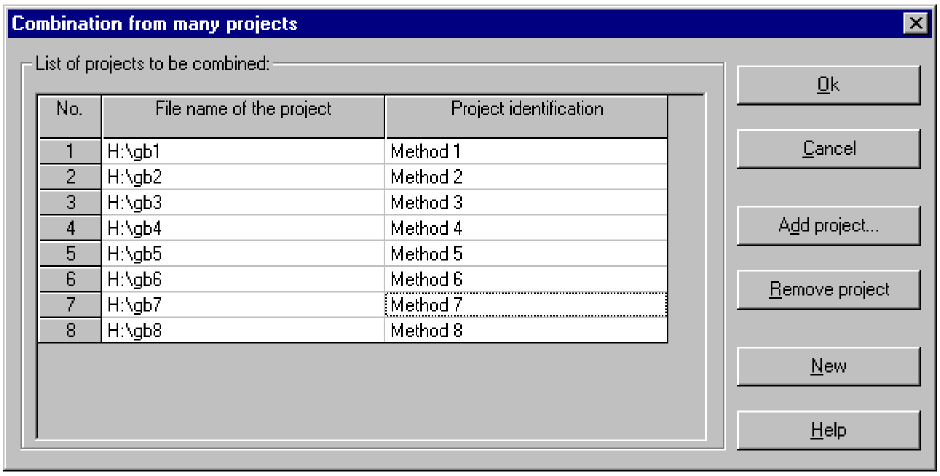


Figure F-3 "Combination of many projects" input Dialog box

4.4 File Menu–"Print" command

By "Print" command data and results can be graphically plotted or printed. Only the objects currently displayed on the drawing are printed. Figure F-4 shows "Print" Dialog box. The printer group box contains controls for selecting the printer and changing its properties. Use "Name" Combo box to select the printer and use "Properties" Button to set printer settings. The number of printing copies can be defined in "Copies" Input box.

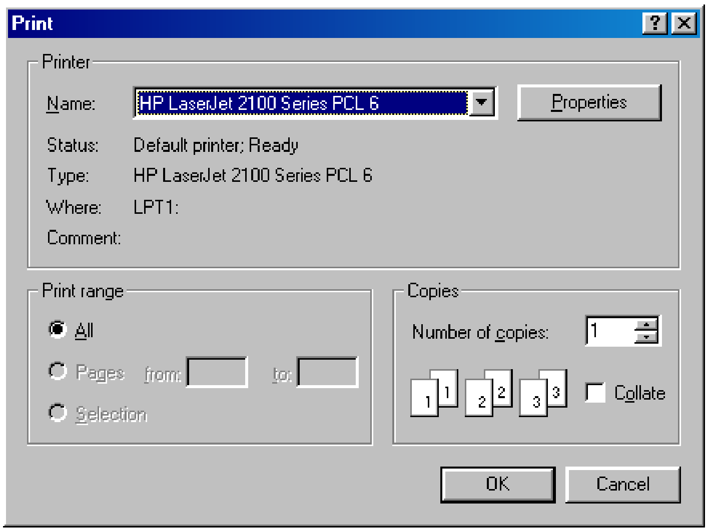


Figure F-4 "Print" Input box

4.5 File Menu–"Page setup" command

By "Page setup" command the standard "Page setup" Dialog box can be displayed with options to specify the printer, page orientation, paper size and paper source, as well as other printing options. Figure F-5 shows "Page setup" Dialog box.

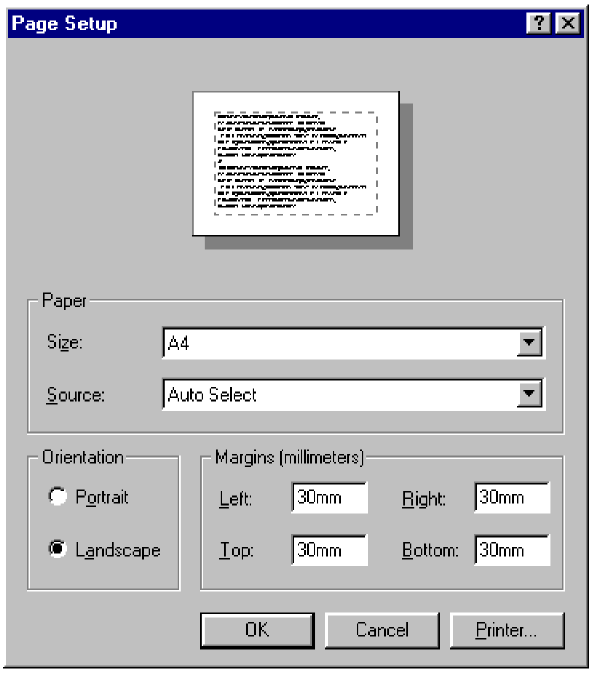


Figure F-5 "Page setup" Dialog box

4.6 File Menu–"Send to Excel" command

By this command, it is possible to export results as diagrams to MS Excel.

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

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

4.8 File Menu–"Exit" command

By "Exit" command the current project is closed and ELPLA-Section is quitted (Figure F-7).

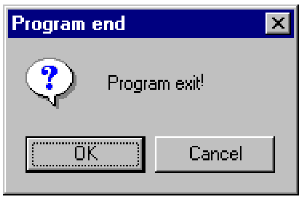


Figure F-7 "Exit" Message box

5 View Menu

The View Menu commands are:

- Status bar
- Tool bars

5.1 View Menu–"Status bar" command

"Status bar" command displays a status bar on the screen down. The status bar displays information about the progress of the current operation.

5.2 View Menu–"Tool bars" command

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

6 Sections Menu

The Section menu is the main menu, which is used to display the specified drawing with the setting given by options, format and window menus.

The Sections Menu commands are:

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

6.1 Sections Menu–"Section in x-direction" command

By "Section in x-direction" command the calculated results (settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement) and input data (additional settlements) can be plotted at a specified section in x-direction.

When this command is chosen, the selection Dialog box shown in Figure F-8 appears. In this Dialog box select one item to draw, then click "OK" Button.

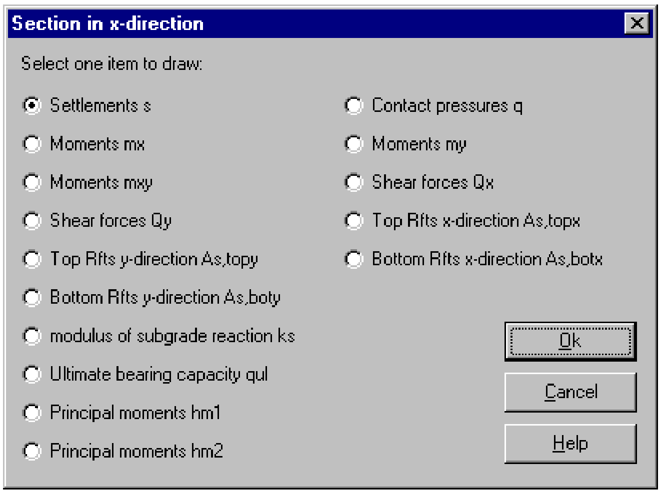


Figure F-8 "Section in x-direction" items

For drawing results at a specified section in x-direction, ranges of the section are required as shown in Figure F-9.

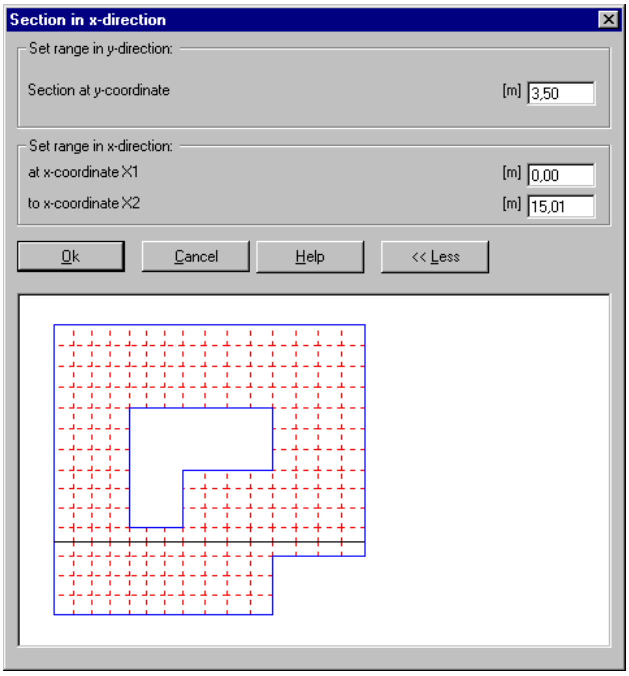


Figure F-9 "Section in x-direction" Dialog box

As an example, Figure F-10 shows a specified section of settlement in x-direction.

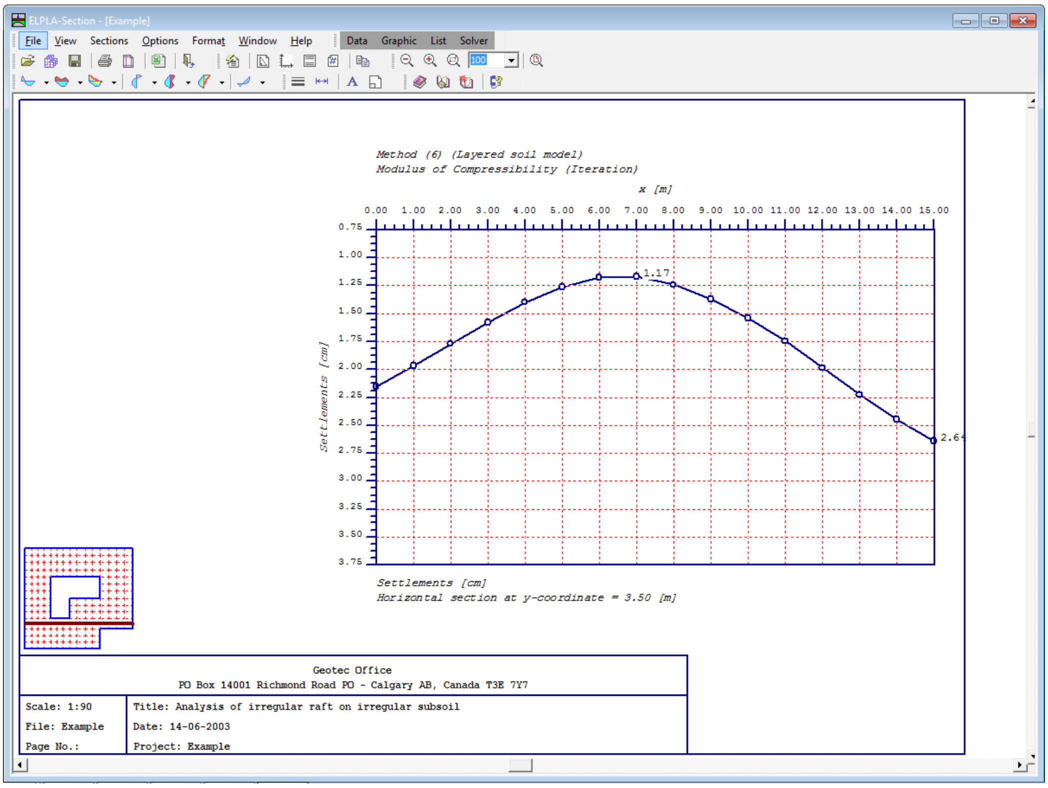


Figure F-10 Section of settlement in x-direction

6.2 Sections Menu–"Max./ Min. values in x-direction" command

By "Max./ Min. values in x-direction" command the extreme values of results (settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement) and input data (additional settlements) can be plotted at a specified range in x-direction. When the command is chosen, the selection Dialog box shown in Figure F-11 appears. In this Dialog box select one item to draw and then click "OK" Button.

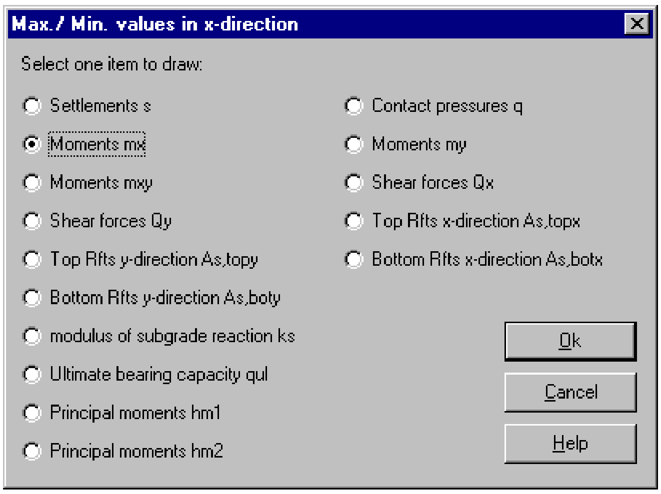


Figure F-11 "Max./ Min. values in x-direction" items

For drawing extreme values of results at a specified range in x-direction, ranges of the section are required as shown in Figure F-12.

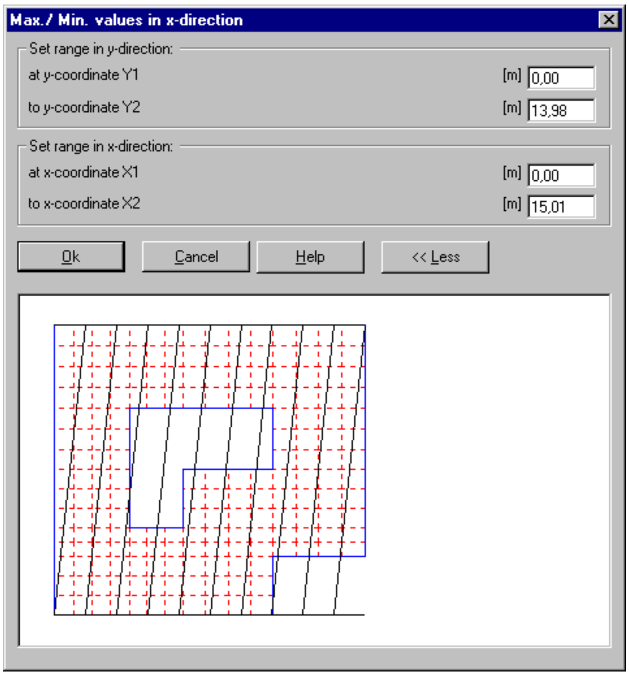


Figure F-12 "Section in x-direction" Dialog box

As an example, Figure F-13 shows extreme values of moments m_x in x-direction for the whole raft.

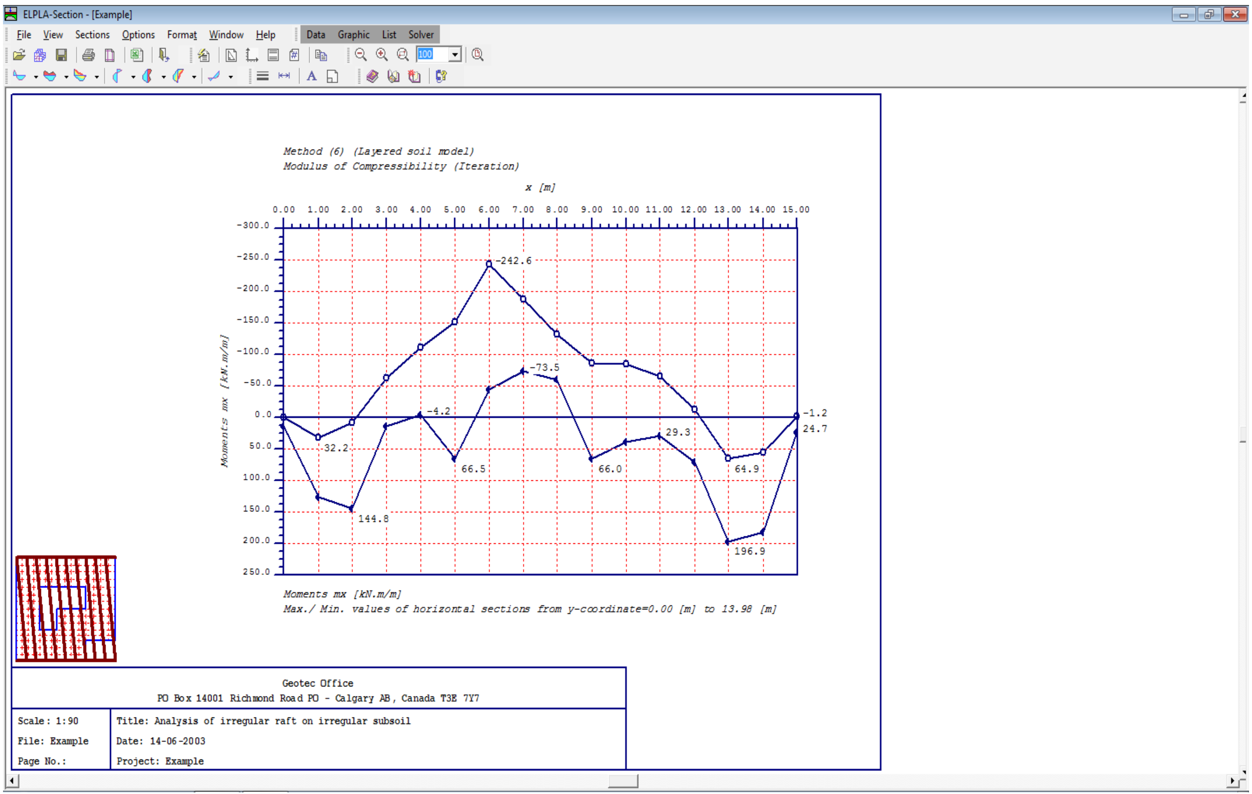


Figure F-13 Extreme values of moments m_x in x-direction for the whole raft

6.3 Sections Menu–"Overlapping in x-direction" command

Values of calculated results (settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement) and input data (additional settlements) can be plotted in many sections for a specified range in x-direction in one graph.

When "Overlapping in x-direction" command is chosen, the selection Dialog box shown in Figure F-14 appears. In this Dialog box select one item to draw, then click "OK" Button.

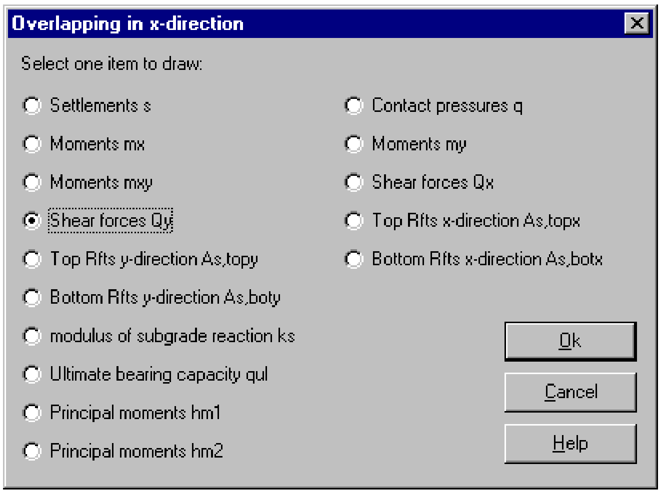


Figure F-14 "Overlapping in x-direction" items

For drawing values of calculated results at a specified range in x-direction in one graph, ranges of the section are required as shown in Figure F-15.

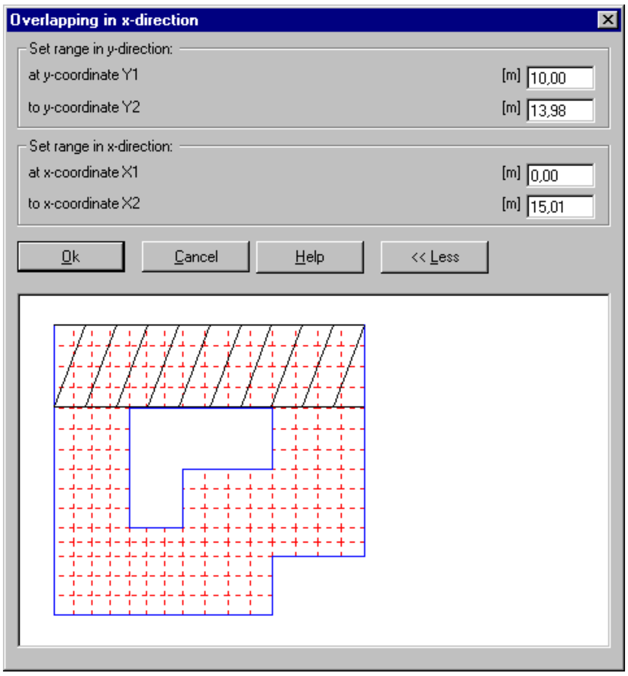


Figure F-15 "Overlapping in x-direction" Dialog box

As an example, Figure F-16 shows values of shear forces Q_y in x-direction in a specified zone of the raft.

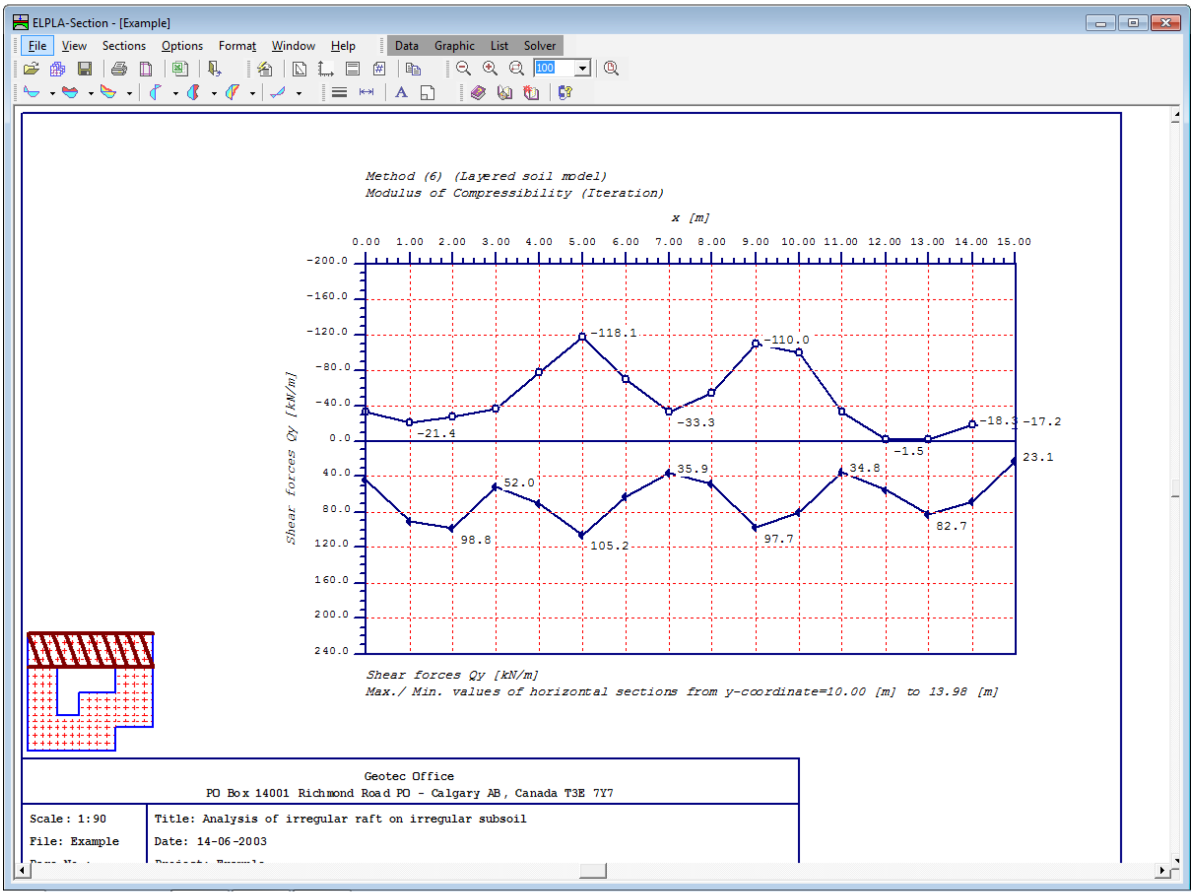


Figure F-16 Sections of shear forces Q_y in x-direction for a specified zone

6.4 Sections Menu–"Section in y-direction" command

By "Section in y-direction" command the calculated results (settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement) and input data (additional settlements) can be plotted at a specified section in y-direction.

When this command is chosen, the selection Dialog box shown in Figure F-17 appears. In this Dialog box select one item to draw, then click "OK" Button.

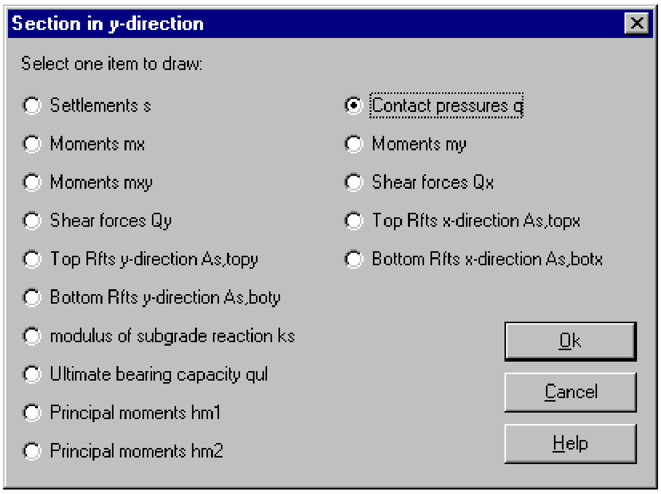


Figure F-17 "Section in y-direction" items

For drawing results at a specified section in y-direction, ranges of the section are required as shown in Figure F-18.

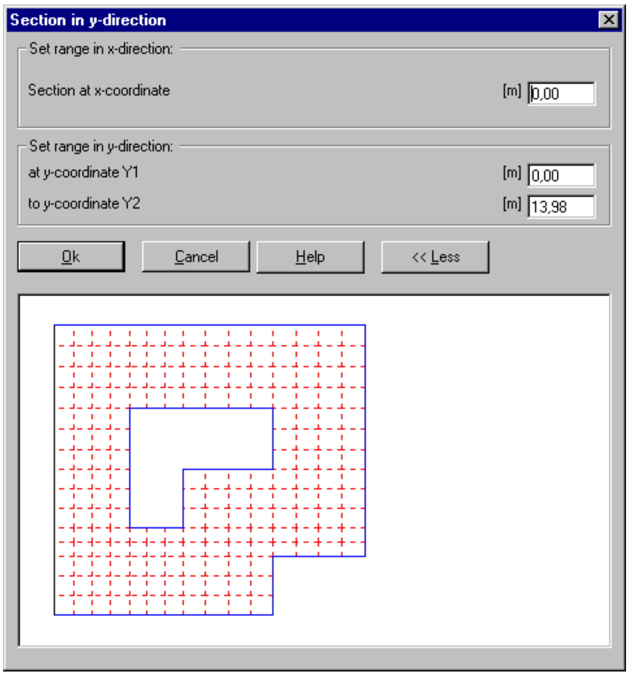


Figure F-18 "Section in y-direction" Dialog box

As an example, Figure F-19 shows a specified section of contact pressures in y-direction.

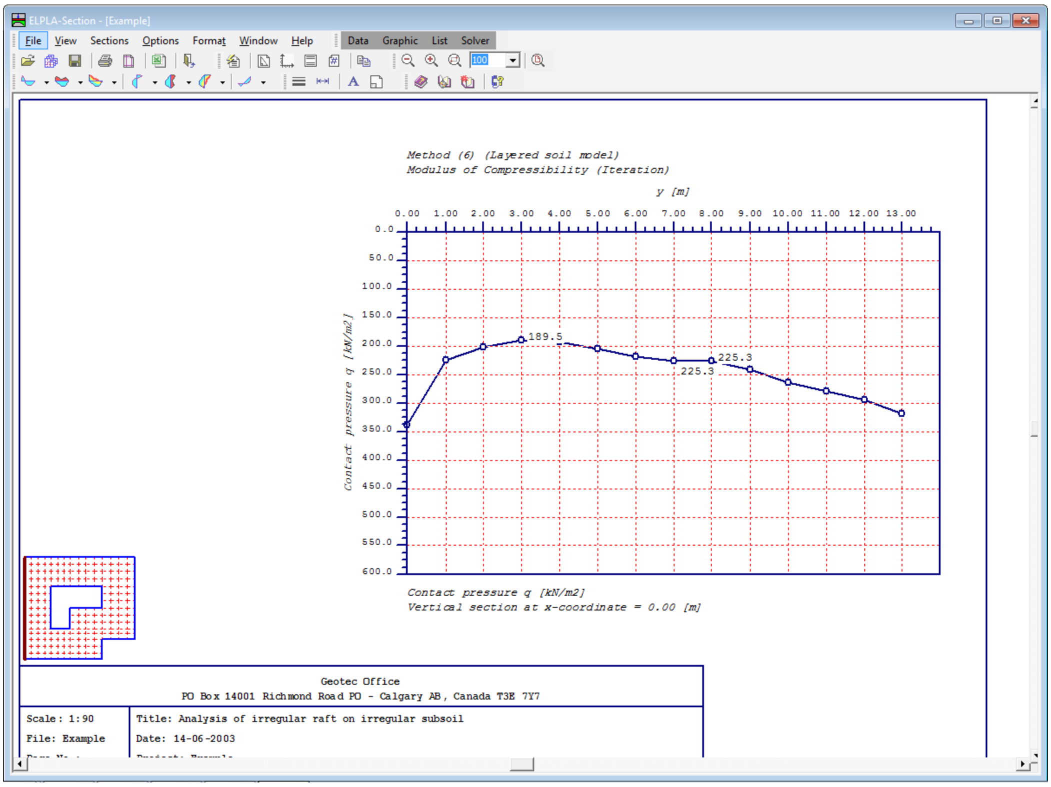


Figure F-19 Section of contact pressures in y-direction

6.5 Sections Menu–"Max./ Min. values in y-direction" command

By "Max./ Min. values in y-direction" command the extreme values of results (settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement) and input data (additional settlements) can be plotted at a specified range in y-direction.

When the command is chosen, the selection Dialog box shown in Figure F-20 appears. In this Dialog box select one item to draw, then click "OK" Button.

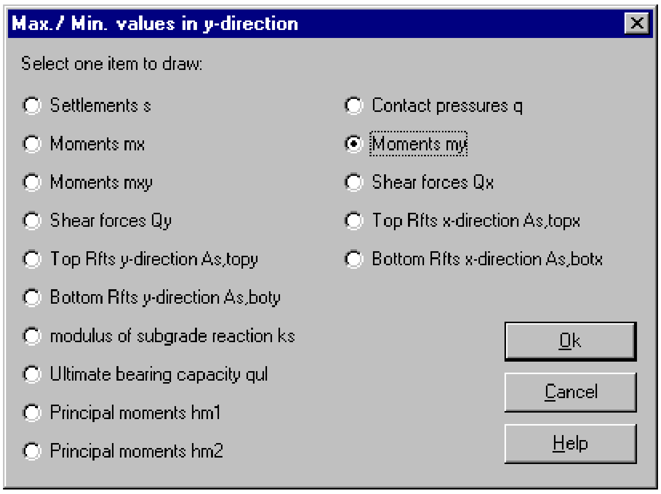


Figure F-20 "Max./ Min. values in y-direction" items

For drawing extreme values of results at a specified range in y-direction, ranges of the section are required as shown in Figure F-21.

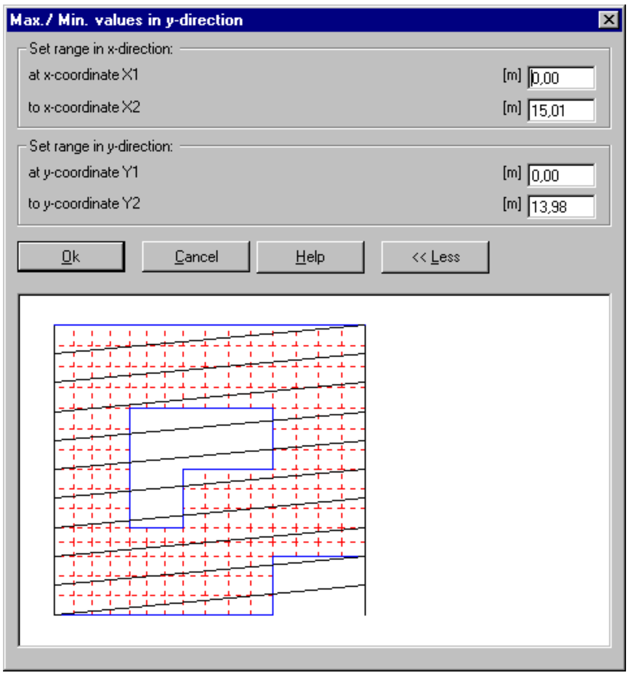


Figure F-21 "Section in y-direction" Dialog box

As an example, Figure F-22 shows the extreme values of moments m_y in y-direction for the whole raft.

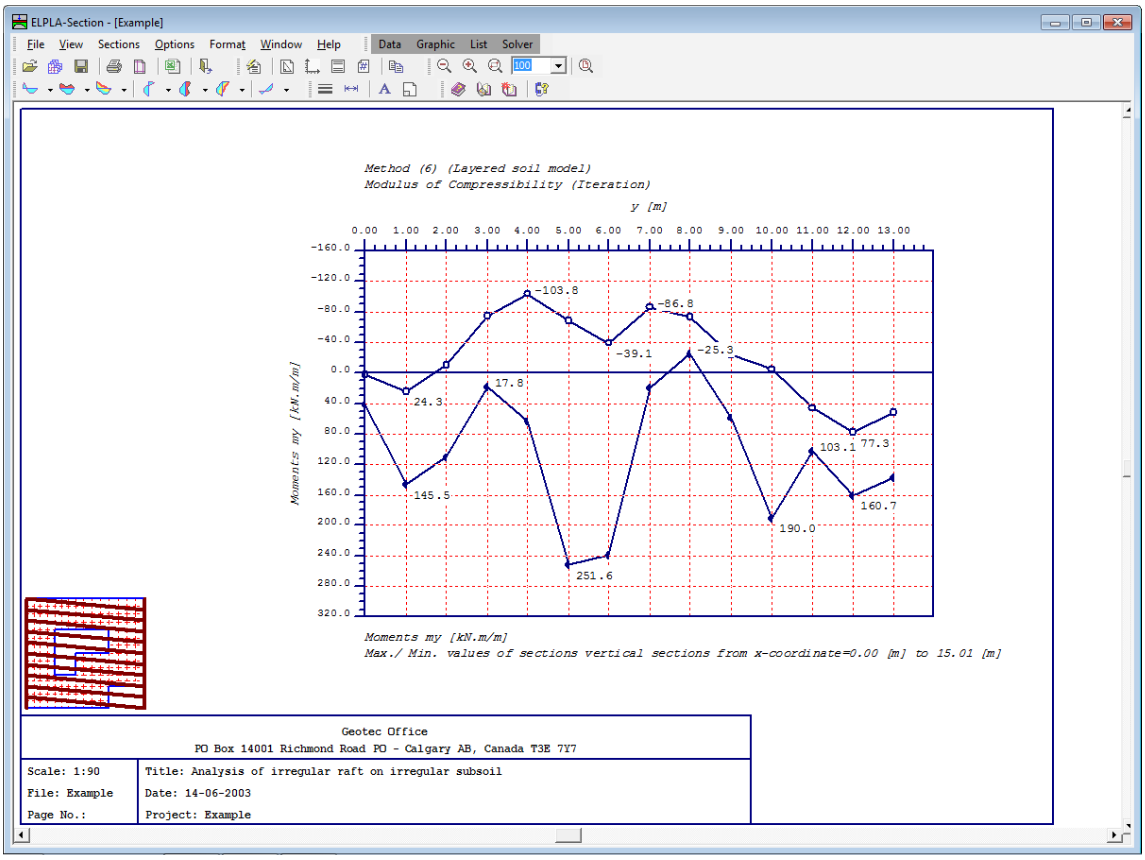


Figure F-22 The extreme values of moments m_y in y-direction for the whole slab

6.6 Sections Menu–"Overlapping in y-direction" command

By "Overlapping in y-direction" command the values of calculated results (settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement) and input data (additional settlements) can be plotted at a specified range in y-direction in one graph.

When this command is chosen, the selection Dialog box shown in Figure F-23 appears. In this Dialog box select one item to draw, then click "OK" Button.

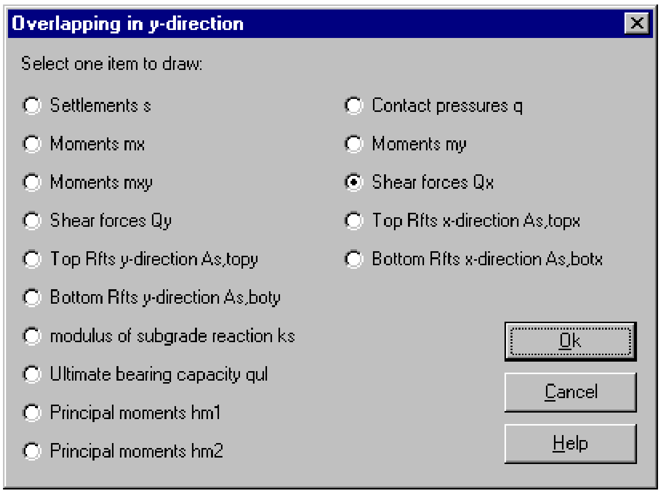


Figure F-23 "Overlapping in y-direction" items

For drawing values of calculated results at a specified range in y-direction in one graph, the ranges of the section required as shown in Figure F-24.

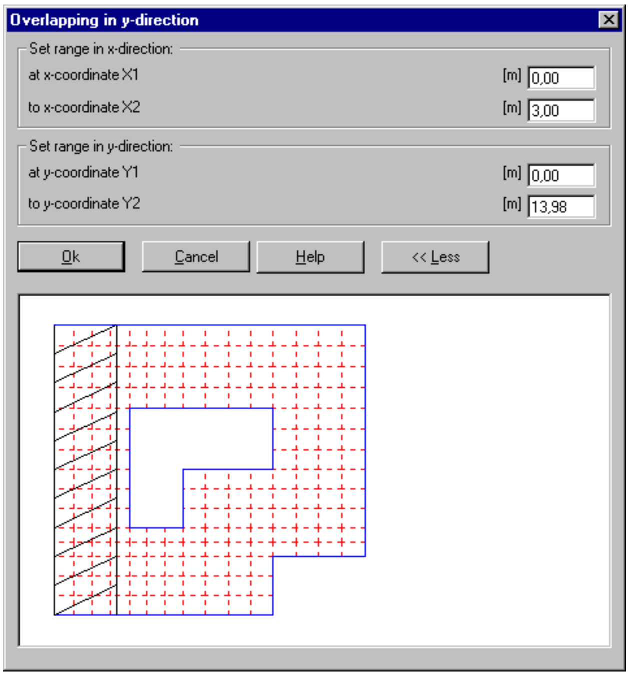


Figure F-24 "Section in y-direction" Dialog box

As an example, Figure F-25 shows values of shear forces Q_x in y-direction in a specified zone of the raft.

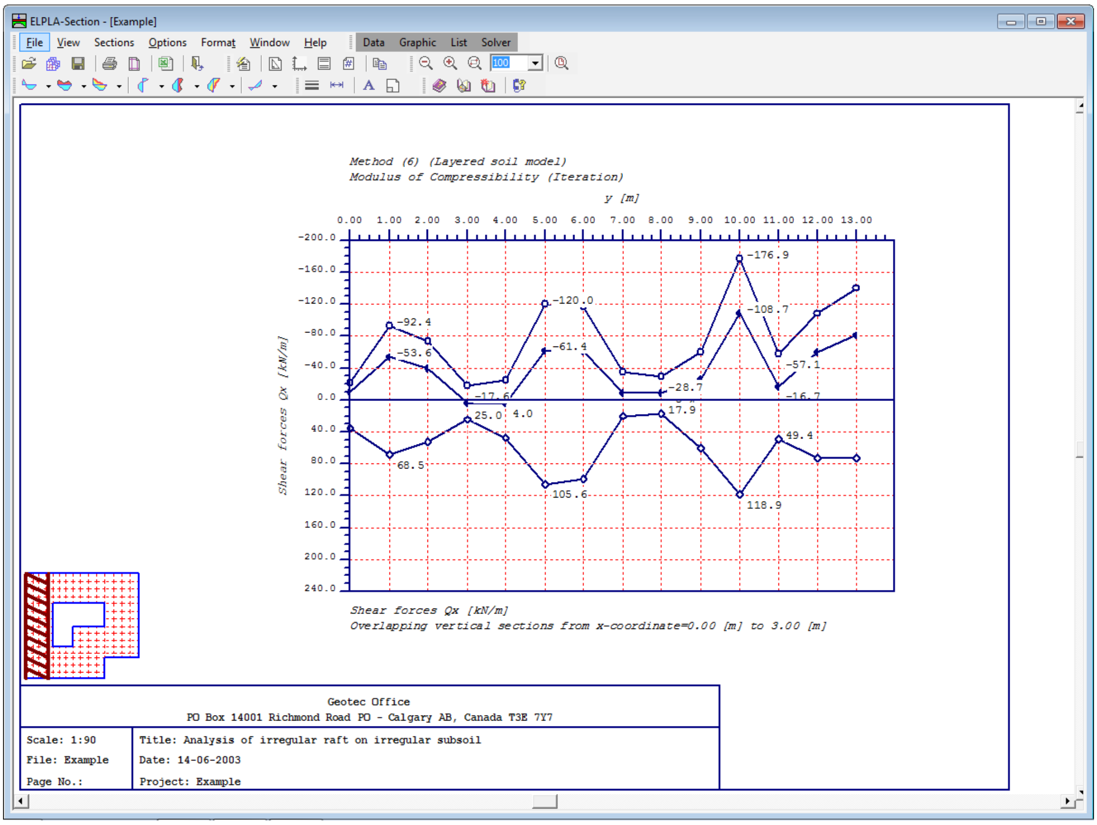


Figure F-25 Sections of shear forces Q_x in y-direction

6.7 Sections Menu–"Arbitrary section" command

By "Arbitrary section in x-direction" command the calculated results (settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction and reinforcement) and input data (additional settlements) can be plotted at any arbitrary section.

When "Arbitrary section" command is chosen, the selection Dialog box shown in Figure F-26 appears. In this Dialog box select one item to draw, then click "OK" Button.

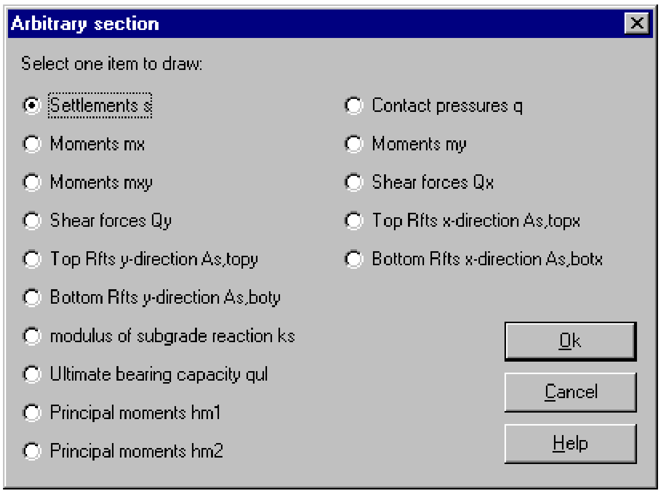


Figure F-26 "Section in x-direction" items

For drawing results at any arbitrary section, ranges of the section are required as shown in Figure F-27.

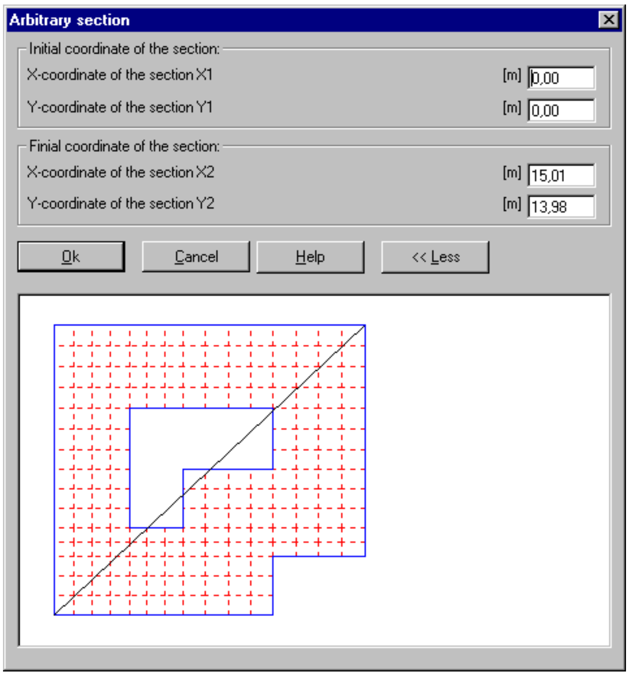


Figure F-27 "Section in x-direction" Dialog box

As an example, Figure F-28 shows an arbitrary section of settlement.

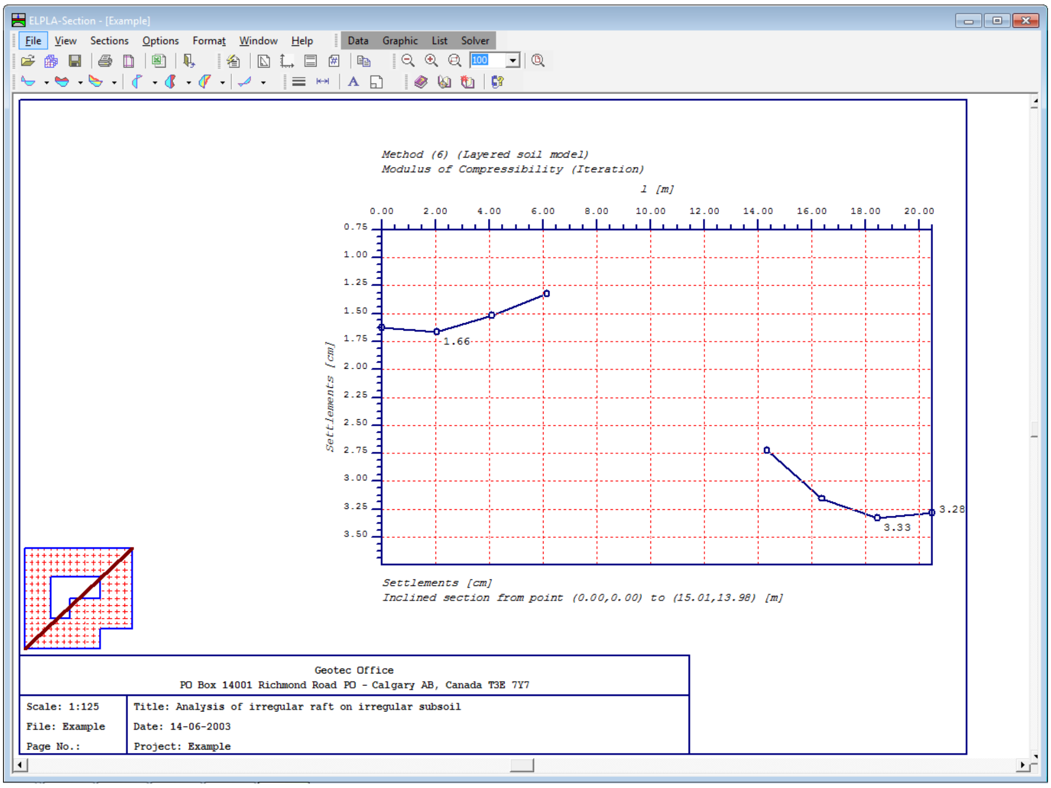


Figure F-28 Arbitrary section of settlement

7 Options Menu

The Options Menu has the following commands:

- Plot parameters
- Scale
- Axes
- Title
- Page No.
- Copy

7.1 Options Menu–"Plot parameters" command

Plot parameters may be set as default values by the program, or be fully specified by the user. The following plot parameters can be specified (Figure F-29):

- Display grid
- Page with or without frame
- Display values on lines
- Display marks on lines

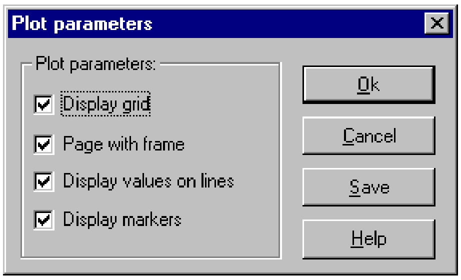


Figure F-29 "Plot parameters" Dialog box

7.2 Options Menu–"Scale" command

By "Scale" command the scale factor of the drawing can be defined as shown in Figure F-30. The defaulted value for the scale factor is chosen to pass the active printer-paper format.

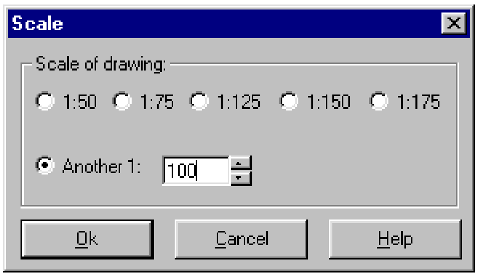


Figure F-30 "Scale" Dialog box

7.3 Options Menu–"Axes" command

By "Axes" command scaled axes can be defined on the drawing, Figure F-31.

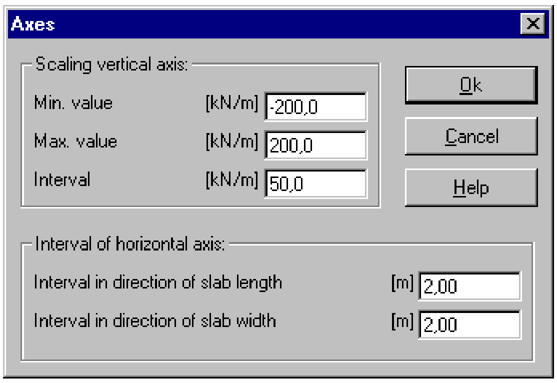


Figure F-31 "Axes" Dialog box

7.4 Options Menu–"Title" command

By "Title" command the text data (two line texts above the drawing and two line texts under the drawing) can be defined, Figure F-32. Default texts are:

- A-Title1: Method No.
- A-Title2: Name of method
- U-Title1: Name of drawing
- U-Title2: Ranges that define the section

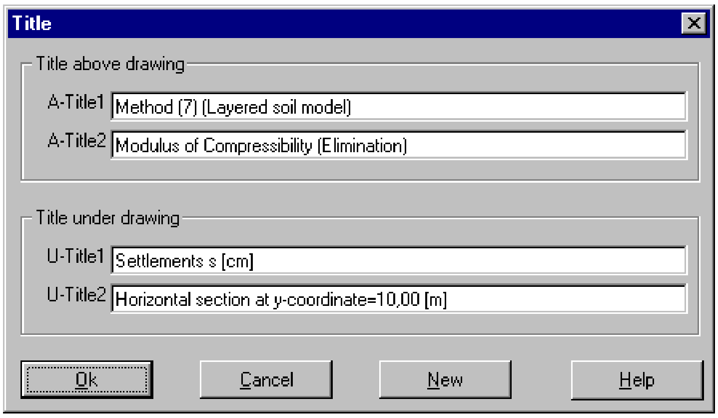


Figure F-32 "Title" Dialog box

7.5 Options Menu–"Page No." command

By "Page No." command the page No. can be defined, Figure F-33.

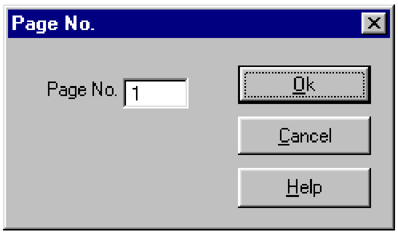


Figure F-33 "Page No." Dialog box

7.6 Options Menu–"Copy" command

By "Copy" command the current drawing can be copied in Metafile-Format to Clipboard. Then it can be inserted directly to another Windows programs such as Word, WordPerfect and AutoCAD.

8 Format Menu

The Format menu has the following commands:

- Line formats
- Max. ordinate
- Font
- Legend

8.1 Format Menu–"Line formats" command

Color, style and thickness of drawing lines can be defined, Figure F-34. The way a line is drawn depends on the setting of the color and style properties. There are available 15 different colors and 5 styles for line formats.

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

- Grid
- Page boundary
- Identification box
- Axes
- Location of the section
- Sections

Note: The asterisk (*) matches drawing section number.

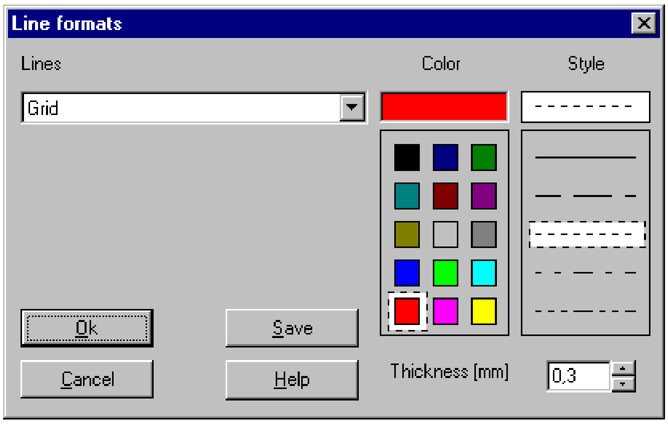


Figure F-34 "Line formats" Dialog box

8.2 Options Menu–"Max. ordinate" command

By "Max. ordinate" command the maximum ordinate of the vertical axis and the maximum size of markers can be defined in [mm], Figure F-35.

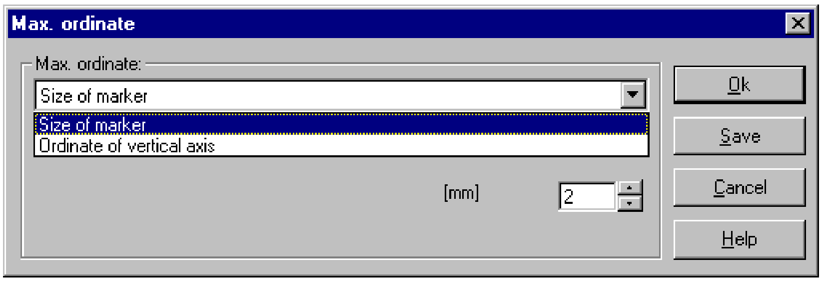


Figure F-35 "Max. ordinate" Dialog box

8.3 Format Menu–"Font" command

By "Font" command font size (Figure F-36) and font type (Figure F-37) for the drawing can be defined.

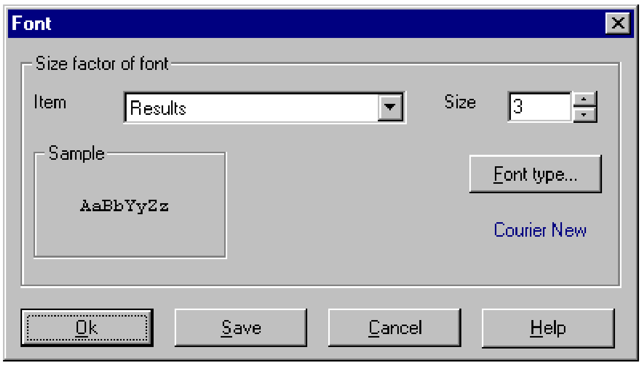


Figure F-36 "Font size" Dialog box

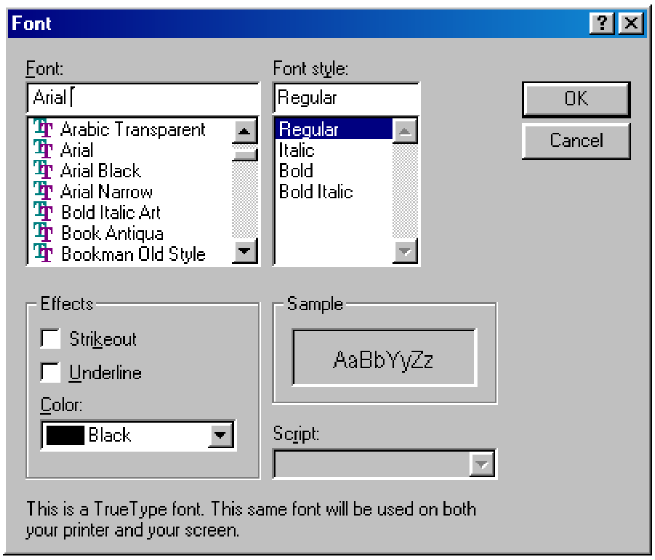


Figure F-37 "Font" Dialog box

8.4 Format Menu–"Legend" command

By "Legend" command the height and width of legends (Identification box, symbol and location of the section) can be specified as shown in Figure F-38.

Display legend
In "Display legend" Group box check the legend that you wish to display.

Legend size
In "Legend size" Group box the height and width of the legend can be defined.

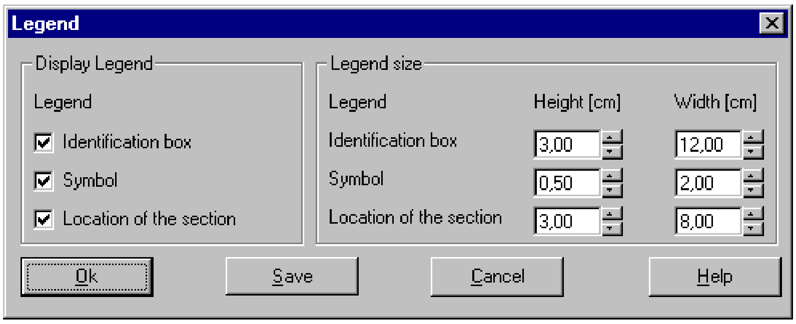


Figure F-38 "Legend" Dialog box

9 Window Menu

The Window menu has the following commands:

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

9.1 Window Menu–"Zoom in" command

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

9.2 Window Menu–"Zoom out" command

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

9.3 Window Menu–"Zoom window" command

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

9.4 Window Menu–"Zoom %" command

When you choose "Zoom %" command, the following Dialog box appears, Figure F-39.

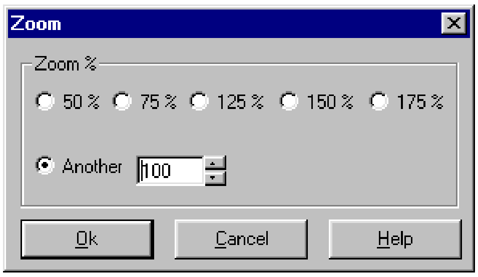


Figure F-39 "Zoom %" Dialog box

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

9.5 Window Menu–"Original size" command

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

10 Help Menu

The Help Menu commands are:

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

10.1 Help Menu–"Contents" command

"Contents" command displays a help file in HTML-Format containing the complete ELPLA User's Guide, Figure F-40.

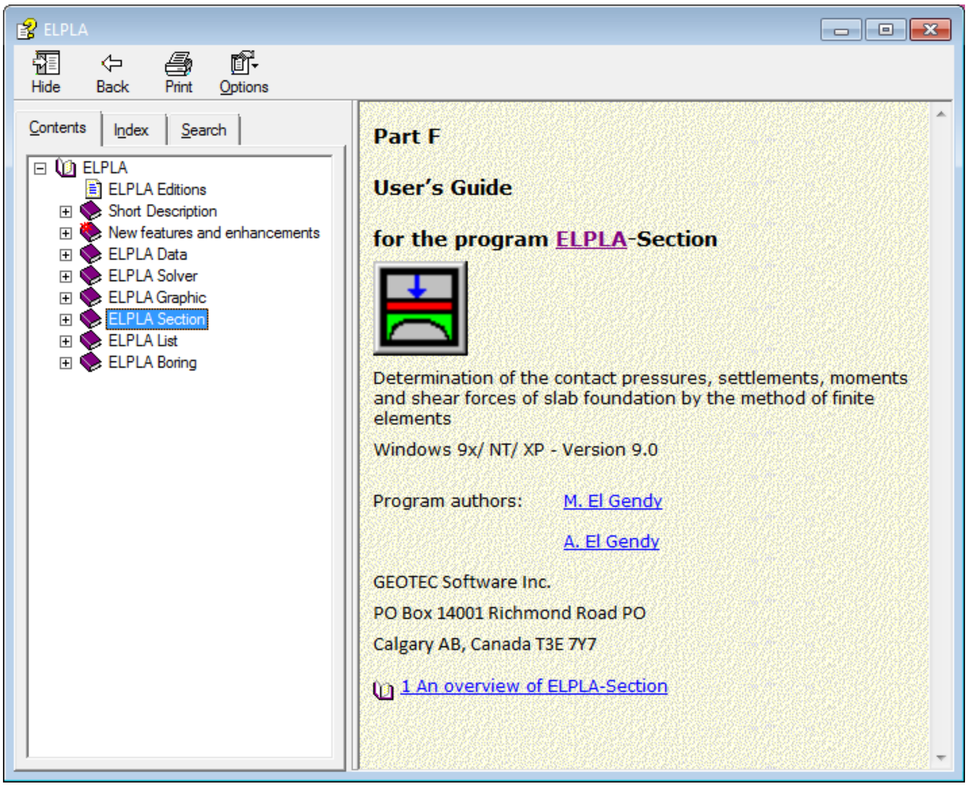


Figure F-40 Menu "Contents"

10.2 Help Menu–"Short description of ELPLA" command

"Short description of ELPLA" command gives a short description of ELPLA package.

10.3 Help Menu–"New in ELPLA" command

"New in ELPLA" command summarizes the new features and enhancements in ELPLA.

10.4 Help Menu–"About ELPLA-Section" command

Clicking the command "About ELPLA-Section" displays the information form of ELPLA-Section as shown in Figure F-41 This gives information about ELPLA-Section and the calculation method of the loaded project.

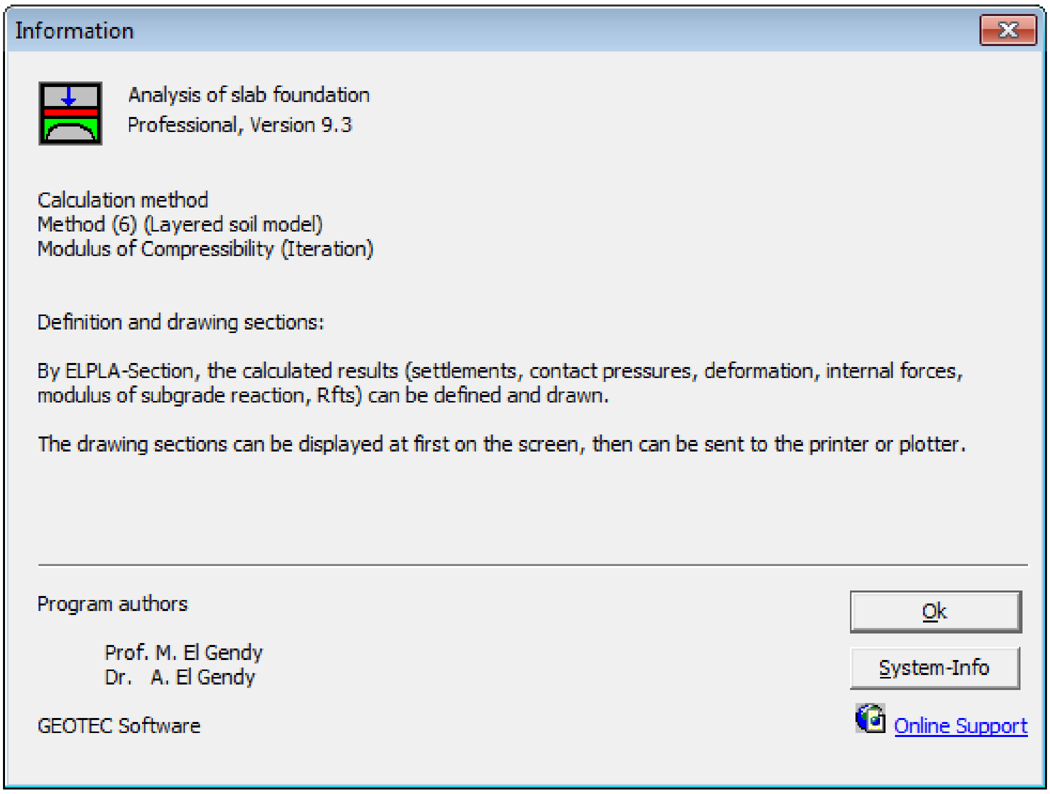


Figure F-41 Information form of ELPLA-Section

11 Tips and Tricks

11.1 Keyboard

The user can obtain all menu titles and commands also through Shortcut keys. The action of the Shortcut keys is listed in Table F-4 to Table F-11:

Table F-4 Shortcut keys of menu head

Shortcut keys	Action
[Alt+f]	Calling menu head "File"
[Alt+v]	"View"
[Alt+s]	"Sections"
[Alt+o]	"Options"
[Alt+t]	"Format"
[Alt+w]	"Window"
[Alt+h]	"Help"

Table F-5 Shortcut keys of File-Command

Shortcut keys	Action
[Ctrl +o] or [Alt+f] then [o]	Calling command "Open"
[Alt+f] then [c]	"Combination of many projects"
[Alt+f] then [w]	"Make WMF-File"
[Ctrl +p] or [Alt+f] then [p]	"Print"
[Alt+f] then [u]	"Page setup"
[Alt+f] then [s]	"Send to Excel"
[Alt+f] then [1]	Calling the first project from the last four loaded projects
[Alt+f] then [2]	Calling the second project from the last four loaded projects
[Alt+f] then [3]	Calling the third project from the last four loaded projects
[Alt+f] then [4]	Calling the fourth project from the last four loaded projects
[Ctrl+q] or [Alt+f] then [x]	Calling command "Exit"

Table F-6 Shortcut keys of View-Command

Shortcut keys	Action
[Alt+v] then [b]	Calling command "Status bar"
[Alt+v] then [t]	"Tool bars"
[Alt+v] then [t], then [f]	"Tool bars-File"
[Alt+v] then [t], then [s]	"Tool bars-Sections"
[Alt+v] then [t], then [o]	"Tool bars-Options"
[Alt+v] then [t], then [t]	"Tool bars-Format"
[Alt+v] then [t], then [w]	"Tool bars-Window"
[Alt+v] then [t], then [h]	"Tool bars-Help"
[Alt+v] then [t], then [r]	"Reset Toolbar"

Table F-7 Shortcut keys of Sections-Command

Shortcut keys	Action	
[Alt+s] then [x]	Calling command	"Section in x-direction"
[Alt+s] then [m]		"Max./ Min. values in x-direction"
[Alt+s] then [o]		"Overlapping in x-direction"
[Alt+s] then [y]		"Section in y-direction"
[Alt+s] then [a]		"Max./ Min. values in y-direction"
[Alt+s] then [r]		"Overlapping in y-direction"
[Alt+s] then [a]		"Arbitrary section"

Table F-8 Shortcut keys of Options-Command

Shortcut keys	Action	
[Alt+o] then [l]	Calling command	"Plot parameters"
[Alt+o] then [s]		"Scale"
[Alt+o] then [a]		"Axes"
[Alt+o] then [t]		"Title"
[Alt+o] then [p]		"Page No."
[Alt+o] then [c]		"Copy"

Table F-9 Shortcut keys of Format-Command

Shortcut keys	Action	
[Alt+t] then [l]	Calling command	"Line formats"
[Alt+o] then [x]		"Max. ordinate"
[Alt+t] then [f]		"Font"
[Alt+t] then [d]		"Legend"

Table F-10 Shortcut keys of Window-Command

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

Table F-11 Shortcut keys of Help-Command

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

11.2 Mouse

By double-clicking the left mouse Button on a specified screen position, the user can obtain almost menus of the program.

- By double-clicking on *Legend*, *Firm header*, *Title* or *Project identification*, the corresponding menu appears
- By double-clicking on *Scale* in the identification box, "Scale"-Menu appears
- By double-clicking on *File name* in the identification box, "Open"-Menu appears
- By double-clicking on *Page No*. in the identification box, "Page No."-Menu appears
- By clicking the right mouse Button at any position on the screen, the user can also obtain the Popup- Options-Menu, Figure F-42

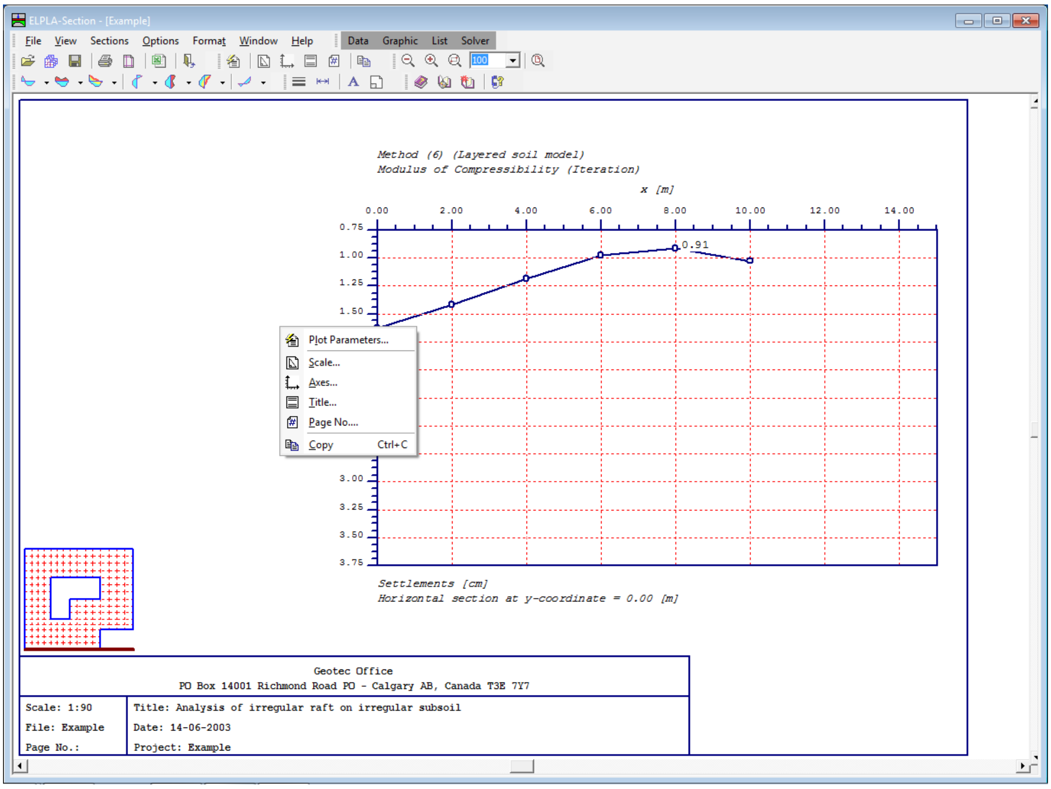


Figure F-42 Menu "Popup- Options"

12 Samples for graphical drawings using ELPLA-Section

ELPLA-Section gives the ability to present many different graphical drawings for results. In this paragraph the graphical drawings for results of example problem gb7, an irregular raft on irregular subsoil, is presented.

The pages P1 to P3 show some graphical presentations of example gb7 in different possibilities according to Table F-12. There is also a great number of presentation possibilities that are not consider in this paragraph. Color presentation is also possible by ELPLA-Section.

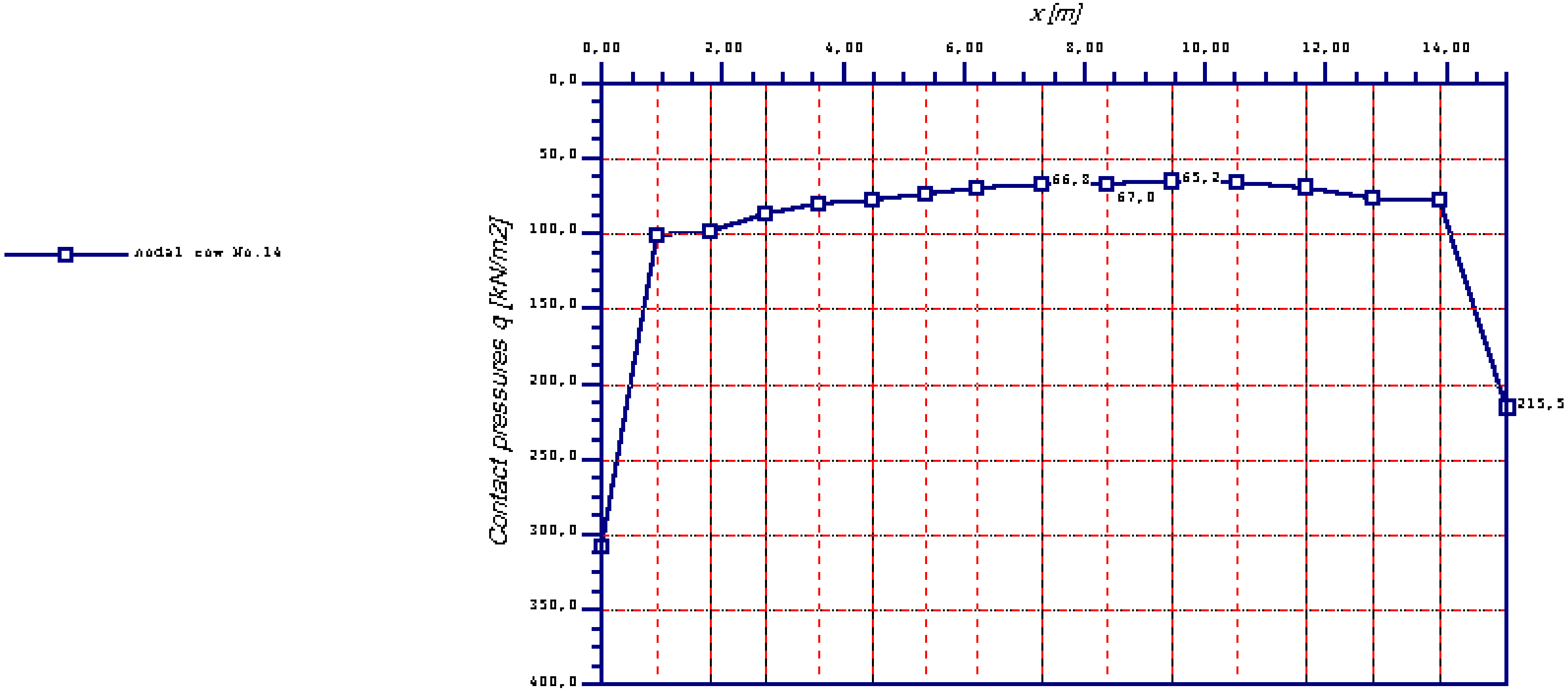
Table F-12 Graphical drawings for results

Presentation	Page
Contact pressure q	P1
Max./ Min. values of moments m_x	P2
Overlapping of 3 Sections of shear forces Q_y	P3

13 Index

Method (6)

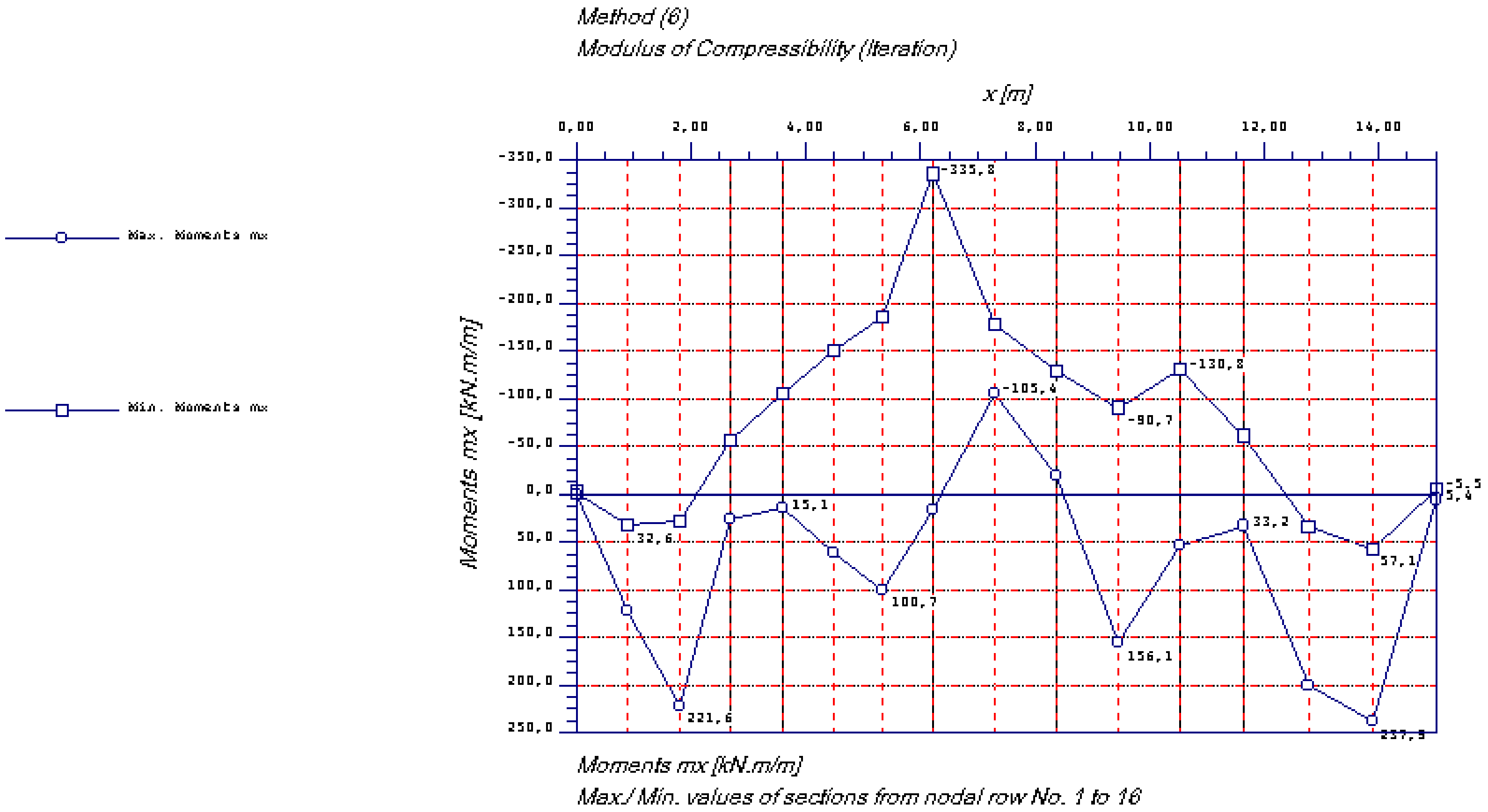
Modulus of Compressibility (Iteration)



Contact pressures q [kN/m2]

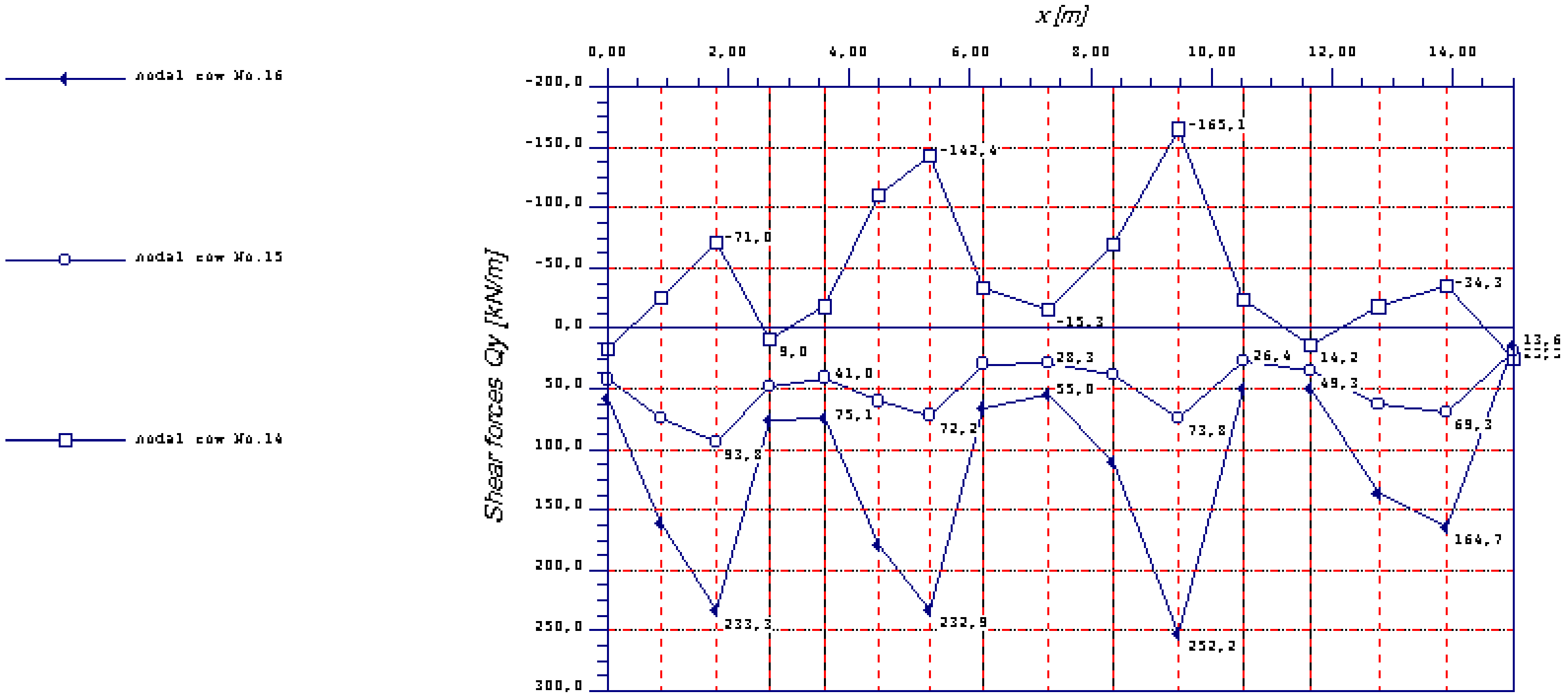
Section at nodal row No. 14

GEOTEC-Software	
Scale: 1: 100	Title: An irregular raft on irregular subsoil
File: gb7	Date: 13.07.1998
Page No.: 21	Project: Method 7



GEOTEC-Software	
Scale: 1: 100	Title: An irregular raft on irregular subsoil
File: [illegible]	Date: [illegible]
Page No.: [illegible]	Project: Method [illegible]

Method (6)
Modulus of Compressibility (Iteration)



Shear forces Qy [kN/m]
Overlapping sections from nodal row No. 14 to 16

GEOTEC-Software	
Scale: 1: 100	Title: An irregular raft on irregular subsoil
File: gb1	Date: 13.01.1998
Page No.: 83	Project: Method 1

Part G

User's Guide
for the program ELPLA-List

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

Version 9.2

Program authors: M. El Gendy
A. El Gendy

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

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

Table of Contents Page

1 An overview of ELPLA- List

ELPLA-List is used to display and print input data and calculated results. Listing data and results can be displayed first on the screen, and then can be sent to the printer or MS Word.

The data and results can be viewed as follows:

- 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

The data and results, if desired, can be saved as ASCII-format Files, in which they can be exported to other Windows applications to prepare reports or add further information.

2 Description of ELPLA-List

ELPLA-List is a 32-bit software product that operates under MS Windows 9x/ NT/ ME/ XP. The common "what you see is what you get" of Windows applications makes it easy to learn how to use ELPLA-List, especially if you are already familiar with the Windows environment.

The program package ELPLA consists of 7 separate programs. They can be run independently. The name and short description of the seven separate programs are given in Table G-1. The usage of the program is typically such that first data files are created describing a certain problem by ELPLA-Data. Then the project problem is analyzed by ELPLA-Solver. Finally, the results can be presented as graphical drawing, graphs and tables using the five separate programs ELPLA-Graphic, ELPLA-Section, ELPLA-List, ELPLA-Boring and GEOTEC-Editor.

Table G-1 Names and description of the seven separate programs

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

In order to use ELPLA-List, first the user must define the project data by ELPLA-Data, and then analyze the problem by ELPLA-Solver. Table G-2 gives a list of files, which are read or created by ELPLA-List. The files can be classified in four groups.

Table G-2 Names of file groups

Group	Saved from the program
A Main data files	ELPLA-Data
B List files (*.LST)	ELPLA-List
C Project data files	ELPLA-Data
D Result files	ELPLA-Solver

Further more, Table G-3 shows filename, contents and groups of all files that may be read by ELPLA-List.

Table G-3 Names and contents of files

A Main data files

Filename	Contents
FIRMA	Firm header
STEU	Default directory for files that are saved by ELPLA
NOFORMAT	Number formats
RFT	Design code parameters
UNITS	System of units

B List file

Filename	Contents
PAGELAYO.LST	Page format
FONT.LST	Font format

C Project data files

Filename	Contents
*. AUF	3 lines text to identify the project
*. BAU	Soil properties
*. LDH	Data of limit depth
* .PO1	System data (Analysis of isolated raft)
*. PC1	Load data
*. PL6	FE-Net data
*. P21	Data of slab properties/ levels/ coordinates
*. P23	Reinforcement data
*. GL1	Girder data
*. P31	Data of supports/ boundary conditions
*. P35	Data of spring supports
*. P41	File of boring fields
*. PT1	Data for temperature change
*. PP1	File of neighboring foundations
*. PV1	Data of additional soil settlements

D Result files

Filename	Contents
*. PL4	Area around nodes
*. PC7	Foundation properties
*. PC8	Average contact pressure, eccentricity and area of slab
*. PS1	Deformations of the rigid slab (w_o, tan θ_{xo}, tan θ_{xo})
*. PT2	Load vector due to temperature change
*. PP2	Load vector due to neighboring foundations
*. PW2	Moduli of subgrade reactions of borings
*. QUB	Main ultimate bearing capacity (q_b)
*. PD1	Flexibility matrix for loading
*. PD2	Flexibility matrix for reloading
*. PE1	Soil stiffness matrix
*. LD1	Limit depth
*. GH1	Internal forces of grid elements
*. PT3	Displacements due to temperature change (s_t)
*. PP3	Settlements due to neighboring foundations (s_e)
*. PV2	Load vector due to additional settlements
*. QUN	Ultimate bearing capacities at nodes (q_{ult})
*. PW1	Modulus of subgrade reaction (k_s)
*. PH1	Settlements (s)
*. PH2	Contact pressures (q)
*. PH3	Moments (m_x)
*. PH4	Moments (m_y)
*. PH5	Moments (m_{xy})
*. PH6	Shear forces (Q_x)
*. PH7	Shear forces (Q_y)
*. PH8	Settlements due to reloading (s_w)
*. PH9	Overburden pressures (Q_u)
*. H10	Support reactions (V)
*. H11	Support reactions (M_y)
*. H12	Support reactions (M_x)
*. H13	Reinforcement of the slab (A_{sx1})
*. H14	Reinforcement of the slab (A_{sx2})
*. H15	Reinforcement of the slab (A_{sy1})
*. H16	Reinforcement of the slab (A_{sy2})
*. THY	Rotations about y-axis (θ_y)
*. THZ	Rotations about z-axis (θ_z)
*. PPU	Punching results

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

Next paragraphs describe the purpose and function of each ELPLA-List command.

3 Starting ELPLA-List

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

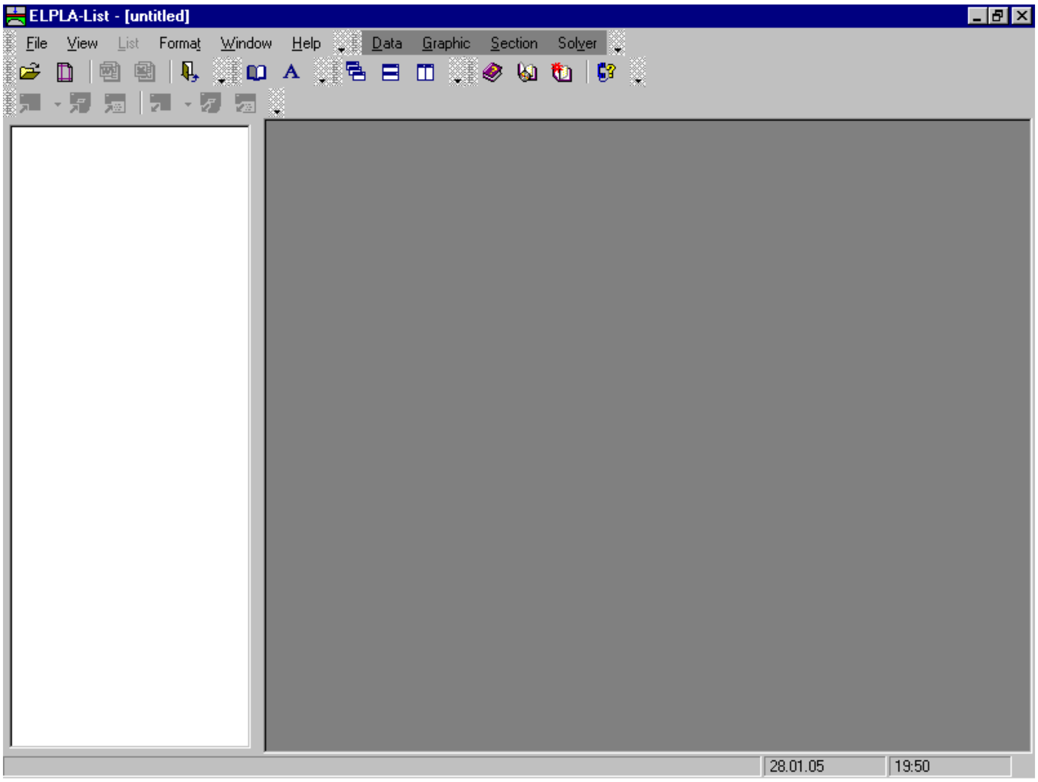


Figure G-1 Introduction screen of ELPLA-List

The menu head of Figure G-1 contains the following 6 commands:

- File
- View
- List
- Format
- Window
- Help

After clicking one of the six menu commands other sub-commands become available. These are presented and described in the following paragraphs 4 to 10.

4 File Menu

The File Menu commands are:

- Open
- Close project
- Print setup
- Send to Word
- Send to Excel
- Files 1, 2, 3, 4
- Exit

4.1 File Menu –"Open" command

By "Open" command the current project is closed, if one is loaded, and an existing project is opened. Figure G-2 shows "Open" Dialog box used to open a specified project.

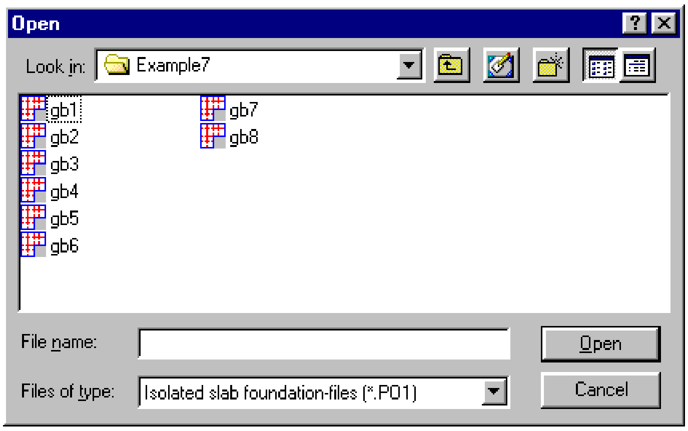


Figure G-2 "Open project" Dialog box

4.2 File Menu –"Close project" command

By "Close project" command the current project is closed.

4.3 File Menu –"Print setup" command

By this command the standard "Print setup" Dialog box can be displayed with options to specify printer, page orientation, paper size and paper source, as well as other printing options.

Table G-3 shows "Print setup" Dialog box. The printer group box contains controls for selecting the printer and changing its properties. Use "Name" combo box to select the printer and use "Properties" Button to set printer settings.

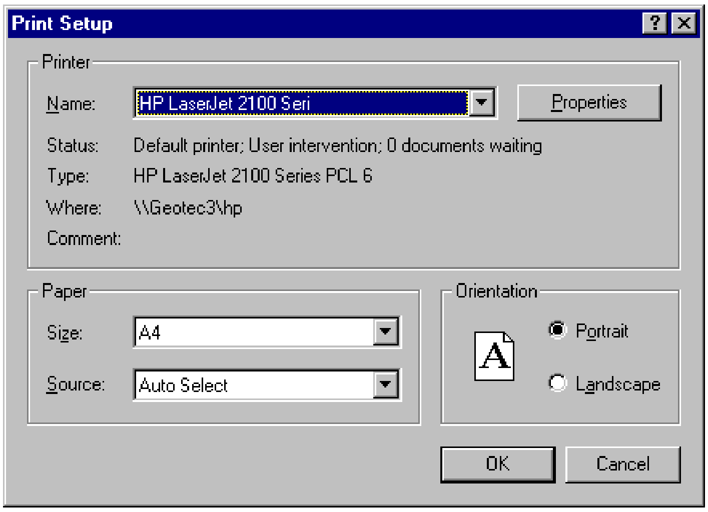


Figure G-3 "Print setup" Dialog box

4.4 File Menu –"Send to Word" command

By "Send to Word" command it is possible to export data to MS Word, Figure G-4.

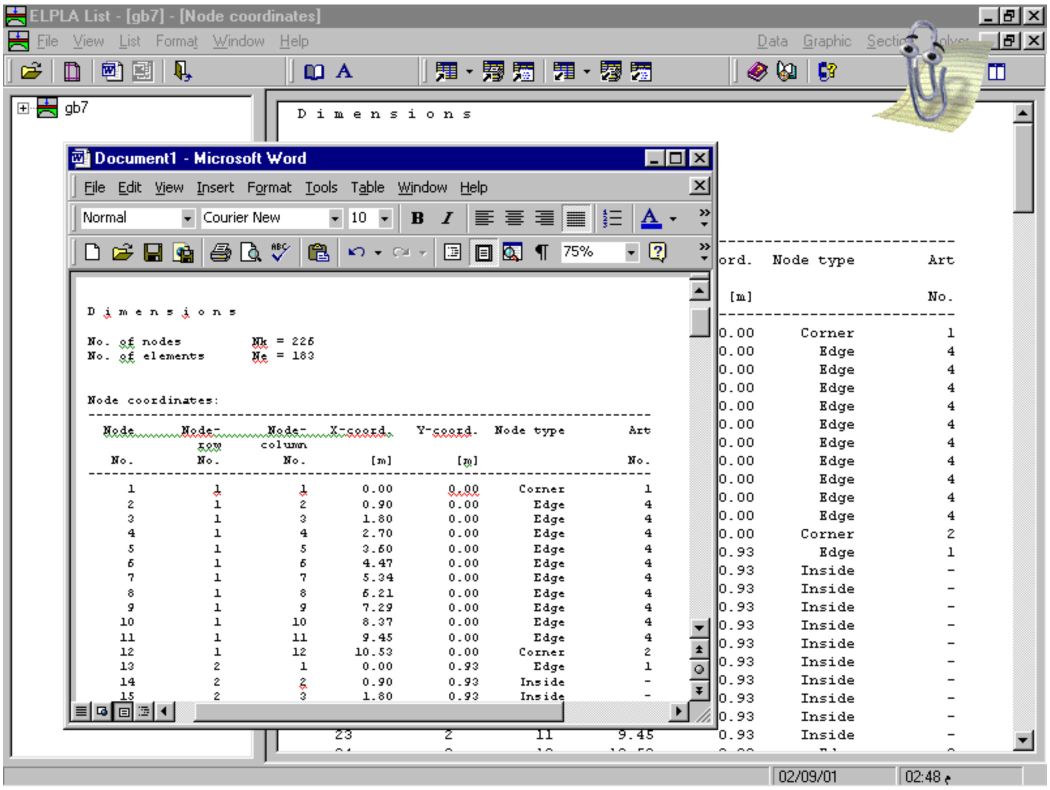


Figure G-4 Export data to MS Word

4.5 File Menu –"Send to Excel" command

By "Send to Excel" command it is possible to export results to MS Excel, Figure G-5.

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

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

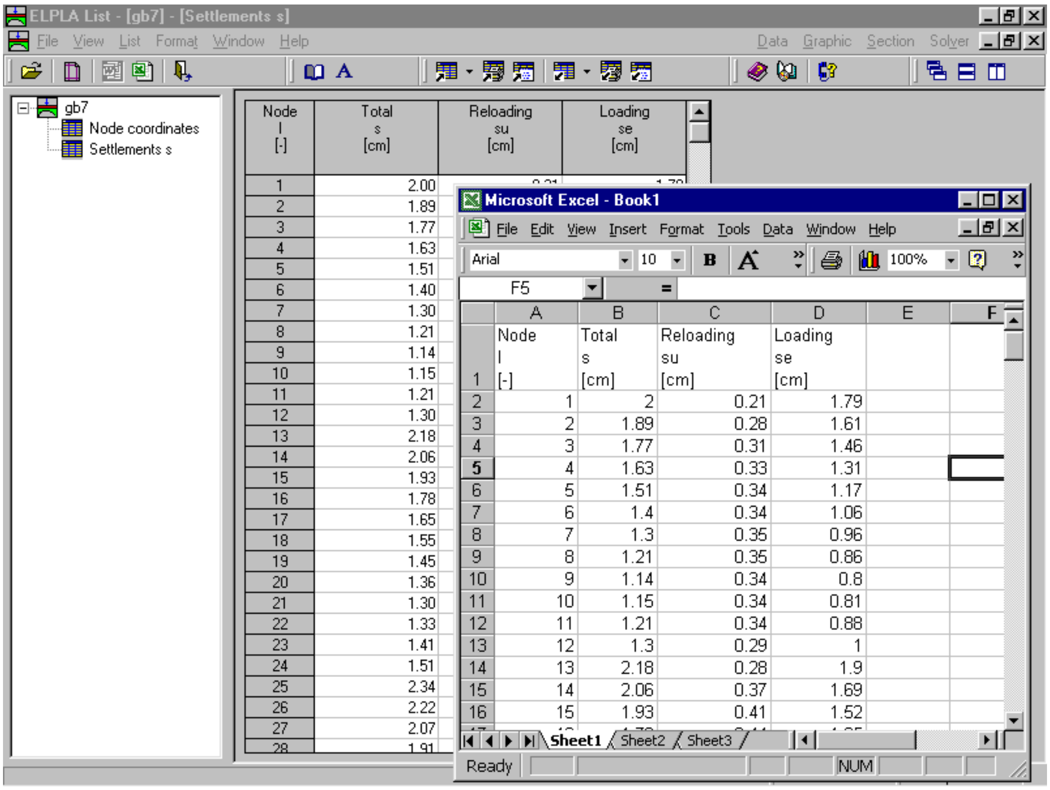


Figure G-5 Export results to MS Excel

4.7 File Menu –"Exit" command

By "Exit" command the current project is closed and ELPLA-List is quitted, Figure G-6.

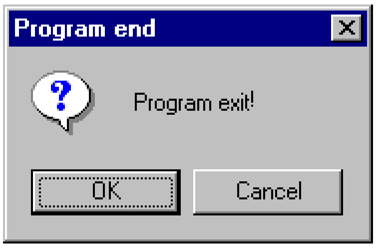


Figure G-6 "Exit" Message box

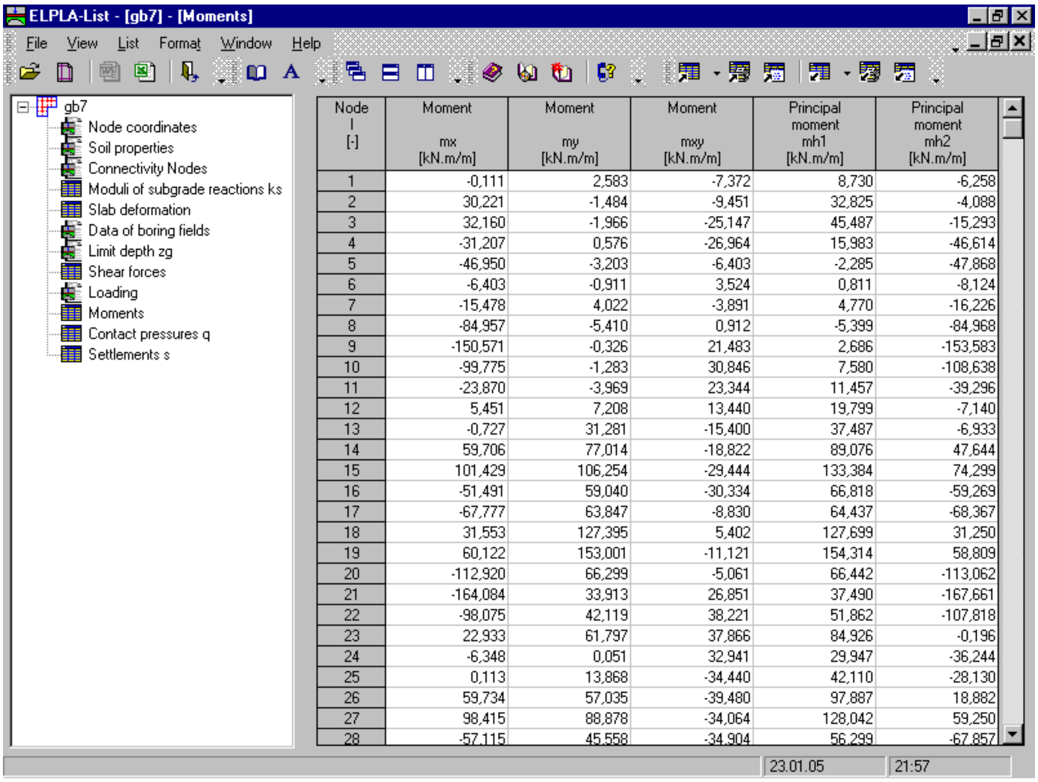
5 View Menu

The View Menu commands are:

- Project explorer
- Status bar
- Tool bars

5.1 View Menu–"Project explorer" command

"Project explorer" command displays an explorer window on the screen at the left. It is possible to swap between data and results easily, Figure G-7.



Node I [-]	Moment mx [kN.m/m]	Moment my [kN.m/m]	Moment mxy [kN.m/m]	Principal moment mh1 [kN.m/m]	Principal moment mh2 [kN.m/m]
1	-0,111	2,583	-7,372	8,730	-6,258
2	30,221	-1,484	-9,451	32,825	-4,088
3	32,160	-1,966	-25,147	45,487	-15,293
4	-31,207	0,576	-26,964	15,983	-46,614
5	-46,950	-3,203	-6,403	-2,285	-47,868
6	-6,403	-0,911	3,524	0,811	-8,124
7	-15,478	4,022	-3,891	4,770	-16,226
8	-84,957	-5,410	0,912	-5,399	-84,968
9	-150,571	-0,326	21,483	2,686	-153,583
10	-99,775	-1,283	30,846	7,580	-108,638
11	-23,870	-3,969	23,344	11,457	-39,296
12	5,451	7,208	13,440	19,799	-7,140
13	-0,727	31,281	-15,400	37,487	-6,933
14	59,706	77,014	-18,822	89,076	47,644
15	101,429	106,254	-29,444	133,384	74,299
16	-51,491	59,040	-30,334	66,818	-59,269
17	-67,777	63,847	-8,830	64,437	-68,367
18	31,553	127,395	5,402	127,699	31,250
19	60,122	153,001	-11,121	154,314	58,809
20	-112,920	66,299	-5,061	66,442	-113,062
21	-164,084	33,913	26,851	37,490	-167,661
22	-98,075	42,119	38,221	51,862	-107,818
23	22,933	61,797	37,866	84,926	-0,196
24	-6,348	0,051	32,941	29,947	-36,244
25	0,113	13,868	-34,440	42,110	-28,130
26	59,734	57,035	-39,480	97,887	18,882
27	98,415	88,878	-34,064	128,042	59,250
28	-57,115	45,558	-34,904	56,299	-67,857

Figure G-7 "Project explorer"-Window

5.2 View Menu–"Status bar" command

"Status bar" command displays a status bar on the screen down. The status bar displays information about the progress of the current operation.

5.3 View Menu–"Tool bars" command

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

6 List Menu

The List menu is the main menu, which is used to view or print the calculated results and input project data with the setting given by option format menus.

The Menu List commands 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

6.1 List Menu –"Display tables of data" command

By "Display tables of data" command the input project data can be tabulated.

The input project data, which can be tabulated, are:

- Node coordinates
- Node boundaries
- Data of supports/ Boundary conditions
- Spring supports
- Girders
- Loading
- Slab properties/ Foundation level/ Global coordinates
- Data of reinforcement
- Data of temperature change
- Data of neighboring foundations
- Data of additional soil settlements
- Data of boring fields
- Soil properties

When the command is chosen, the selection Dialog box shown in Figure G-8 appears. In this Dialog box select one item to be tabulated and click "OK" Button. As an example, Figure G-9 shows a list of soil data.

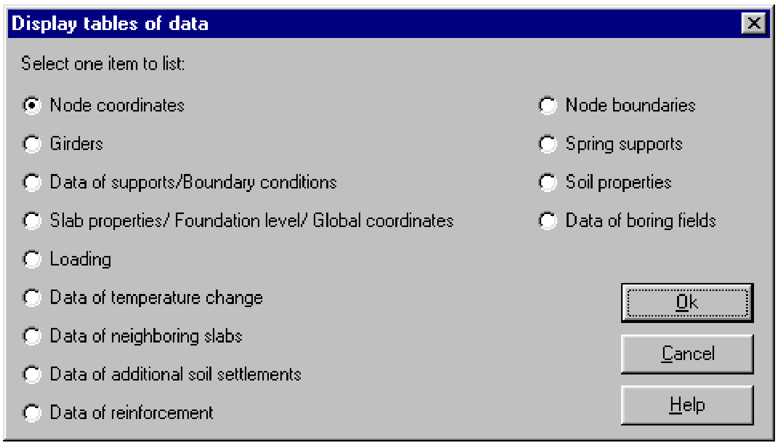


Figure G-8 "Display tables of data" items

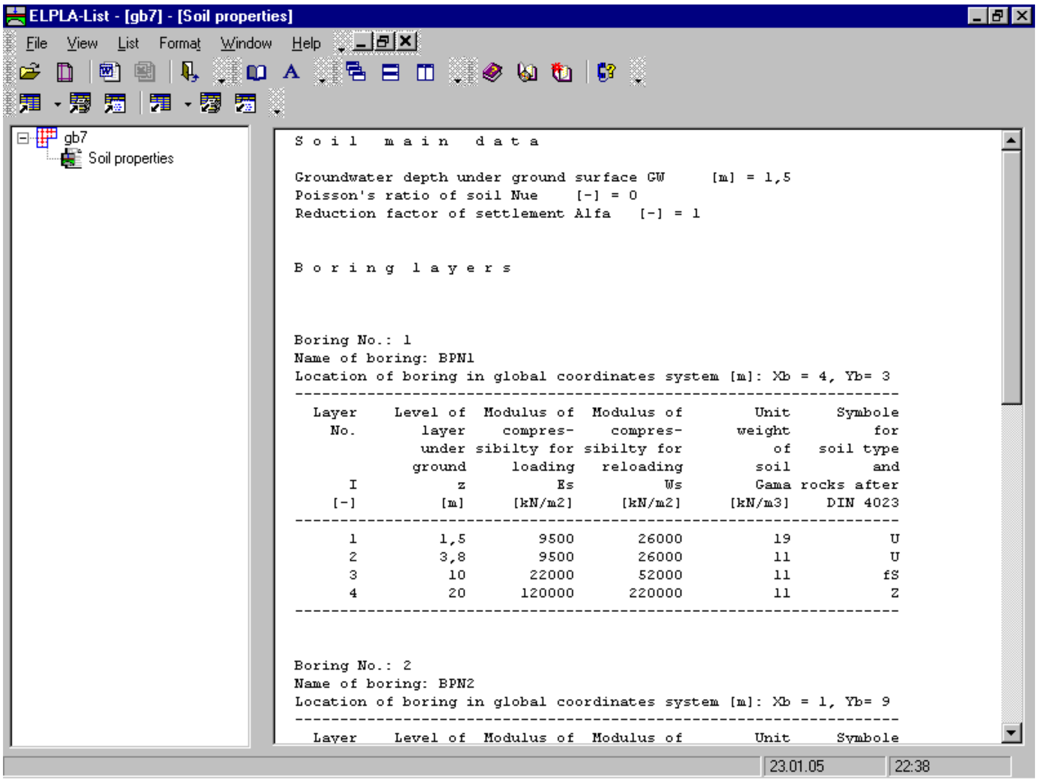


Figure G-9 List of node coordinates

6.2 List Menu –"Print tables of data" command

By "Print tables of data" command the title page of ELPLA can be printed. The input project data that can be viewed can be also printed. When "Print tables of data" command is chosen, the selection Dialog box shown in Figure G-10 appears. In this Dialog box check the items that you want to print, then click "OK" Button.

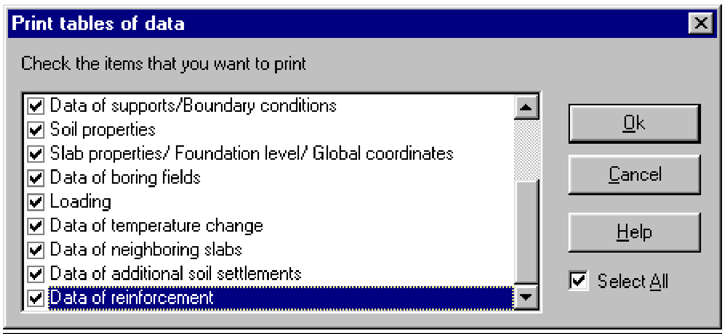


Figure G-10 "Print tables of data" items

For printing tables of data, the header options, page numbering and also number of copies are required as shown in Figure G-11.

Header

- To print headers (Company), check the control box "Print header (Company)"
- To print an identification header, check the control box "Print identification header"

Page numbering

- By the option "Start at" renumbering pages can be defined. To change the page number, type the new number in "Start at" Text box
- If the option "No page numbering" is activated, the page numbers will not be included in the document

Copies

- In Dialog box "No. of copies" the number of printing copies can be defined

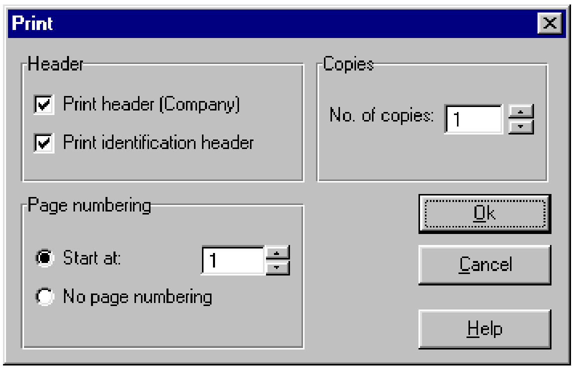


Figure G-11 "Print" Dialog box

6.3 List Menu –"List tables of data through Text-Editor" command

By this command the title page of ELPLA can be displayed through the ELPLA Text-Editor. The input project data that can be viewed can be also displayed. When the command is chosen, the selection Dialog box shown in Figure G-12 appears. In this Dialog box check the items that you want to list through Text-Editor, then click "OK" Button.

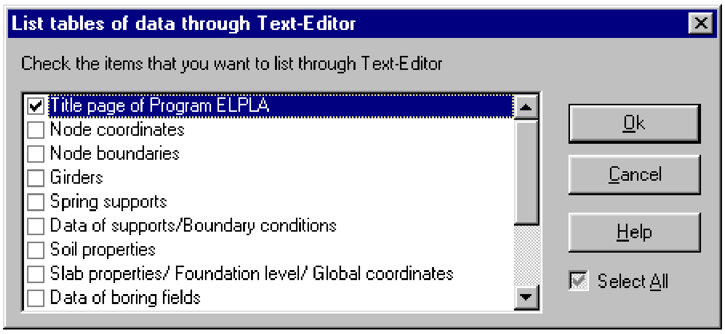


Figure G-12 "List tables of data through Text-Editor" items

For listing tables of data through Text-Editor, the header options and page numbering are required before loading Text-Editor. Figure G-13 shows Title page of ELPLA through the ELPLA Text-Editor (program GEOTEC-Editor). To get information about GEOTEC-Editor, see the User’s Guide of GEOTEC-Editor.

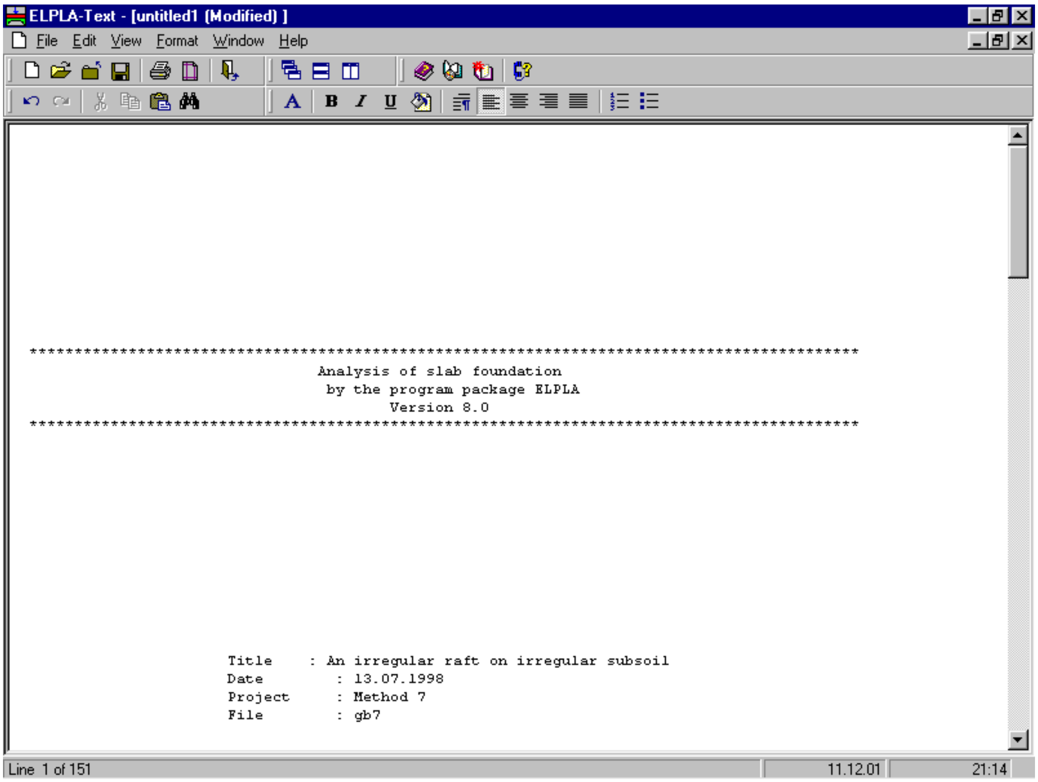


Figure G-13 Title page of ELPLA through GEOTEC-Editor

6.4 List Menu –"Display tables of results" command

By "Display tables of results" command the calculated results can be tabulated.

The calculated results, which can be tabulated, are:

- Settlements, contact pressures, deformation and internal forces
- Flexibility matrix [C_e]
- Flexibility matrix [C_W]
- Stiffness matrix [k_s]
- Modulus of subgrade reaction (k_s)
- Ultimate bearing capacity (q_{ult})
- Settlements due to temperature change (s_t)
- Settlements due to neighboring foundations (s_e)
- Additional soil settlements (s_s)
- Support reactions (V)
- Limit depth (z_g)
- Reinforcement (A_s)
- Punching results

When "Display tables of results" command is chosen, the selection Dialog box shown in Figure G-14 appears. In this Dialog box select one item to be tabulated, then click "OK" Button. As an example, Figure G-15 shows a list of settlements.

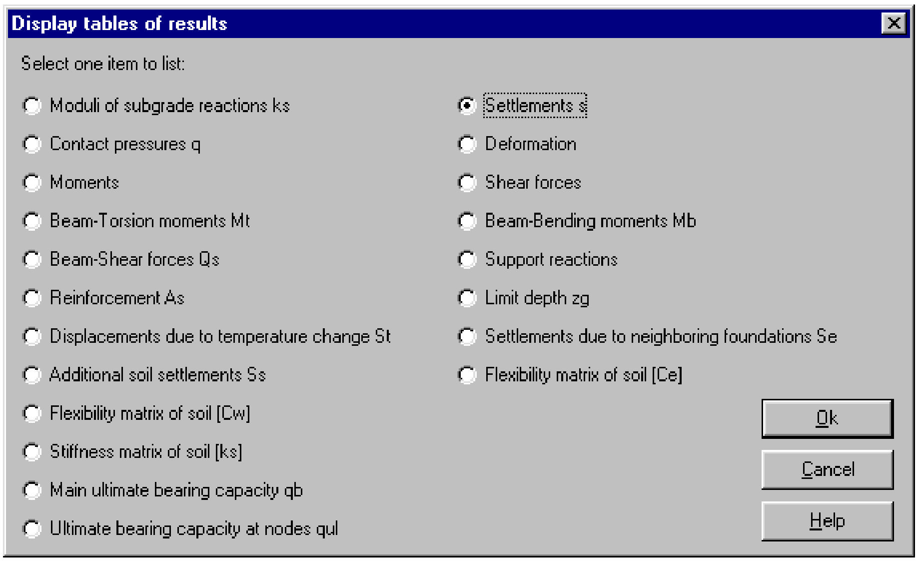


Figure G-14 "Display tables of results" items

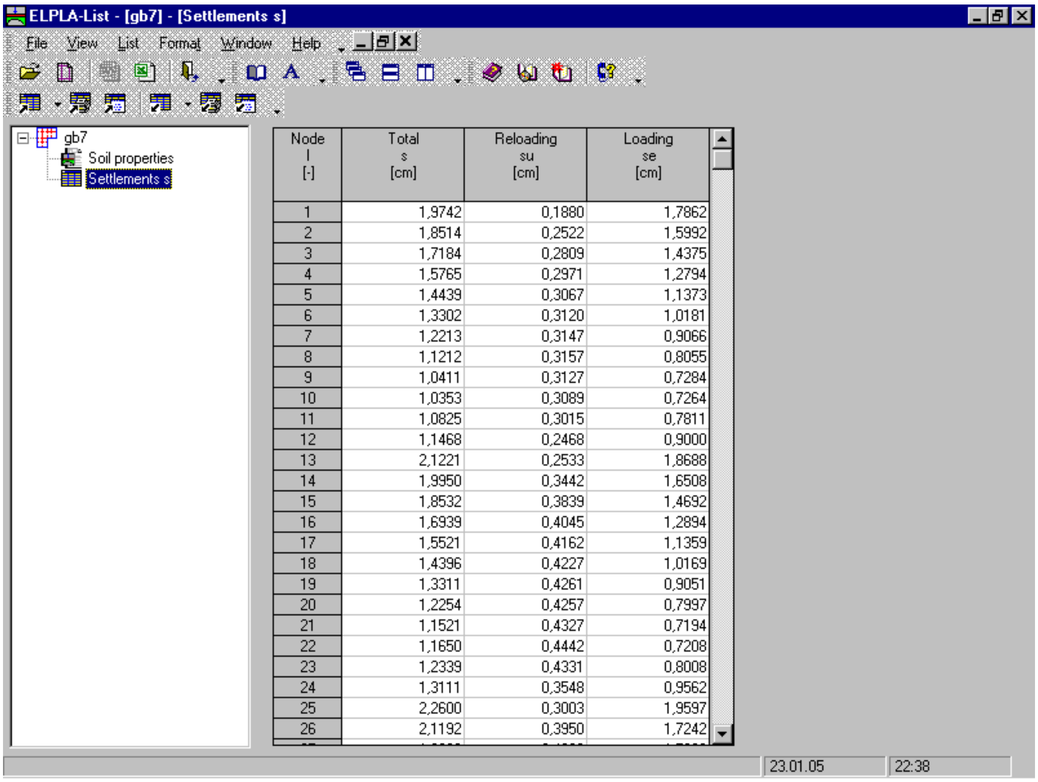


Figure G-15 List of settlements

6.5 List Menu –"Print tables of results" command

The calculated results can be printed. When "Print tables of results" command is chosen, the selection Dialog box shown in Figure G-16 appears. In this Dialog box check the items that you want to print, then click "OK" Button.

For printing tables of results, the header options, page numbering and also number of copies are required as shown in Figure G-11.

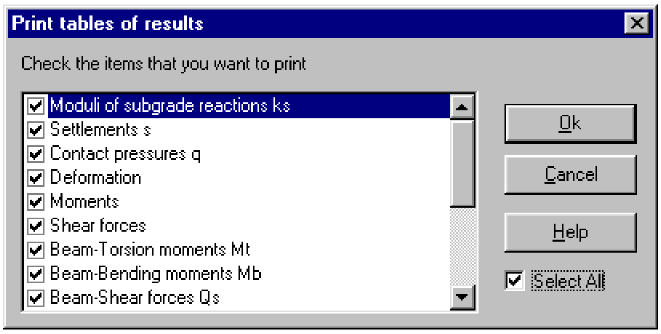


Figure G-16 "Print tables of results" items

6.6 List Menu –"List tables of results through Text-Editor" command

By "List tables of data through Text-Editor" command the calculated results can be displayed through ELPLA Text-Editor. When the command is chosen, the selection Dialog box shown in Figure G-17 appears. In this Dialog box check the items that you want to list through Text-Editor, then click "OK" Button.

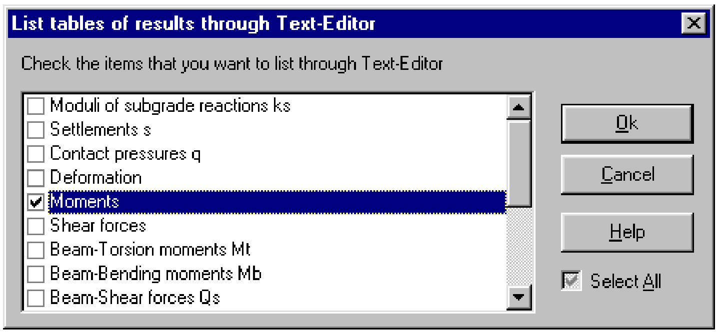


Figure G-17 "List tables of results through Text-Editor" items

For listing tables of data through Text-Editor, the header options and page numbering are required before loading Text-Editor. Figure G-18 shows moments through ELPLA Text-Editor. To get information about GEOTEC-Editor, see the User’s Guide of GEOTEC-Editor.

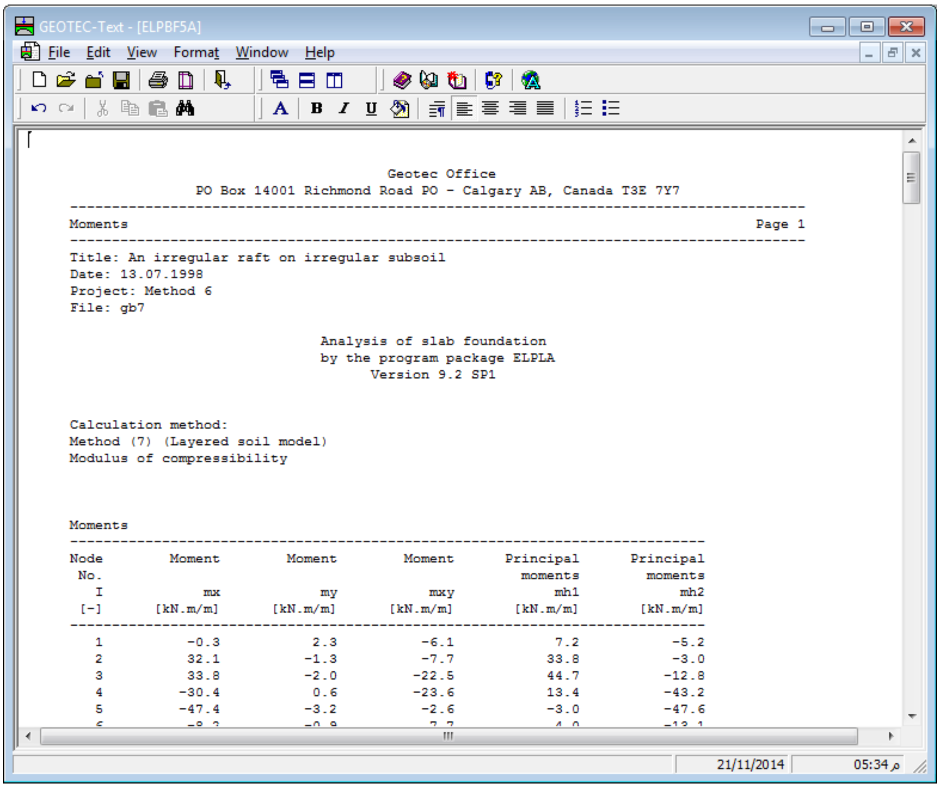


Figure G-18 Moments through GEOTEC-Editor

7 Format Menu

The Format menu has the following commands:

- Page format
- Font

7.1 Format Menu –"Page format" command

By "Page format" command the page margins, number of lines per page and number of characters per line can be defined, Figure G-19. The user has the possibility at any time to specify one of the following left, top or bottom boundaries.

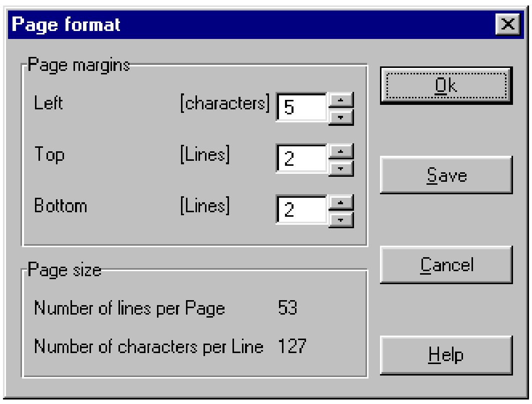


Figure G-19 "Page format" Dialog box

7.2 Format Menu –"Font" command

By this command font size (Figure G-20) and font type (Figure G-21) for the text can be defined.

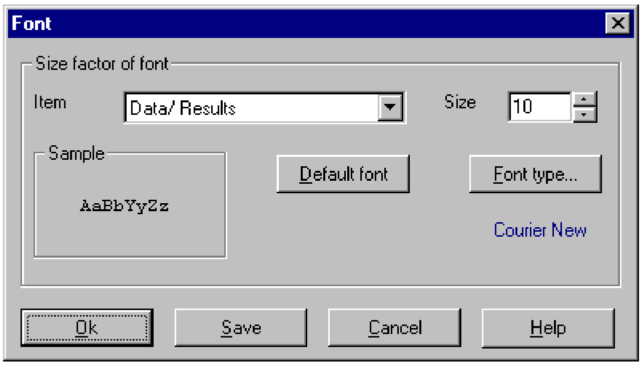


Figure G-20 "Font size" Dialog box

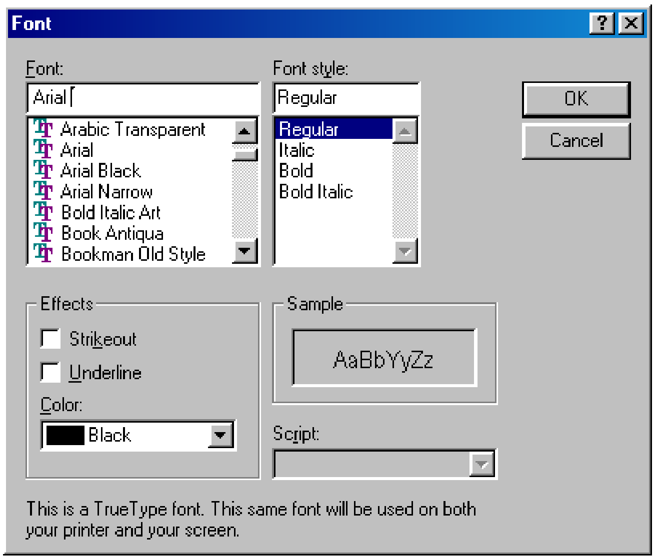


Figure G-21 "Font type" Dialog box

8 Window Menu

The Window menu has the following commands:

- Cascade
- Tile horizontally
- Tile vertically
- Arrange icons
- Window 1, 2, 3, …

8.1 Window Menu–"Cascade" command

By "Cascade" command all non-minimized forms are cascaded.

8.2 Window Menu–"Tile horizontally" command

By "Tile horizontally command all non-minimized forms are tiled horizontally.

8.3 Window Menu–"Tile vertically" command

By "Tile vertically" command all non-minimized forms are tiled vertically.

8.4 Window Menu–"Arrange icons" command

When choosing "Arrange icons" command, the icons of minimized forms are arranged.

8.5 Window Menu–"Window 1, 2, 3 …" command

By "Window 1, 2, 3 …" command the user can display a list of the previous loaded data or results.

9 Help Menu

The Help Menu commands are:

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

9.1 Help Menu–"Contents" command

"Contents" command displays a help file in HTML-Format containing the complete User’s Guide of ELPLA, Figure G-22.

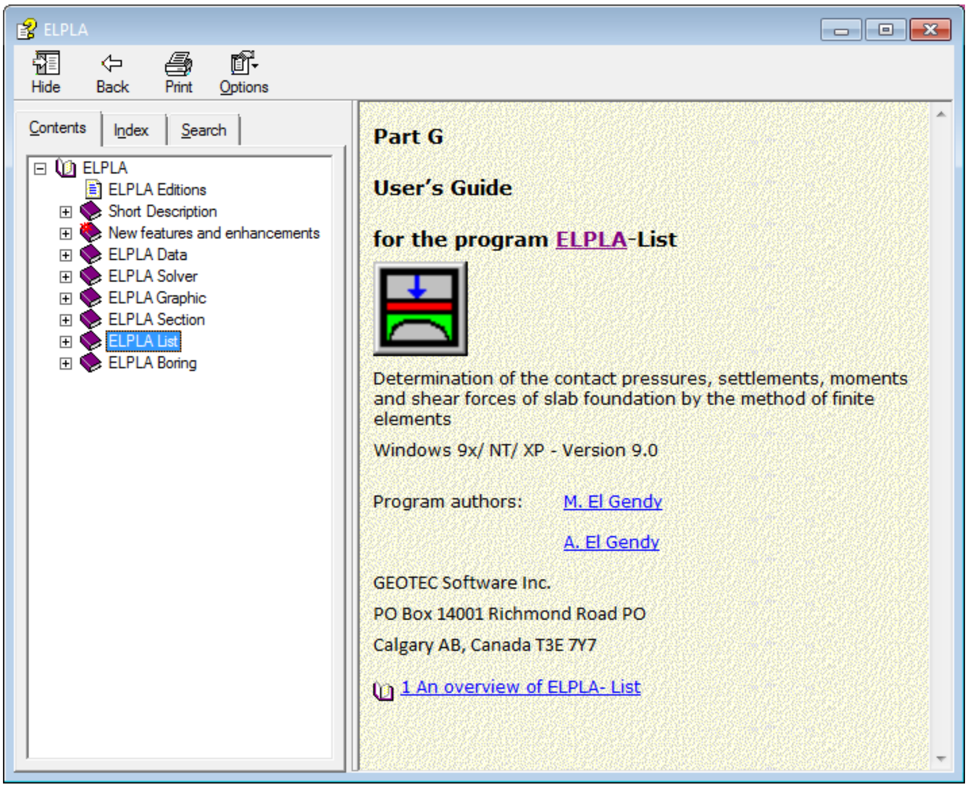


Figure G-22 Menu "Contents"

9.2 Help Menu–"Short description of ELPLA" command

"Short description of ELPLA" command gives a short description of ELPLA package.

9.3 Help Menu–"New in ELPLA" command

"New in ELPLA" command summarizes the new features and enhancements in ELPLA.

9.4 Help Menu–"About ELPLA-List" command

Clicking the command displays the information form of ELPLA-List as shown in Figure G-23, which gives information about ELPLA-List and the calculation method of the loaded project.

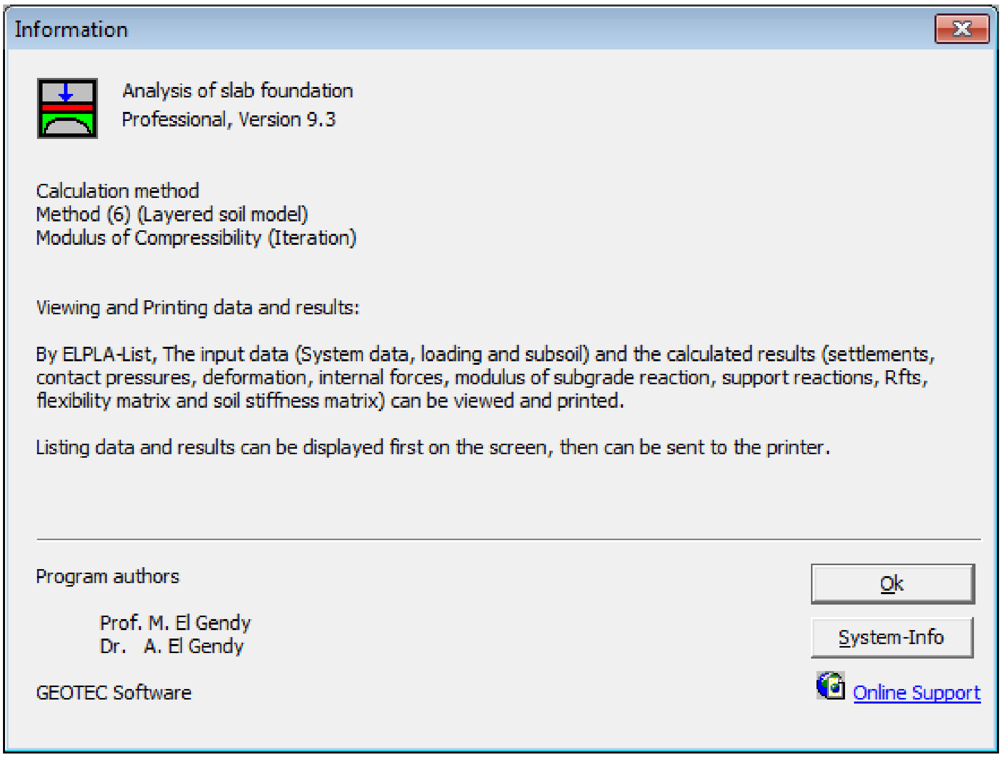


Figure G-23 Information form of ELPLA-List

10 Tips and Tricks

10.1 Keyboard

The user can obtain all menu titles and commands also through Shortcut keys. The action of the Shortcut keys is listed in Table G-4 to Table G-10.

Table G-4 Shortcut keys of menu head

Shortcut keys	Action
[Alt+f]	Calling menu head "File"
[Alt+v]	"View"
[Alt+l]	"List"
[Alt+t]	"Format"
[Alt+w]	"Window"
[Alt+h]	"Help"

Table G-5 Shortcut keys of File-Command

Shortcut keys	Action	
[Ctrl +o] or [Alt+f] then [o]	Calling command	"Open"
[Alt+f] then [c]		"Close project"
[Alt+f] then [u]		"Print setup"
[Alt+f] then [w]		"Send to Word"
[Alt+f] then [e]		"Send to Excel"
[Alt+f] then [1]	Calling the first project from the last four loaded projects	
[Alt+f] then [2]	Calling the second project from the last four loaded projects	
[Alt+f] then [3]	Calling the third project from the last four loaded projects	
[Alt+f] then [4]	Calling the fourth project from the last four loaded projects	
[Ctrl+q] or [Alt+f] then [x]	Calling command "Exit"	

Table G-6 Shortcut keys of View-Command

Shortcut keys	Action	
[Alt+v] then [x]	Calling command	"Project explorer"
[Alt+v] then [b]		"Status bar"
[Alt+v] then [t]		"Tool bars"
[Alt+v] then [t], then [f]		"Tool bars-File"
[Alt+v] then [t], then [l]		"Tool bars-List
[Alt+v] then [t], then [t]		"Tool bars-Format"
[Alt+v] then [t], then [w]		"Tool bars-Window"
[Alt+v] then [t], then [h]		"Tool bars-Help"
[Alt+v] then [t], then [r]		"Reset Toolbar"

Table G-7 Shortcut keys of List-Command

Shortcut keys	Action	
[Alt+l] then [t]	Calling command	"Display tables of data"
[Alt+l] then [a]		"Print tables of data"
[Alt+l] then [b]		"List tables of data through Text-Editor"
[Alt+l] then [e]		"Display tables of results"
[Alt+l] then [p]		"Print tables of results"
[Alt+l] then [l]		"List tables of results through Text-Editor"

Table G-8 Shortcut keys of Format-Command

Shortcut keys	Action	
[Alt+t] then [p]	Calling command	"Page format"
[Alt+t] then [f]		"Font"

Table G-9 Shortcut keys of Window-Command

Shortcut keys	Action	
[Alt+w] then [c]	Calling command	"Cascade"
[Alt+w] then [h]		"Tile horizontally"
[Alt+w] then [v]		"Tile vertically"
[Alt+w] then [a]		"Arrange icons"
[Alt+w] then [1, 2, 3, …]		"Window 1, 2, 3 …"

Table G-10 Shortcut keys of Help-Command

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

10.2 Mouse

By clicking the right mouse Button on the screen, the user can obtain the Popup-Format-Menu, Figure G-24.

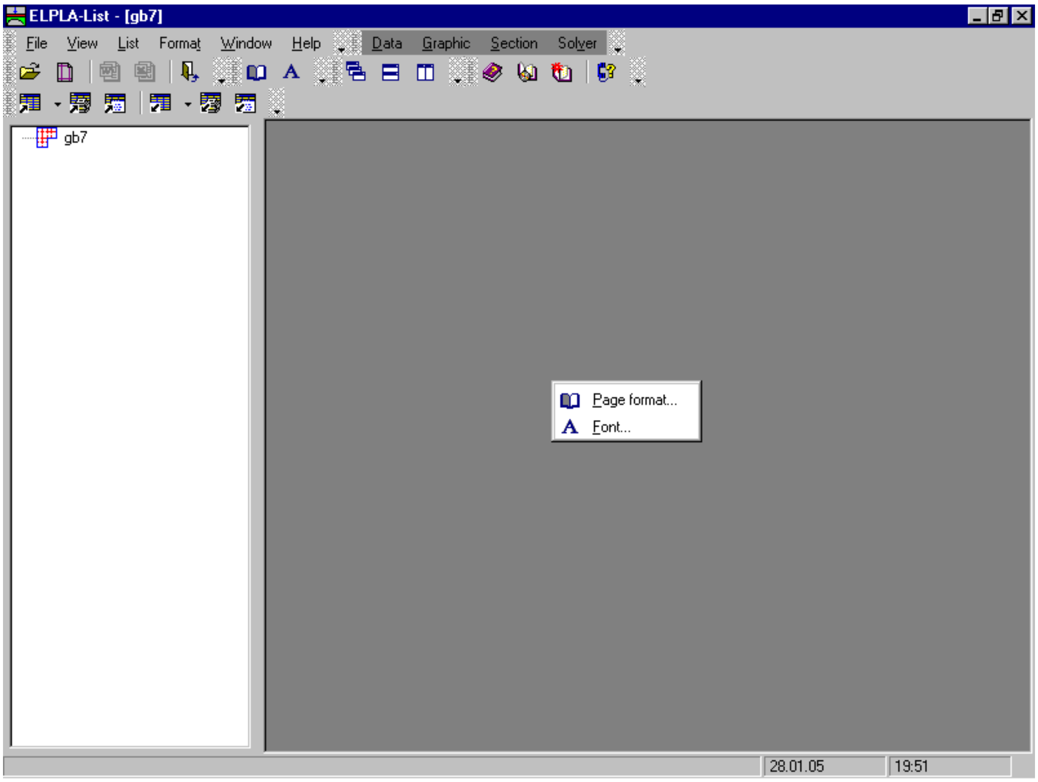


Figure G-24 Menu "Popup-Format"

11 Listing samples for data and results using ELPLA-List

ELPLA-List gives the ability to print data and results in arranged tables. In this paragraph some of the printed text for data and results of example problems gb7, an irregular raft on irregular subsoil, are presented.

The pages 1 to 2 contain some printed text of input project data for the example problem gb7, while pages 3 to 7 contain some printed text of results for the same example problem, according to Table G-11 and Table G-12.

11.1 Listing input project data

Table G-11 Input project data

Presentation	Page
Title page of program ELPLA	-
Loading	1 – 2

11.2 Listing of calculated results

Table G-12 Results

Presentation	Page
Moments m	3 – 7

```
**************************************************************************************
                              Analysis of slab foundation
                              by the program package ELPLA
                                       Version 9.0
**************************************************************************************
```

```
Title         : An irregular raft on irregular subsoil
Date          : 13.07.1998
Project       : Method 7
File          : gb7
```

```
Calculation method:
Method (7) (Layered soil model)
Modulus of Compressibility (Elimination)
```

Program authors: M. El Gendy/ A. El Gendy
GEOTEC: PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

GEOTEC-Software
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Loading

Title : An irregular raft on irregular subsoil
Date : 13.07.1998
Project : Method 7
File : gb7

Analysis of slab foundation
by the program package ELPLA
Version 9.0

Calculation method:
Method (7) (Layered soil model)
Modulus of Compressibility (Elimination)

L o a d i n g
Point loads:

Load No. I [1]	Load value P [kN]	x-position x [m]	y-position y [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	5	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:

Moment No. [1]	Moment value Mx [kN.m]	x-position x [m]	y-position y [m]
1	350	5	1,4

Moments My:

Moment No. [1]	Moment value My [kN.m]	x-position x [m]	y-position y [m]
1	500	9,2	5,5

Line loads:

Load No. I [1]	Load value Pl [kN/m]	Load start x1 [m]	Load start y1 [m]	Load end x2 [m]	Load end y2 [m]
1	89	10,5	4,8	15	2,8

Distributed loads:

Load No. I [1]	Load value P [kN/m2]	Load start x1 [m]	Load start y1 [m]	Load end x2 [m]	Load end y2 [m]
1	120	0	0	0,5	14

Loading data:

Slab weight	Pe	[kN]	= 0,0
Force on slab	Pa	[kN]	= 17927,1
Groundwater force	Pw	[kN]	= 2031,2
Total load (P = Pe + Pa - Pw)	P	[kN]	= 15895,9
Groundwater pressure on slab	Qw	[kN/m2]	= 12,0
Average contact pressure	Qo	[kN/m2]	= 93,9
Sum Mx from loads	Mx	[kN.m]	= 7039,1
Sum My from loads	My	[kN.m]	= -6683,2
Eccentricity of loading in x-direction	ex	[cm]	= -42,04
Eccentricity of loading in y-direction	ey	[cm]	= 44,28
Moment of inertia of slab about x-Axis	Ix	[m4]	= 2923,08
Moment of inertia of slab about y-Axis	Iy	[m4]	= 3423,36
Product of inertia	Ixy	[m4]	= 366,19
Area of slab	A	[m2]	= 169,27

Title : An irregular raft on irregular subsoil
Date : 13.07.1998
Project : Method 7
File : gb7

Analysis of slab foundation
by the program package ELPLA
Version 9.0

Calculation method:
Method (7) (Layered soil model)
Modulus of Compressibility (Elimination)

Moments:

Node No. I [1]	Moment mx [kN.m/m]	Moment my [kN.m/m]	Moment mxy [kN.m/m]	Principal moment mh1 [kN.m/m]	Principal moment mh2 [kN.m/m]
1	-0,2	2,4	-5,3	6,6	-4,4
2	32,6	-1,4	-7,1	34,1	-2,8
3	34,9	-1,9	-22,2	45,4	-12,3
4	-29,2	0,6	-23,4	13,4	-42,0
5	-46,1	-3,2	-2,3	-3,1	-46,2
6	-6,9	-0,9	7,9	4,6	-12,3
7	-17,3	4,0	0,6	4,0	-17,3
8	-87,9	-5,4	5,2	-5,1	-88,2
9	-154,1	-0,3	25,1	3,7	-158,1
10	-102,8	-1,3	33,2	8,6	-112,7
11	-25,6	-4,0	24,3	11,8	-41,4
12	5,7	7,3	13,7	20,3	-7,2
13	-0,7	31,9	-13,0	36,4	-5,2
14	61,6	77,6	-16,2	87,7	51,5
15	103,4	106,5	-26,4	131,4	78,6
16	-50,1	58,9	-26,7	65,1	-56,3
17	-67,6	63,3	-4,8	63,5	-67,7
18	30,5	126,5	9,7	127,4	29,5
19	58,0	151,8	-6,7	152,3	57,5
20	-115,9	64,8	-0,8	64,8	-115,9
21	-166,9	32,6	30,4	37,1	-171,4
22	-99,7	41,4	40,6	52,3	-110,5
23	22,5	61,3	39,1	85,5	-1,7
24	-6,4	-0,9	33,6	30,1	-37,4
25	0,1	14,7	-31,6	39,8	-25,1
26	61,2	57,5	-36,4	95,8	22,9
27	99,5	88,5	-30,6	125,1	62,9
28	-56,6	44,4	-31,0	53,1	-65,3
29	-81,6	70,5	-34,8	78,0	-89,2
30	20,9	147,6	-19,7	150,6	17,9
31	50,1	170,4	2,4	170,5	50,0
32	-139,9	92,9	29,2	96,5	-143,5
33	-176,7	35,5	46,9	45,4	-186,6
34	-105,7	36,2	47,2	50,5	-120,0
35	4,9	51,6	61,0	93,5	-37,0
36	6,3	-18,5	90,6	85,4	-97,5
37	-1,3	-56,0	-36,0	16,6	-73,9
38	35,6	-64,0	-42,4	51,2	-79,7

Continue of table at next page

Continue of table

Node No. I [1]	Moment mx [kN.m/m]	Moment my [kN.m/m]	Moment mxy [kN.m/m]	Principal moment mh1 [kN.m/m]	Principal moment mh2 [kN.m/m]
39	27,7	-69,8	-41,5	43,0	-85,0
40	-39,2	-40,7	-52,4	12,5	-92,3
41	-82,9	-9,8	-73,5	35,7	-128,5
42	-64,9	3,3	-58,5	36,9	-98,6
43	-89,4	15,2	1,2	15,2	-89,4
44	-167,2	38,0	57,2	52,9	-182,1
45	-178,3	5,2	69,5	28,5	-201,7
46	-117,9	-19,5	64,7	12,6	-150,0
47	-52,1	-45,1	83,9	35,4	-132,6
48	-130,8	-48,6	115,7	33,1	-212,5
49	-19,8	10,5	40,8	38,8	-48,1
50	50,5	-1,8	17,9	56,0	-7,3
51	68,3	1,8	3,8	68,5	1,6
52	-0,6	-3,2	2,5	0,9	-4,7
53	-2,2	-76,6	-27,6	6,9	-85,7
54	37,4	-87,1	-32,0	45,1	-94,9
55	30,6	-97,5	-50,0	47,8	-114,7
56	-21,9	-92,4	-75,8	26,5	-140,8
57	-93,2	-61,4	-100,7	24,7	-179,3
58	-114,0	-41,7	-74,2	4,7	-160,4
59	-180,4	-10,8	-25,6	-7,0	-184,2
60	-224,0	9,9	78,8	33,9	-248,1
61	-166,6	-18,4	84,5	19,8	-204,9
62	-123,3	-35,3	80,9	12,7	-171,4
63	-72,8	-38,4	84,3	30,4	-141,7
64	-83,3	-4,7	81,3	46,3	-134,3
65	-16,3	17,9	43,1	47,2	-45,6
66	77,1	32,4	24,4	87,9	21,7
67	86,3	32,0	-0,2	86,3	32,0
68	-2,2	23,0	-3,8	23,6	-2,7
69	-1,5	-32,6	-13,5	3,6	-37,7
70	60,2	-53,2	-15,6	62,3	-55,3
71	67,6	-82,3	-47,4	81,3	-96,0
72	6,8	-94,7	-84,7	54,8	-142,7
73	-91,3	-176,1	-134,9	7,7	-275,1
74	-151,5	24,1	-110,3	77,3	-204,7
75	-185,9	4,7	-48,1	16,1	-197,4
76	-335,8	17,1	140,5	66,2	-384,9
77	-154,3	-35,2	96,2	18,3	-207,9
78	-129,0	-23,6	91,0	28,9	-181,5
79	-57,5	-12,2	95,3	63,1	-132,8
80	-39,9	41,2	49,2	64,4	-63,1
81	4,5	88,3	44,2	107,3	-14,5
82	117,1	90,3	31,7	138,1	69,3
83	123,2	83,1	-7,1	124,4	81,9
84	1,4	102,7	-18,2	105,9	-1,7
85	0,0	51,8	-7,0	52,7	-0,9
86	111,9	100,6	-7,4	115,5	96,9
87	195,3	116,3	-22,2	201,1	110,5
88	-3,6	-24,0	-30,2	18,0	-45,7
89	15,1	-73,6	-33,8	26,5	-85,0
90	16,3	-82,6	157,7	132,1	-198,4
91	-139,6	-23,5	90,4	25,8	-189,0
92	-101,0	10,0	77,9	50,1	-141,2

Continue of table at next page

Continue of table

Node No. I [1]	Moment mx [kN.m/m]	Moment my [kN.m/m]	Moment mxy [kN.m/m]	Principal moment mh1 [kN.m/m]	Principal moment mh2 [kN.m/m]
93	12,0	98,4	71,8	139,0	-28,7
94	-40,8	99,1	30,7	105,6	-47,2
95	-4,7	132,9	22,7	136,6	-8,4
96	200,0	274,1	14,0	276,6	197,5
97	237,9	320,7	-8,6	321,6	237,0
98	-3,3	204,9	-17,4	206,4	-4,8
99	-0,3	39,2	-9,9	41,6	-2,6
100	103,2	66,1	-10,1	105,7	63,5
101	167,2	68,9	9,8	168,2	67,9
102	14,1	-15,2	18,8	23,3	-24,4
103	-0,9	-58,0	13,0	1,9	-60,8
104	-10,6	-69,4	63,2	29,8	-109,7
105	-105,4	-23,4	61,8	9,7	-138,6
106	-97,2	14,8	47,7	32,3	-114,8
107	23,4	67,9	24,3	78,5	12,7
108	-8,5	68,3	22,3	74,3	-14,5
109	8,6	104,2	-14,9	106,4	6,3
110	185,7	210,6	-25,3	226,4	169,9
111	214,3	248,2	-7,0	249,6	212,9
112	-1,4	182,6	-2,3	182,7	-1,5
113	-1,6	-46,3	1,8	-1,5	-46,4
114	59,0	-62,1	2,3	59,1	-62,2
115	68,3	-81,1	23,3	71,9	-84,6
116	25,5	-76,4	34,5	36,1	-87,0
117	-1,8	-84,4	28,9	7,3	-93,5
118	7,4	12,9	33,2	43,4	-23,2
119	-113,5	-7,6	42,9	7,6	-128,7
120	-109,1	-3,7	40,0	9,7	-122,6
121	-9,0	0,7	22,0	18,4	-26,7
122	53,6	-25,8	0,0	53,6	-25,8
123	29,6	21,3	-34,0	59,7	-8,8
124	115,6	26,9	-39,9	130,9	11,6
125	119,8	33,6	-18,4	123,6	29,8
126	0,2	56,1	-15,4	60,0	-3,7
127	-1,7	-68,3	19,3	3,4	-73,5
128	45,1	-78,6	23,1	49,2	-82,8
129	47,8	-86,3	22,7	51,5	-90,0
130	23,5	-83,5	26,1	29,5	-89,5
131	-4,9	-83,4	33,1	7,2	-95,5
132	-9,1	-39,1	-45,2	23,5	-71,7
133	33,2	-33,2	-35,9	49,0	-48,9
134	92,0	-31,6	-33,3	100,3	-40,0
135	92,5	-26,5	-35,6	102,3	-36,4
136	-2,3	-23,9	-38,9	27,2	-53,4
137	-1,0	26,7	36,0	51,4	-25,7
138	75,1	18,4	41,1	96,7	-3,1
139	95,2	1,7	21,3	99,8	-2,9
140	16,2	9,1	18,6	31,6	-6,3
141	8,0	-17,9	52,4	49,0	-58,9
142	7,8	-36,7	-69,0	58,1	-86,9
143	8,8	-2,3	-36,5	40,2	-33,6
144	117,5	19,7	-29,4	125,7	11,5
145	124,8	25,8	-49,4	145,3	5,3
146	0,1	34,7	-57,2	77,1	-42,3

Continue of table at next page

Continue of table

Node No. I [1]	Moment mx [kN.m/m]	Moment my [kN.m/m]	Moment mxy [kN.m/m]	Principal moment mh1 [kN.m/m]	Principal moment mh2 [kN.m/m]
147	0,3	124,8	32,2	132,6	-7,5
148	122,4	204,8	36,2	218,4	108,8
149	221,6	252,2	37,5	277,4	196,4
150	-10,1	95,1	41,2	109,3	-24,3
151	-106,0	47,9	54,2	65,1	-123,1
152	-93,4	0,2	11,4	1,6	-94,7
153	-103,5	2,7	0,8	2,7	-103,5
154	-124,9	-2,4	-11,9	-1,2	-126,1
155	-136,1	-2,6	-16,8	-0,6	-138,1
156	-110,0	-1,9	-26,1	4,1	-115,9
157	-90,7	9,3	-55,7	34,2	-115,6
158	-113,7	-35,6	-106,2	38,5	-187,8
159	-13,0	36,3	-71,6	87,4	-64,1
160	171,0	187,5	-54,6	234,5	124,1
161	209,9	232,5	-49,5	272,0	170,5
162	-5,5	97,8	-46,4	115,6	-23,3
163	-0,6	109,2	34,2	119,0	-10,4
164	92,4	86,2	36,2	125,7	52,9
165	106,8	59,9	50,0	138,6	28,1
166	12,1	85,5	62,6	121,3	-23,8
167	-65,9	69,8	62,3	94,1	-90,2
168	-67,1	38,8	32,8	48,1	-76,5
169	-66,2	20,9	-1,5	20,9	-66,2
170	-108,9	30,6	-19,6	33,3	-111,6
171	-131,4	31,9	-13,7	33,0	-132,5
172	-81,3	25,3	-17,2	28,0	-84,0
173	-48,6	13,2	-55,7	46,0	-81,4
174	-74,9	43,8	-102,0	102,4	-133,6
175	-18,6	54,4	-93,4	118,2	-82,4
176	107,2	50,3	-73,3	157,4	0,1
177	128,7	49,3	-50,2	153,1	25,0
178	2,2	79,0	-41,8	97,3	-16,1
179	-0,3	103,2	48,5	122,3	-19,4
180	96,8	130,9	47,9	164,7	63,0
181	138,9	147,8	44,8	188,3	98,3
182	-11,1	113,3	48,7	130,1	-27,8
183	-54,5	116,1	57,0	133,4	-71,7
184	19,0	150,3	41,5	162,3	7,0
185	40,5	154,1	2,2	154,1	40,5
186	-102,9	101,0	-18,5	102,7	-104,6
187	-137,1	81,4	-11,7	82,0	-137,7
188	-42,6	116,6	-14,7	117,9	-43,9
189	84,4	156,3	-56,5	187,4	53,4
190	-63,0	94,5	-90,8	135,9	-104,4
191	-43,3	71,2	-81,4	113,4	-85,6
192	98,4	93,6	-63,9	159,9	32,0
193	134,4	95,7	-55,0	173,3	56,8
194	-1,2	53,0	-55,2	87,4	-35,6
195	-1,3	77,6	32,3	89,1	-12,8
196	94,6	145,7	32,9	161,8	78,6
197	156,7	182,1	44,8	216,0	122,9
198	-28,2	106,8	49,4	123,0	-44,3
199	-53,0	108,6	38,4	117,2	-61,7
200	61,0	179,0	24,6	183,9	56,1

Continue of table at next page

GEOTEC-Software
PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Continue of table

Node No. I [1]	Moment mx [kN.m/m]	Moment my [kN.m/m]	Moment mxy [kN.m/m]	Principal moment mh1 [kN.m/m]	Principal moment mh2 [kN.m/m]
201	100,7	201,7	15,4	204,0	98,3
202	-98,6	106,3	3,3	106,4	-98,7
203	-136,6	79,1	-14,6	80,1	-137,6
204	-19,9	137,7	-32,2	144,0	-26,2
205	156,1	212,4	-46,6	238,7	129,8
206	-60,7	90,9	-61,1	112,5	-82,2
207	-60,2	61,6	-71,4	94,5	-93,2
208	88,6	114,1	-63,2	165,8	36,9
209	134,9	123,9	-43,7	173,4	85,4
210	-5,2	33,1	-35,3	54,1	-26,2
211	-1,1	4,1	20,2	21,8	-18,8
212	41,6	-2,3	21,5	50,3	-11,1
213	51,5	-3,3	43,9	75,9	-27,6
214	-18,1	1,0	50,3	42,6	-59,7
215	-39,2	-0,1	27,6	14,2	-53,5
216	20,1	-1,7	13,4	26,5	-8,1
217	16,9	-3,8	18,9	28,1	-15,0
218	-75,7	0,6	11,5	2,3	-77,4
219	-127,1	-1,1	-14,9	0,6	-128,8
220	-20,6	-0,1	-37,5	28,5	-49,2
221	59,9	-4,0	-39,9	79,1	-23,2
222	-33,3	0,6	-46,2	32,9	-65,5
223	-61,0	-0,6	-66,2	41,9	-103,6
224	34,1	-0,7	-61,2	80,3	-47,0
225	57,1	-3,5	-35,4	73,4	-19,7
226	5,4	4,0	-21,3	26,1	-16,6

12 Index

Part H

User's Guide
for the program ELPLA-Boring

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

Version 9.2

Program authors: M. El Gendy
A. El Gendy

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

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

Table of Contents Page

1 An overview of ELPLA-Boring

ELPLA-Boring is used to edit and display graphically the boring logs (boring layers, soil material, foundation level and water table). The drawing of the boring logs can be displayed first on the screen, and then can be sent to the printer or plotter. The program draws the soil layers by different symbols according to the German Standard DIN 4023.

ELPLA-Boring can also display the stress in soil under a specified point on the foundation with effective stress beside the corresponding boring log. The stress in soil is used to determine the limit depth of the soil layers.

The drawings, if desired, can be saved as WMF-format files, in which they can be exported to other Windows applications to prepare reports, slide presentations, or add further information.

2 Description of ELPLA-Boring

ELPLA-Boring is a 32-bit, graphical software product that operates under Microsoft Windows 9x/ NT/ ME/ XP. The common "what you see is what you get" of Windows applications makes it easy to learn how to use ELPLA-Boring, especially if you are already familiar with the Windows environment.

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

The usage of the program is typically such that first data files are created describing a certain problem by ELPLA-Data, then the project problem is analyzed using ELPLA-Solver. Finally, 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-Editor.

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

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

In order to use ELPLA-Boring, first the user must define the project data by ELPLA-Data, and then must analyze by ELPLA-Solver. Table H-2 gives a list of files, which are read or created by ELPLA-Boring. The files can be classified in four groups.

Table H-2 Names of file groups

Group	Saved from the program
A Main data files	ELPLA-Data
B Boring files (*.BOR)	ELPLA-Boring
C Project data files	ELPLA-Data
D Final result files	ELPLA-Solver

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

Table H-3 Names and contents of files

A Main data files

Filename	Contents
FIRMA	Firm header
STEU	Default directory for files that are saved by ELPLA
NOFORMAT	Number formats
UNITS	System of units

B Boring files

Filename	Contents
LINEFORM. BOR	Line formats
FONT. BOR	Font data
LEGENDE. BOR	Legend data
PAINT. BOR	Fill color data
PLOTPAR. BOR	Plot parameters
NODISPLA. BOR	Data of display values
ORDINATE. BOR	Max. width data

C Project data files

Filename	Contents
*. AUF	3 lines text to identify the project
*. P21	Data of slab properties/ levels/ coordinates
*. BAU	Soil properties
*. LDH	Data of the limit depth

D Final result files

Filename	Contents
*. LD1	Results of the limit depth

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

3 Starting ELPLA-Boring

Start ELPLA-Boring by clicking on the program icon. The introduction screen (Figure H-1) appears.

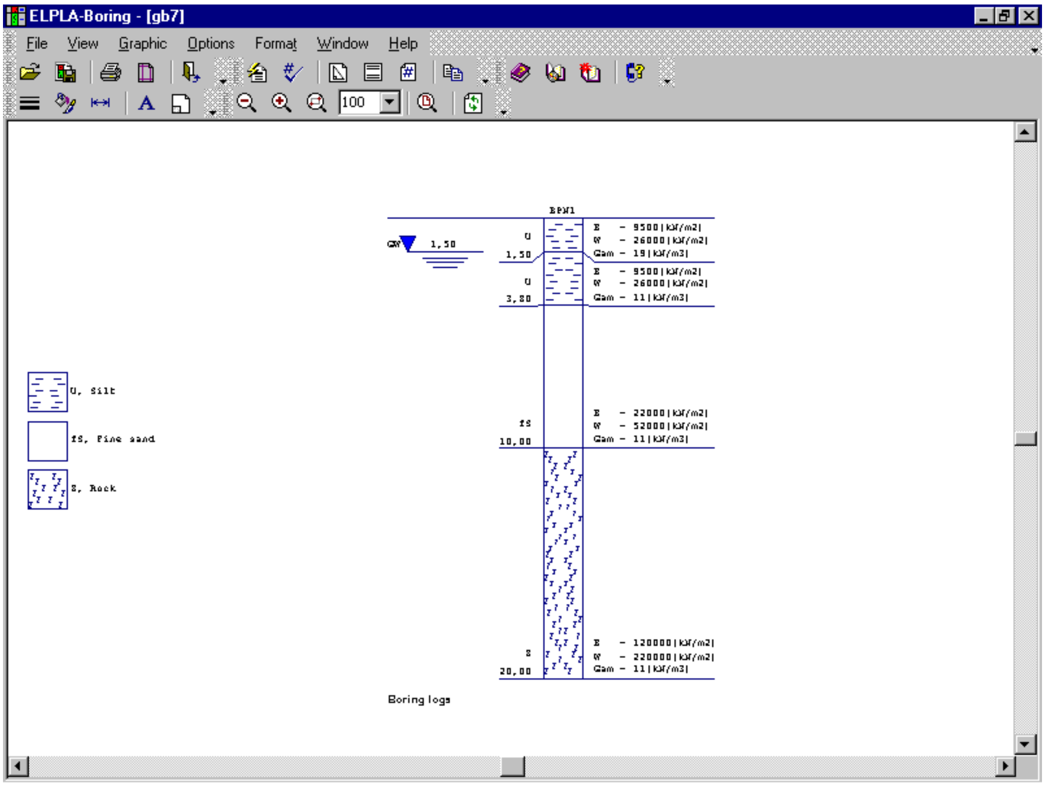


Figure H-1 Introduction screen of ELPLA-Boring

The menu head of Figure H-1 contains the following 7 commands:

- File
- View
- Graphic
- Options
- Format
- Window
- Help

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

4 File Menu

The File Menu commands are:

- Open
- Make WMF-File
- Print
- Page setup
- Files 1, 2, 3, 4
- Exit

4.1 File Menu–"Open" command

By "Open" command the current project is closed, if one is loaded, and an existing project is opened. Figure H-2 shows “Open” Dialog box used to open a specified project.

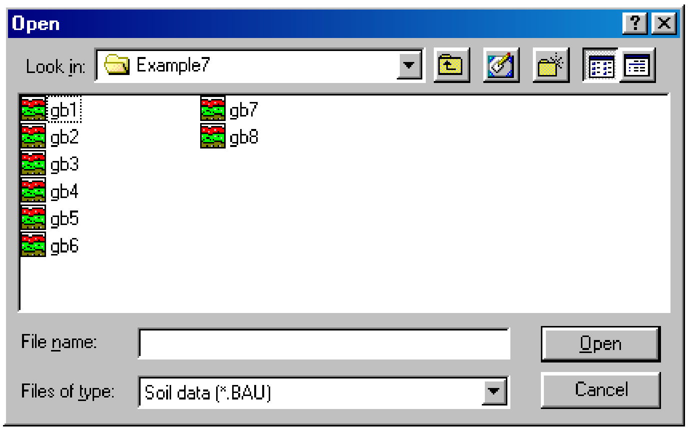


Figure H-2 "Open project" Dialog box

4.2 File Menu–"Make WMF-File" command

By "Make WMF-File" command the drawing can be saved in a format that can be read by other programs. This feature allows you to include your drawing in reports and presentations and to enhance the drawing using other drawing or CAD software packages. The drawing can be exported in the Windows Metafile (WMF) format.

Figure H-3 shows the “Save as” Dialog box used to export the drawing with the filename you wish to give the exported file, including extension and the directory in which to save the file. If the file name already exists, you may select to overwrite the existing file.

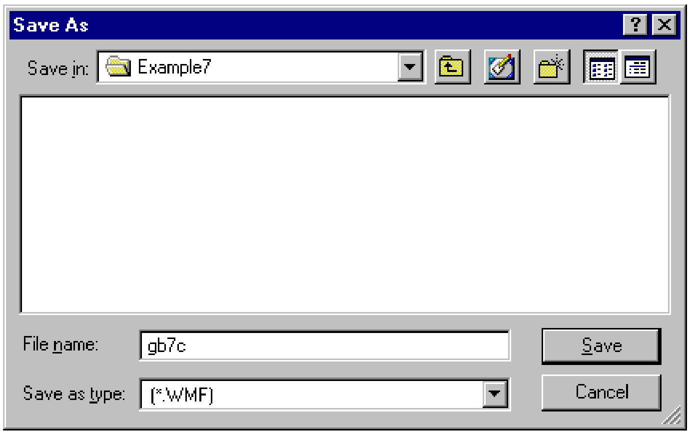


Figure H-3 "Save as" Dialog box

4.3 File Menu–"Print" command

By "Print" command data and results can be graphically plotted or printed. Only the objects currently displayed on the drawing are printed. Figure H-4 shows "Print" Dialog box. The printer group box contains controls for selecting the printer and changing its properties. Use "Name" Combo box to select the printer and use "Properties" Button to set printer settings. The number of printing copies can be defined in "Copies" Input box.

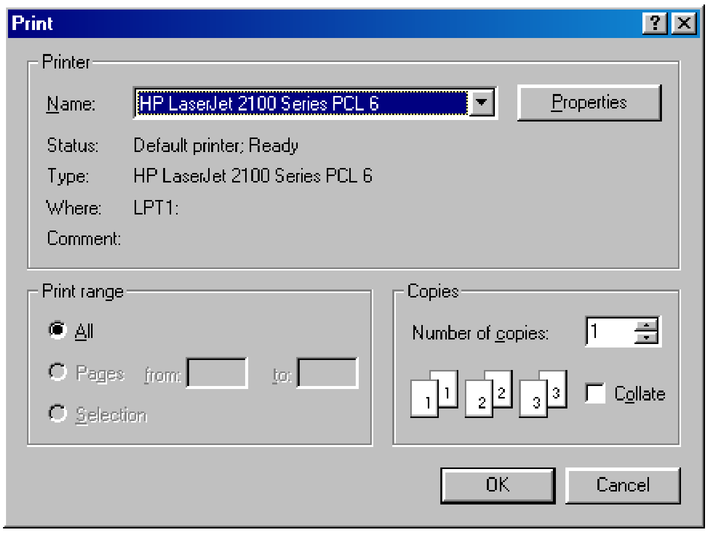


Figure H-4 "Print" Input box

4.4 File Menu–"Page setup" command

By "Page setup" command the standard "Page setup" Dialog box can be displayed with options to specify the printer, page orientation, paper size, and paper source, as well as other printing options (Figure H-5).

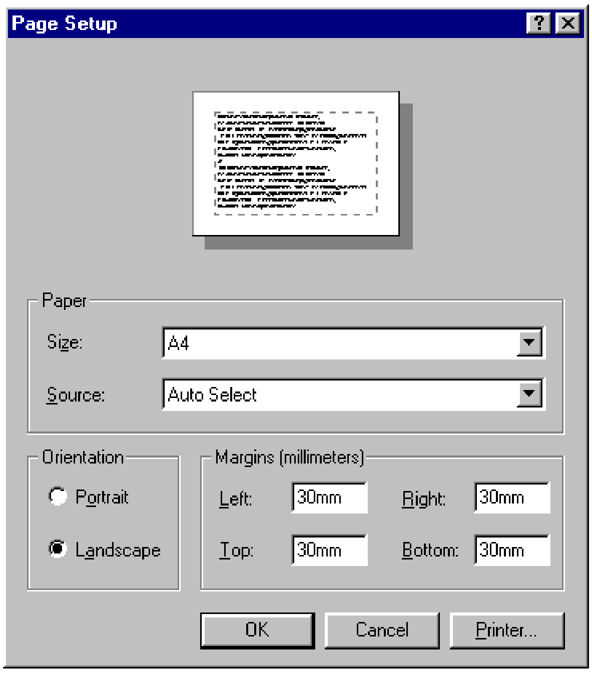


Figure H-5 "Page setup" Dialog box

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

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

4.6 File Menu–"Exit" command

By "Exit" command the current project is closed and ELPLA-Boring is quitted, Figure H-6.

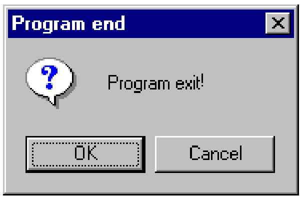


Figure H-6 "Exit" Message box

5 View Menu

The View Menu commands are:

- Status bar
- Tool bars

5.1 View Menu–"Status bar" command

"Status bar" command displays a status bar on the screen down. The status bar displays information about the progress of the current operation.

5.2 View Menu–"Tool bars" command

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

6 Graphic Menu

The graphic menu is the main menu, used to display the specified drawing. The Graphic Menu command is:

- Drawing boring logs/ Limit depth

6.1 Graphic Menu–"Boring logs/ Limit depth" command

By this command the boring logs (boring layers, soil material, foundation level and water table) can be displayed. The stress in soil under the foundation with effective stress can be displayed beside the corresponding boring log. The stress in soil is used to determine the limit depth of the soil layers.

When "Drawing boring logs/ Limit depth" command is chosen, the selection Dialog box shown in Figure H-7 appears. In this Dialog box select the boring logs or the limit depths to draw, then click "OK" Button. Figure H-8 shows boring logs, while Figure H-9 shows a limit depth of soil layers.

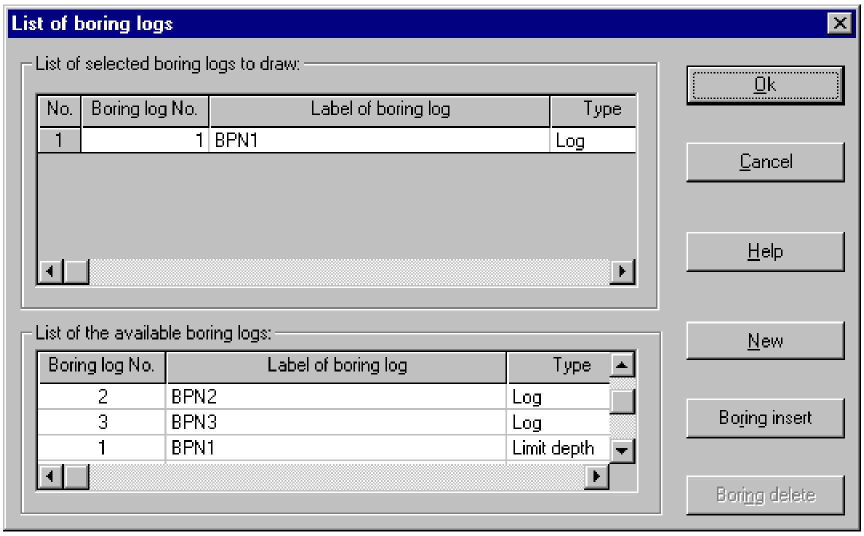


Figure H-7 "List of borings" Dialog box

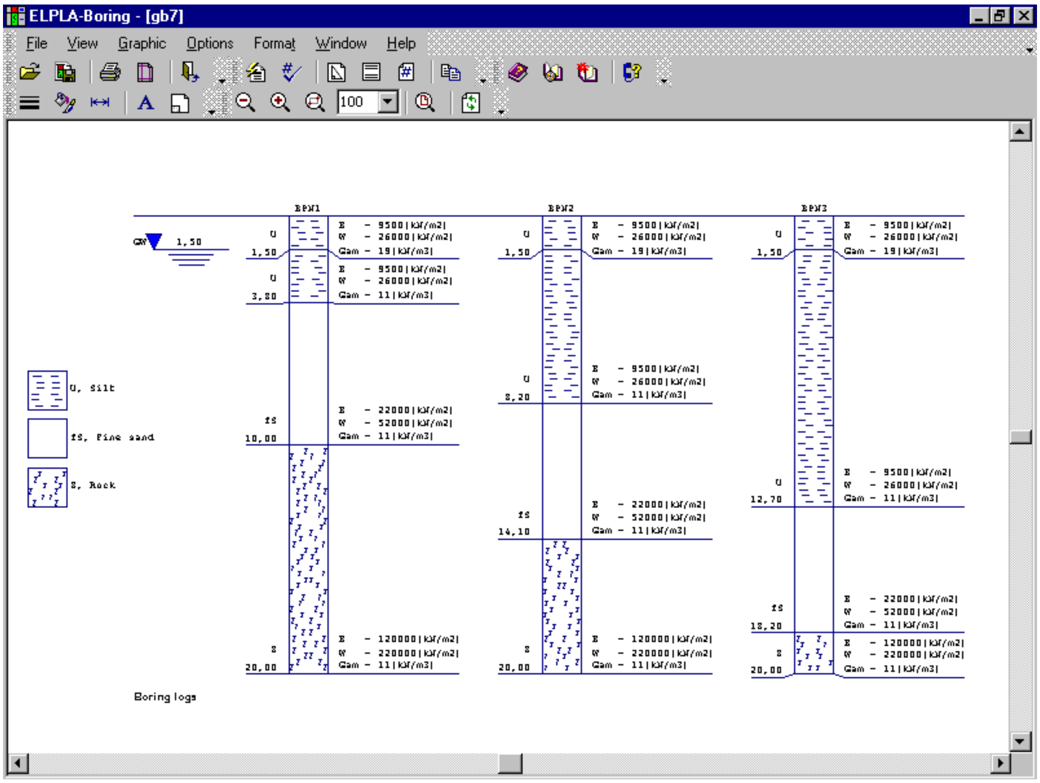


Figure H-8 Drawing of 3 borings with multi-layers and different soil material

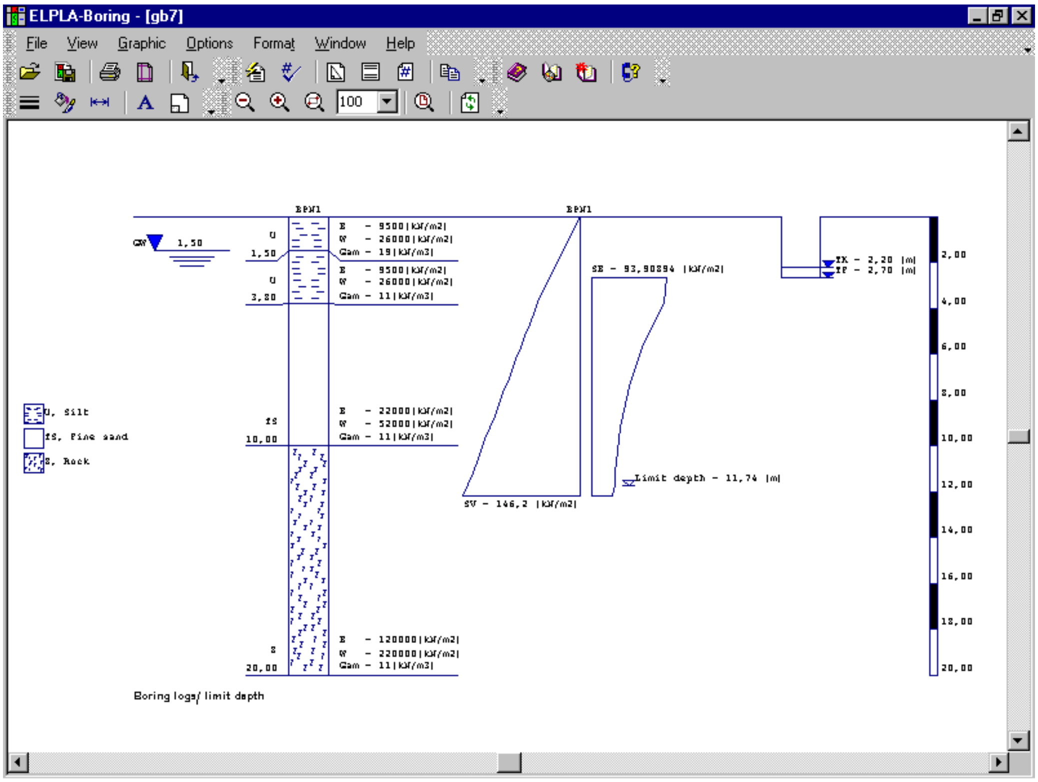


Figure H-9 Limit depth of the soil layers

7 Options Menu

The Options menu has the following commands:

- Plot parameters
- Display values
- Scale
- Title
- Page No.
- Copy

7.1 Options Menu–"Plot parameters" command

Figure H-10 shows the parameters, set as default values by the program, or specified by the user.

- Color soil layers
- Draw the water table
- Simple drawing of boring logs
- Setting soil colors according to DIN 4023
- Display soil properties C, Phi and Nue
- Color foundation
- Draw foundation
- Draw measurement bar
- Color limit depth
- Page with frame

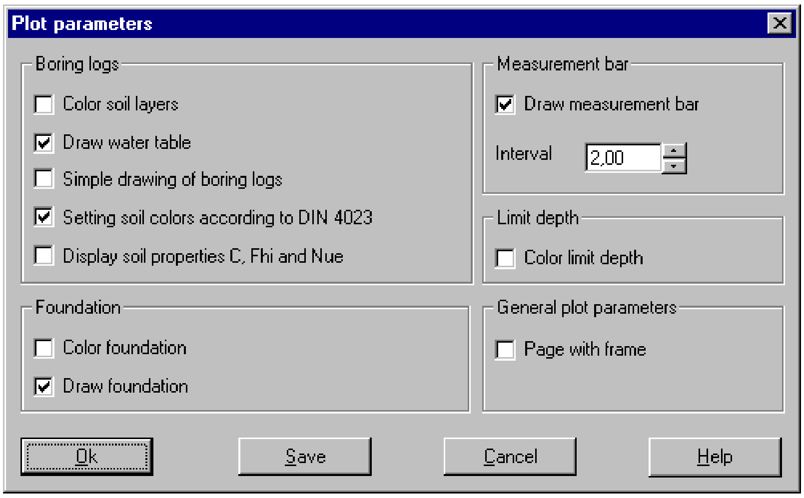


Figure H-10 "Plot parameters" Dialog box

7.2 Options Menu–"Display values" command

By "Display values" command the values of the following items can be displayed, if desired, on the drawing (Figure H-11):

- Label of the boring
- Layer description
- Layer depth
- Display text of soil symbols
- Measurement bar
- Foundation
- Water level
- Limit depth
- Stress value

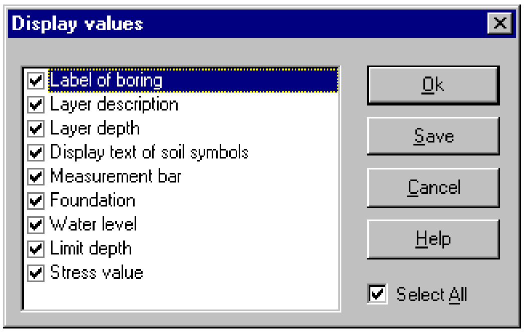


Figure H-11 "Display values" Dialog box

7.3 Options Menu–"Scale" command

By "Scale" command the scale of the drawing can be defined as shown in Figure H-12. The defaulted value for the scale factor is chosen to pass the active printer-paper format.

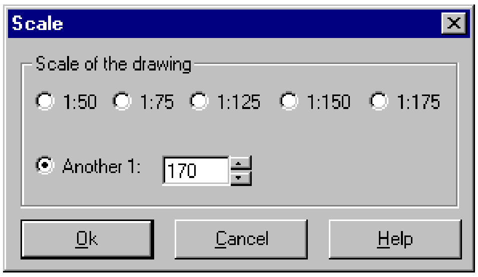


Figure H-12 "Scale" Dialog box

7.4 Options Menu–"Title" command

By "Title" command the text data (two line texts above the drawing and two line texts under the drawing) can be defined, Figure H-13.

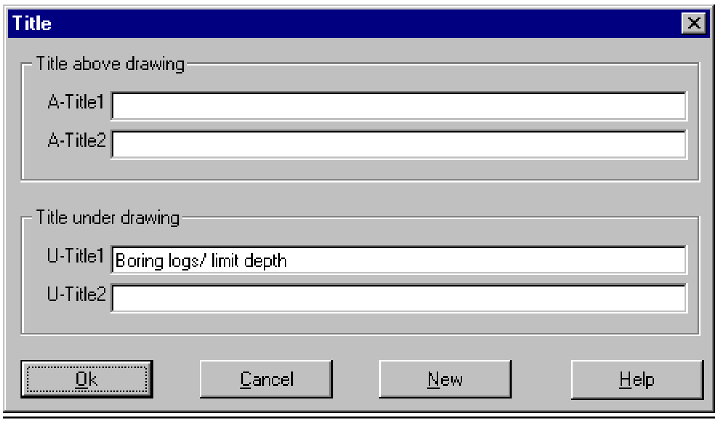


Figure H-13 "Title" Dialog box

7.5 Options Menu–"Page No." command

By "Page No." command the page No. can be defined, Figure H-14.

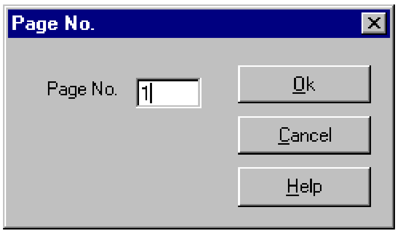


Figure H-14 "Page No. " Dialog box

7.6 Options Menu–"Copy" command

By "Copy" command the current drawing can be copied in Metafile-Format to Clipboard. Then it can be inserted directly to other Windows programs such as Word, WordPerfect and AutoCAD.

8 Format Menu

The Format menu has the following commands:

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

8.1 Format Menu–"Line formats" command

Color, style and thickness of drawing lines can be defined (Figure H-15). The way a line is drawn depends on the setting of the color and style properties. 15 different colors and 5 styles for line formats are available. The following list shows the available lines, which can be formatted:

- Identification box
- Page boundary
- Boring boundary
- Soil layer levels
- Soil symbols
- Groundwater
- Foundation
- Measurement bar
- Limit depth

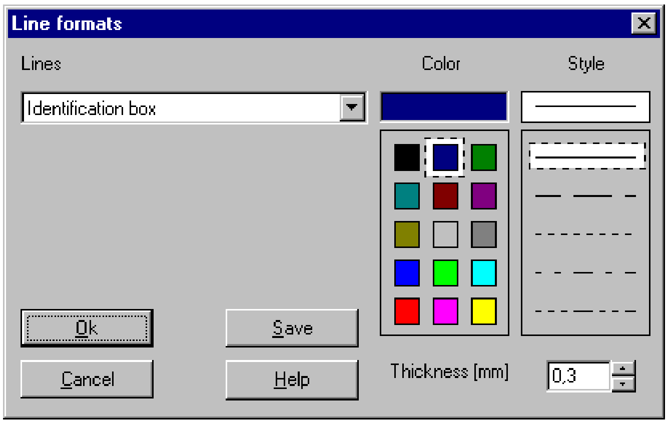


Figure H-15 "Line formats" Dialog box

8.2 Format Menu–"Fill color" command

The following list shows available items, which can be filled with a specified color, Figure H-16:

- Groundwater
- Foundation
- Measurement bar
- Stress due to foundation
- Stress from neighboring foundations
- Stress from soil weight

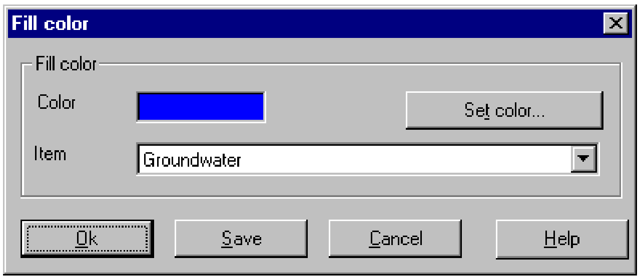


Figure H-16 "Fill color" Dialog box

8.3 Format Menu–"Max. width" command

By "Max. width" command the maximum width for the drawing can be defined, Figure H-17.

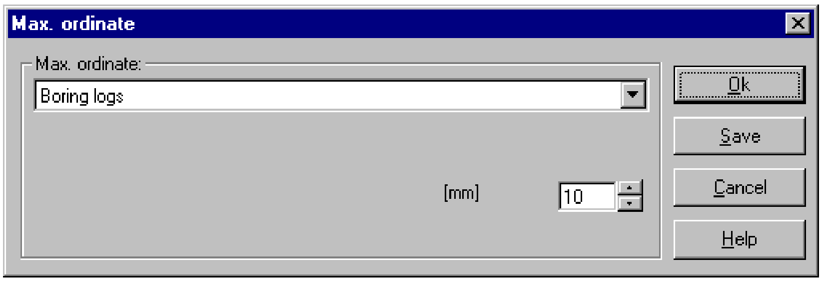


Figure H-17 "Max. width" Dialog box

8.4 Format Menu–"Font" command

By "Font" command the font size (Figure H-18) and font type (Figure H-19) for the drawing can be defined.

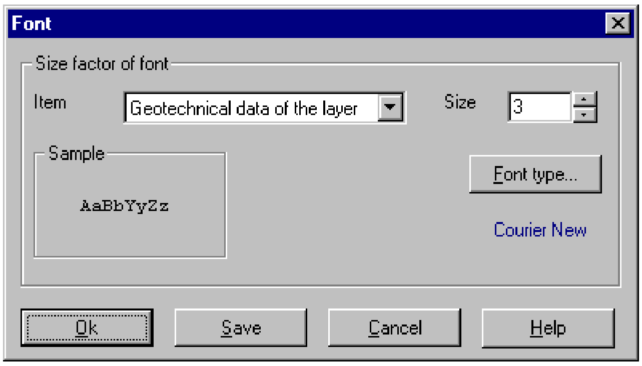


Figure H-18 "Font size" Dialog box

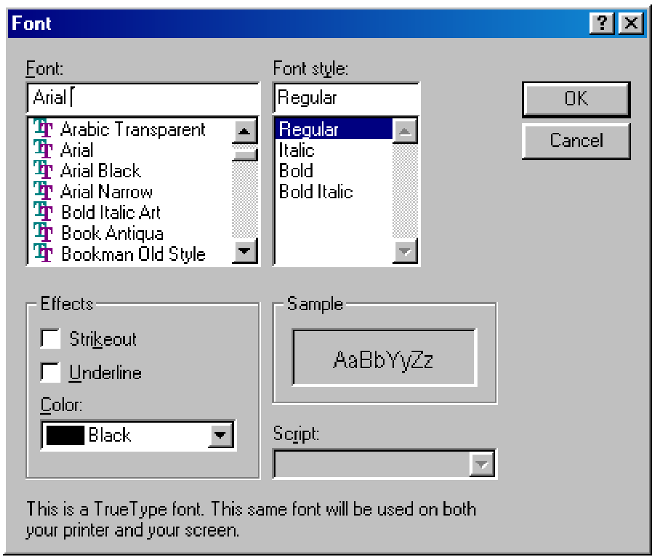


Figure H-19 "Font type" Dialog box

8.5 Format Menu–"Legend" command

By "Legend" command height and width of legends can be specified as shown in Figure H-20. Also the legend that you wish to display can be chosen.

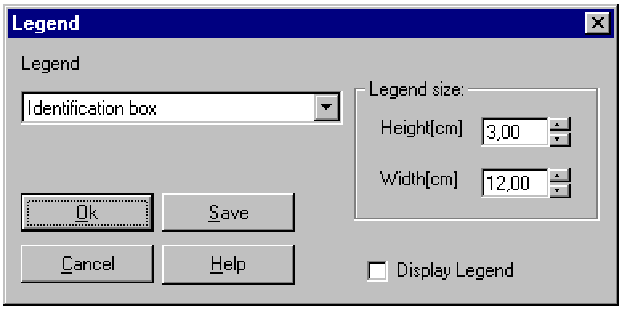


Figure H-20 "Legend" Dialog box

9 Window Menu

The Window menu has the following commands:

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

9.1 Window Menu–"Zoom in" command

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

9.2 Window Menu–"Zoom out" command

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

9.3 Window Menu–"Zoom window" command

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

9.4 Window Menu–"Zoom %" command

By "Zoom %" command the size of drawing on the screen can be specified (Figure H-21). Choosing "Zoom %" allows you to increase or decrease the size at which the drawing is displayed. Choosing 100% displays the drawing at its original size. Clicking on the percentage, changes the drawing size to the specified percentage. The drawing can be displayed at any size by typing the desired percentage in the specified Edit box.

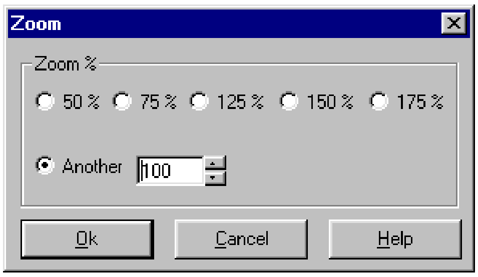


Figure H-21 "Zoom %" Dialog box

9.5 Window Menu–"Original size" command

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

10 Help Menu

The Help Menu commands are:

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

10.1 Help Menu–"Contents" command

"Contents" command displays a help file in HTML-Format containing the complete ELPLA User’s Guide, Figure H-22.

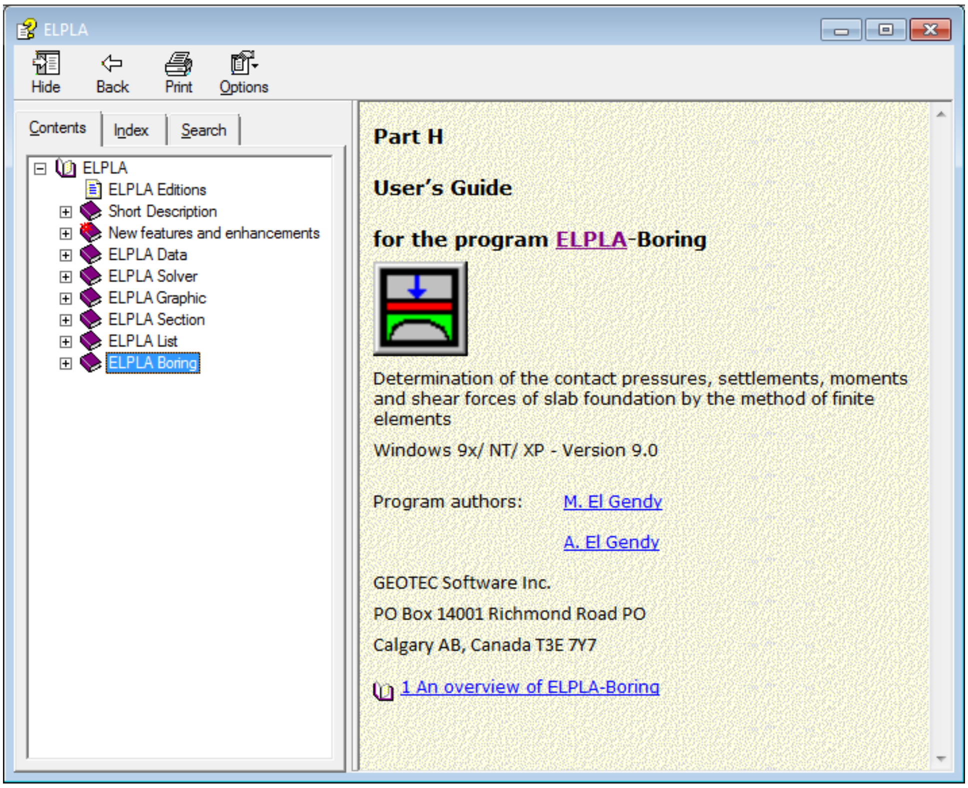


Figure H-22 Menu "Contents"

10.2 Help Menu–"Short description of ELPLA" command

"Short description of ELPLA" command gives a short description of ELPLA package.

10.3 Help Menu–"New in ELPLA" command

"New in ELPLA" command summarizes the new features and enhancements in ELPLA.

10.4 Help Menu–"About ELPLA-Boring" command

The command, as shown in Figure H-23, gives information about ELPLA-Boring.

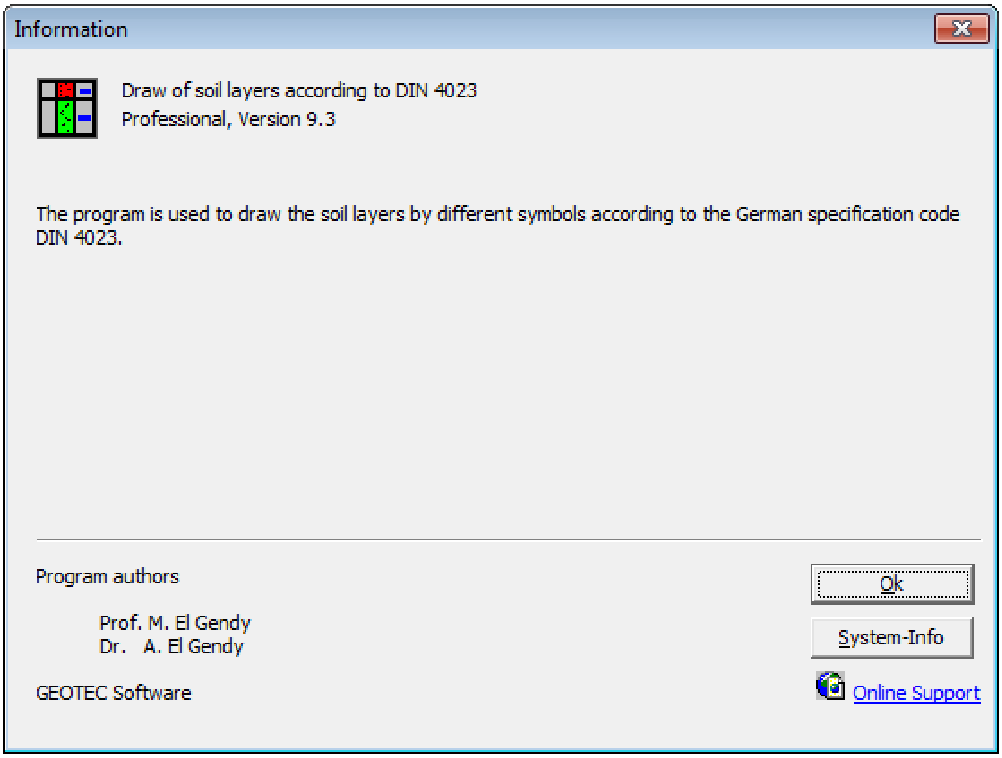


Figure H-23 Information form of ELPLA-Boring

11 Tips and Tricks

11.1 Keyboard

The user can obtain all menu titles and commands also through Shortcut keys. The action of the Shortcut keys is listed in Table H-4 to Table H-11:

Table H-4 Shortcut keys of menu head

Shortcut keys	Action
[Alt+f]	Calling menu head "File"
[Alt+v]	"View"
[Alt+g]	"Graphic"
[Alt+o]	"Options"
[Alt+t]	"Format"
[Alt+w]	"Window"
[Alt+h]	"Help"

Table H-5 Shortcut keys of File-Command

Shortcut keys	Action
[Ctrl +o] or [Alt+f] then [o]	Calling command "Open"
[Alt+f] then [w]	"Make WMF-File"
[Ctrl +p] or [Alt+f] then [p]	"Print"
[Alt+f] then [u]	"Page setup"
[Alt+f] then [1]	Calling the first project from the last four loaded projects
[Alt+f] then [2]	Calling the second project from the last four loaded projects
[Alt+f] then [3]	Calling the third project from the last four loaded projects
[Alt+f] then [4]	Calling the fourth project from the last four loaded projects
[Ctrl+q] or [Alt+f] then [x]	Calling command "Exit"

Table H-6 Shortcut keys of View-Command

Shortcut keys	Action
[Alt+v] then [b]	Calling command "Status bar"
[Alt+v] then [t]	"Tool bars"
[Alt+v] then [t], then [f]	"Tool bars-File"
[Alt+v] then [t], then [g]	"Tool bars-Graphic"
[Alt+v] then [t], then [o]	"Tool bars-Options"
[Alt+v] then [t], then [t]	"Tool bars-Format"
[Alt+v] then [t], then [w]	"Tool bars-Window"
[Alt+v] then [t], then [h]	"Tool bars-Help"
[Alt+v] then [t], then [r]	"Reset Toolbar"

Table H-7 Shortcut keys of Graphic-Command

Shortcut keys	Action
[Alt+g] then [o]	Calling command "Drawing boring logs/ Limit depth"

Table H-8 Shortcut keys of Options-Command

Shortcut keys	Action	
[Alt+o] then [l]	Calling command	"Plot parameters"
[Alt+o] then [d]		"Display values"
[Alt+o] then [s]		"Scale"
[Alt+o] then [t]		"Title"
[Alt+o] then [p]		"Page No."
[Alt+o] then [c]		"Copy"

Table H-9 Shortcut keys of Format-Command

Shortcut keys	Action	
[Alt+t] then [l]	Calling command	"Line formats"
[Alt+t] then [i]		"Fill color"
[Alt+t] then [x]		"Max. width"
[Alt+t] then [f]		"Font"
[Alt+t] then [d]		"Legend"

Table H-10 Shortcut keys of Window-Command

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

Table H-11 Shortcut keys of Help-Command

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

11.2 Mouse

By double-clicking the left mouse Button on a specified screen position, the user can obtain the almost menu of the program.

- By double-clicking on *legend*, *firm header*, *title* or *project identification*, the corresponding menu appears
- By double-clicking on *scale* in the identification box, "Scale"-Menu appears
- By double-clicking on *file name* in the identification box, "Open"-Menu appears
- By double-clicking on *page No.* in the identification box, "Page No."-Menu appears
- By clicking the right mouse Button at any position on the screen, the user can also obtain the "Popup-Options"-Menu, Figure H-24

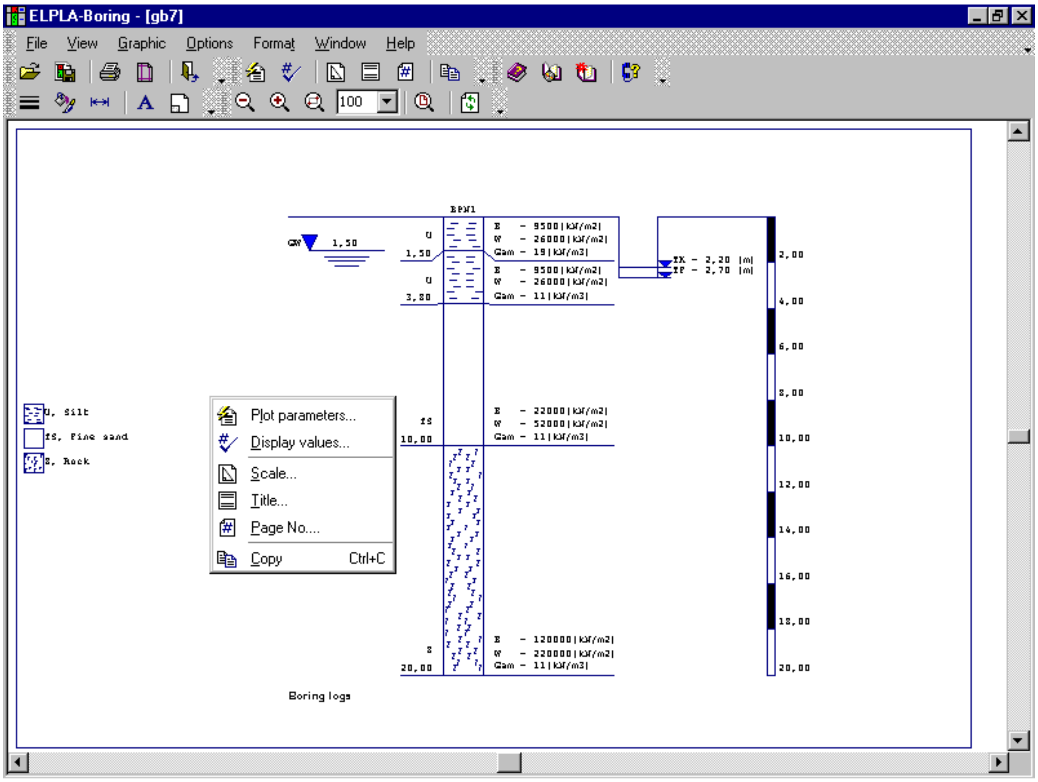


Figure H-24 Menu "Popup-Options"

12 Samples for graphical drawings using ELPLA-Boring

ELPLA-Boring draws the soil layers by different symbols, where the same layers are presented with the same symbols for easy identification. For drawing the boring logs, the main symbols of German specification code are considered in ELPLA-Boring. It is possible to draw a total number of 46 soil and rock arts in boring logs according to German Standard DIN 4023.

Samples for symbols of soil layers according to German Standard DIN 4023 are presented in page P1. The pages P2 to P3 show also some graphical presentations of boring logs and limit depth according to Table H-12 and Table H-13.

12.1 Graphical drawings of boring logs

Table H-12 Boring logs

Presentation	Page
Boring logs according to DIN 4023	P1
Simple presentation of boring logs	P2

12.2 Graphical drawings of limit depth

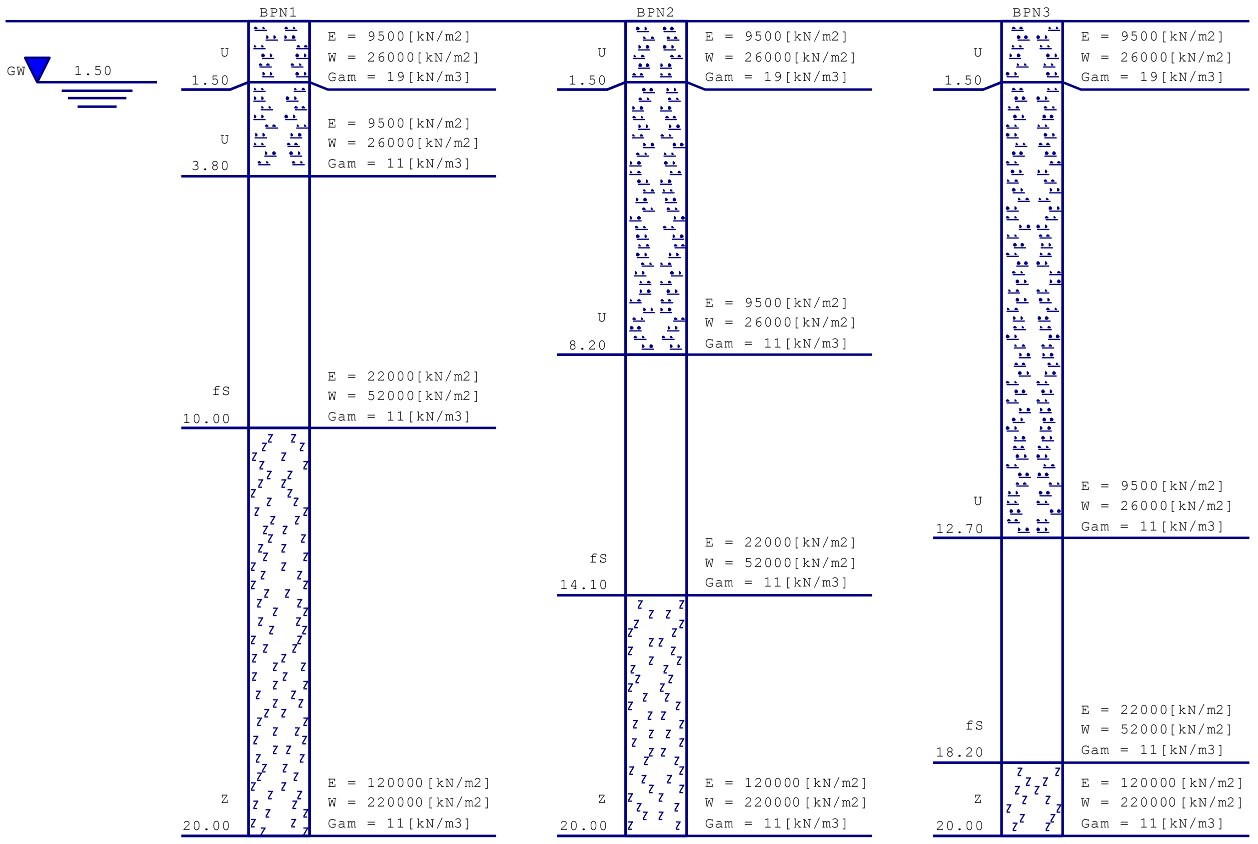
Table H-13 Limit depth

Presentation	Page
Limit depth	P3

13 Reference

[1] DIN 4023. Baugrund- und Wasserbohrungen. Zeichnerische Darstellung der Ergebnisse Beuth-Verlag, Berlin 1984

14 Index

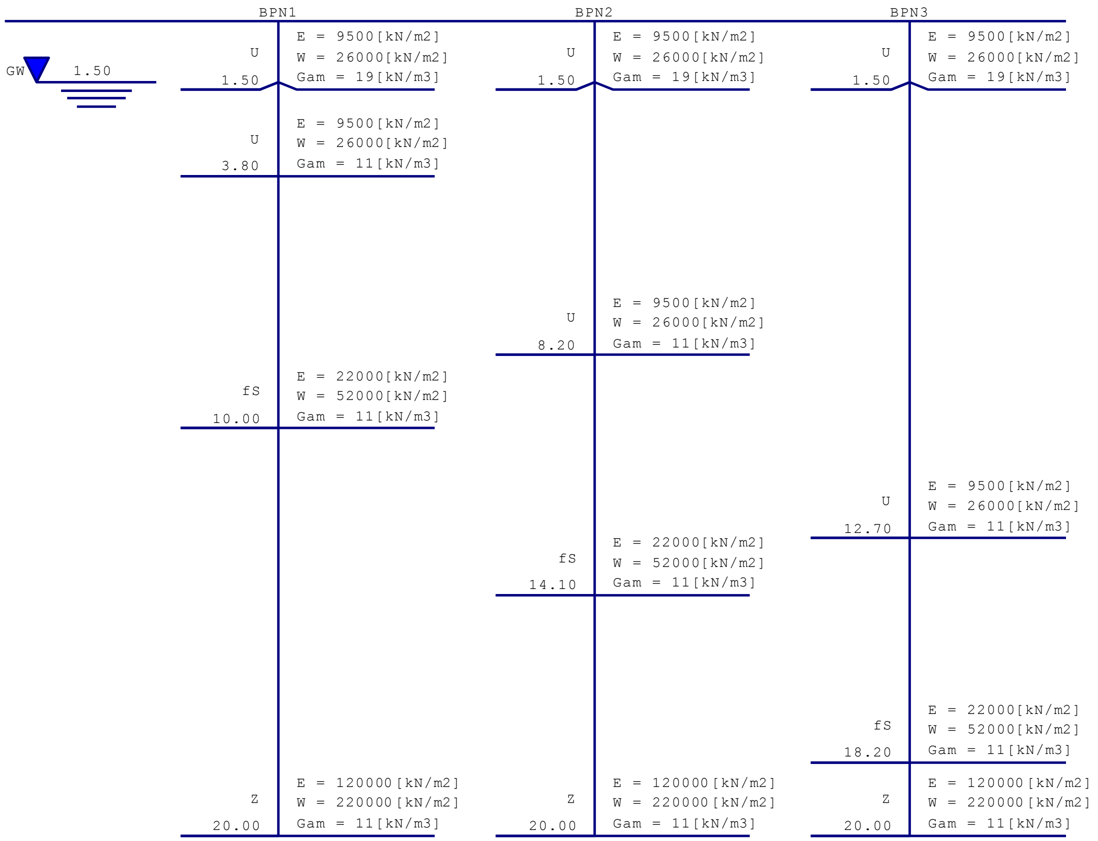


Boring Logs

Geotec Office

PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:150	Title: An irregular raft on irregular subsoil
File: gb3	Date: 13.07.1998
Page No.:	Project: Method 3

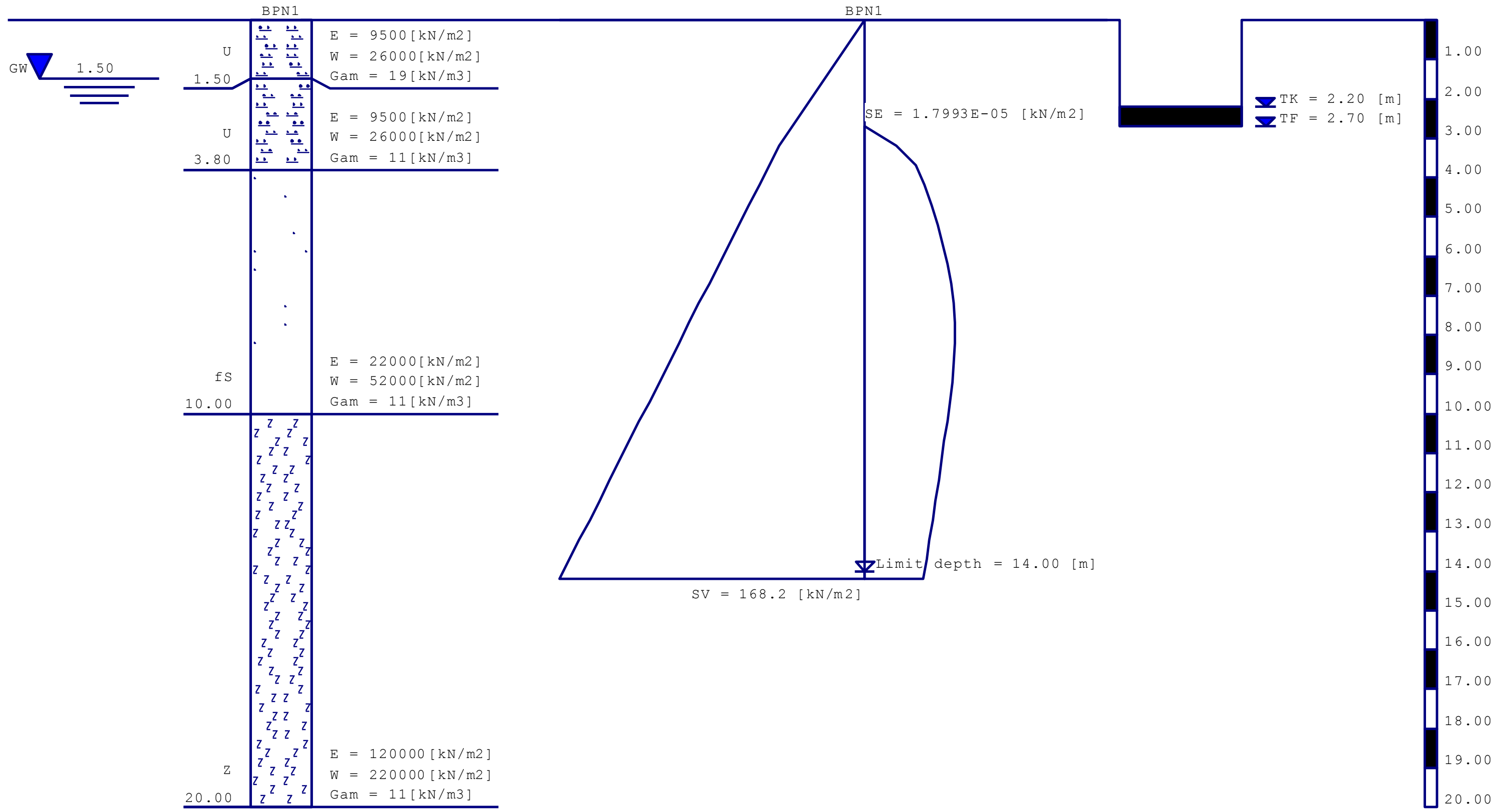


Boring Logs

Geotec Office

PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:150	Title: An irregular raft on irregular subsoil
File: gb3	Date: 13.07.1998
Page No.:	Project: Method 3



Boring Logs/ Limit Depth

Geotec Office

PO Box 14001 Richmond Road PO - Calgary AB, Canada T3E 7Y7

Scale 1:155	Title: An irregular raft on irregular subsoil
File: gb3	Date: 13.07.1998
Page No.:	Project: Method 3

Part I

User's Guide
for the program GEOTEC-Editor

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

Version 9.2

Program authors: M. El Gendy
A. El Gendy

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

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

Table of Contents Page

Page

1 An overview of GEOTEC-Editor

GEOTEC-Editor is a simple word processing program. The project data or results can be displayed, changed or printed through GEOTEC-Editor. Data or results, if desired, can be saved as ASCII-format or RTF-Format files. These files can be exported to other Windows-applications (for example WordPerfect or Word) to prepare reports or add further information.

2 Description of GEOTEC-Editor

GEOTEC-Editor is a 32-bit software product that operates under Microsoft Windows 9x/ NT/ ME/ XP. The common "what you see is what you get" of Windows applications makes it easy to learn how to use GEOTEC-Editor, especially if you are already familiar with the Windows environment.

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

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

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

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

The next paragraphs describe the purpose and function of each GEOTEC-Editor command.

3 Starting GEOTEC-Editor

Start GEOTEC-Editor by clicking the program icon in the Windows "Start"-Menu. The introduction screen (Figure I-1) appears.

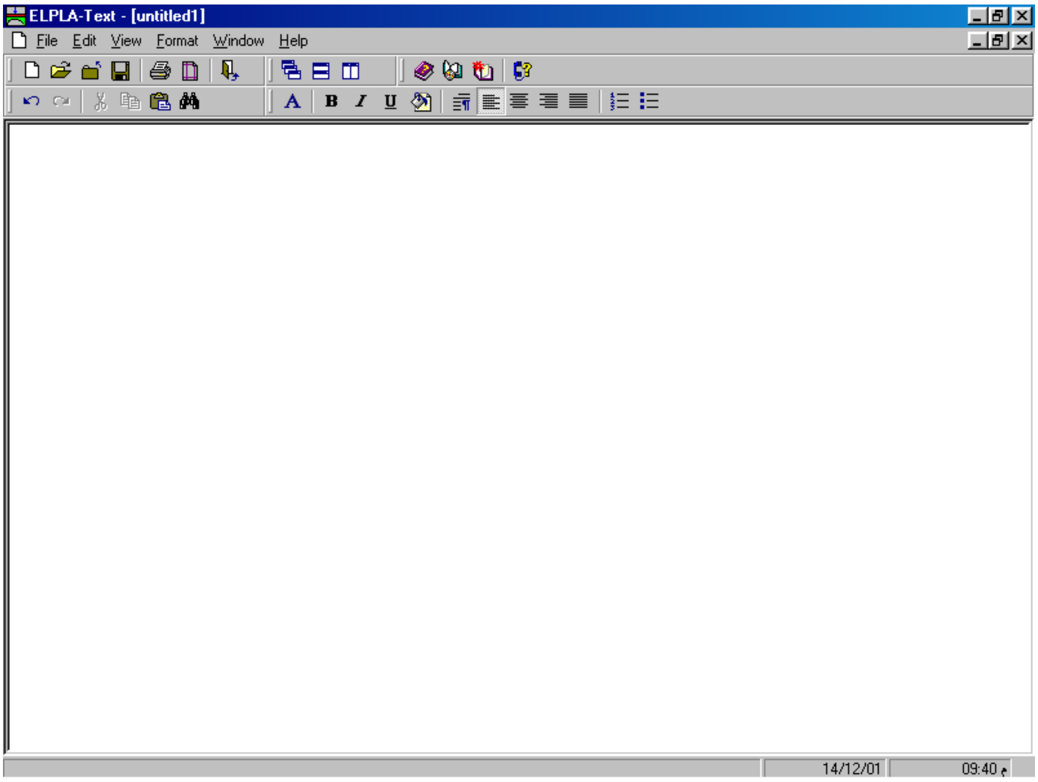


Figure I-1 Introduction screen of GEOTEC-Editor

The menu head of Figure I-1 contains the following 6 commands:

- File
- Edit
- View
- Format
- Window
- Help

After clicking a command, other sub-commands or options become available. These commands and their sub-commands are presented and described in the following paragraphs 4 to 10.

4 File Menu

The File Menu commands are:

- New
- Open
- Close
- Save
- Save as
- Print
- Page setup
- Files 1, 2, 3, 4
- Exit

4.1 File Menu –"New" command

By "New" command a new file is created.

4.2 File Menu –"Open" command

By "Open" command an existing file is opened. Figure I-2 shows "Open" Dialog box used to open a specified file.

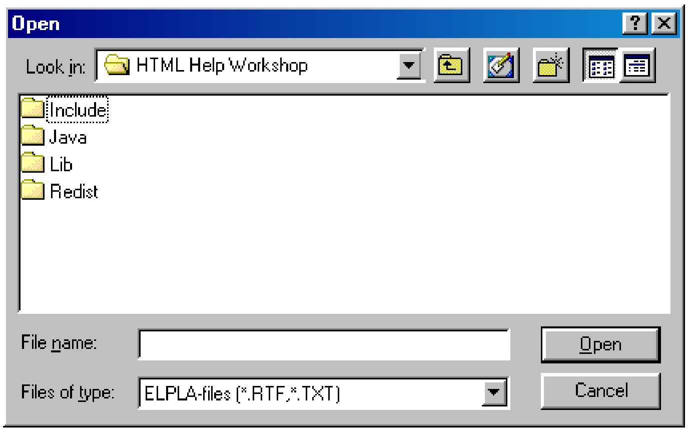


Figure I-2 "Open" Dialog box

4.3 File Menu –"Close" command

By "Close" command the current file is closed.

4.4 File Menu –"Save" command

By "Save" command the current file is saved.

4.5 File Menu –"Save as" command

By "Save as" command the current file is saved under a new file name.

4.6 File Menu–"Print" command

By "Print" command text can be printed. Only the text file currently displayed is printed. Figure I-3 shows "Print" Dialog box. The printer group box contains controls for selecting the printer and changing its properties. Use "Name" Combo box to select the printer and use "Properties" Button to set printer settings. The number of printing copies can be defined in "Copies" Input box (Figure I-3). For more information about printer settings, see your Windows documentation.

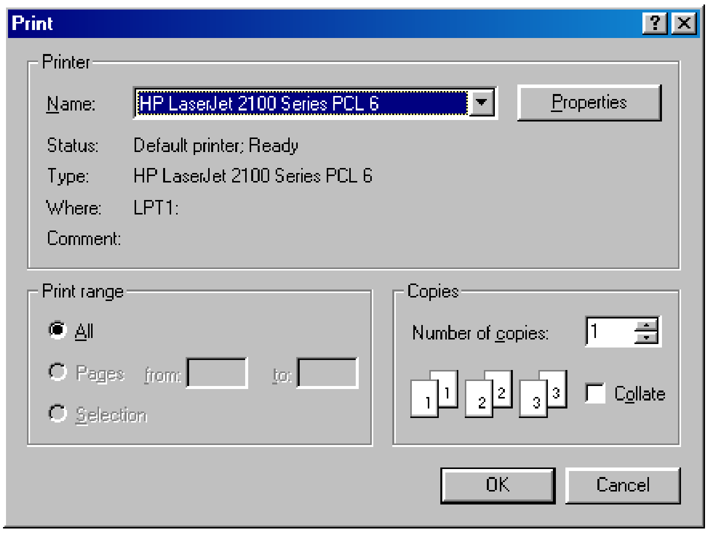


Figure I-3 "Print" Input box

4.7 File Menu–"Page Setup" command

By "Page setup" command the standard "Page setup" Dialog box can be displayed with options to specify the printer, page orientation, paper size, and paper source, as well as other printing options. Figure I-4 shows "Page setup" Dialog box.

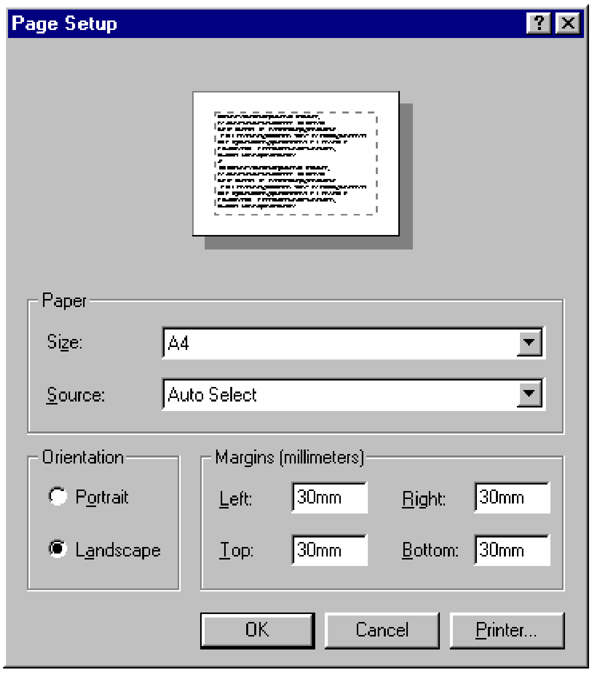


Figure I-4 "Page setup" Dialog box

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

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

4.9 File Menu –"Exit" command

By "Exit" command all files are closed and GEOTEC-Editor is quitted, Figure I-6.

Figure I-5 "Exit" Message box

5 Edit Menu

The Edit Menu commands are:

- Undo
- Redo
- Cut
- Copy
- Paste
- Clear
- Select all
- Find
- Find next
- Replace

5.1 Edit Menu–"Undo" command

By "Undo" command the user can undo the last action.

5.2 Edit Menu–"Redo" command

By "Redo" command the user can redo the last action.

5.3 Edit Menu–"Cut" command

Here the selected text can be removed, so that it can be placed in another part of the text file.

5.4 Edit Menu–"Copy" command

By "Copy" command the selected text is copied in the Clipboard so that it can be placed in another part of the text file.

5.5 Edit Menu–"Paste" command

By "Paste" command the contents of the Clipboard are placed in the text file.

5.6 Edit Menu–"Clear" command

By "Clear" command the selected text can be removed from the text file.

5.7 Edit Menu–"Select all" command

By "Select all" command all text at once can be selected.

5.8 Edit Menu–"Find" command

By "Find" command the user can search for a specified text. Figure I-6 shows "Find" Dialog box. In "Find what" Text box the user enters the text to search for. Then click "Find next" Button. GEOTEC-Editor will start searching according to the following options:

Match case
Finds only text that has the same pattern of uppercase and lowercase characters as the text was specified in "Find what" Text box

Find whole words only
Finds only whole words instead of searching for text inside longer words

Find in selection only
Searches inside only the selected text instead of searching for text inside all file

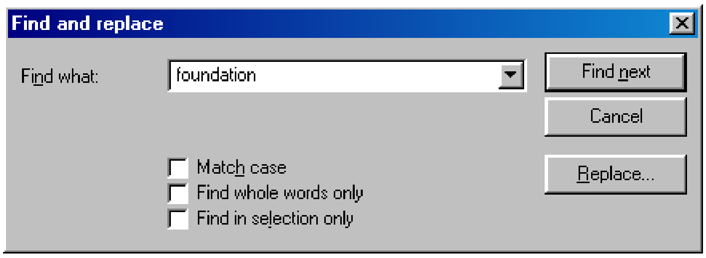


Figure I-6 "Find" Message box

5.9 Edit Menu–"Find next" command

By "Find next" command the user can search for more instances of the same text that was specified in "Find what" Text box of Figure I-6.

5.10 Edit Menu–"Replace" command

By "Replace" command the user can search for and replace a specified text. Figure I-7 shows "Replace" Dialog box. In "Find what" Text box the user enters the text to search for. In "Replace with" Text box the user enters the text to replace with "Find what" Text box. Then click "Replace" Button. GEOTEC-Editor searches for the next instance of the text that matches the text in "Find what" Text box and replaces it with the text in "Replace with" Text box according to the following options:

Match case
Finds and replaces only text that has the same pattern of uppercase and lowercase characters as the text was specified in "Find what"

Find whole words only
Finds and replaces only whole words instead of searching for text inside longer words

Find in selection only
Searches and replaces inside only the selected text instead of searching for text inside all file

Note
By "Replace All" command all instances of the text can be replaced

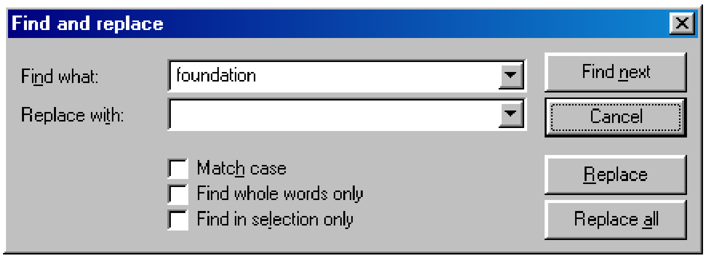


Figure I-7 "Replace" Message box

6 View Menu

The View Menu commands are:

- Status bar
- Tool bars

6.1 View Menu–"Status bar" command

"Status bar" command displays a status bar on the screen down. The status bar displays information about the progress of the current operation.

6.2 View Menu–"Tool bars" command

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

7 Format Menu

The Format Menu commands are:

- Font
- Paragraph
- Numbering
- Bullets
- Protect

7.1 Format Menu –"Font" command

By "Font" command the font type, style, or size (Figure I-8) for the selected text, if existing, or new text can be defined.

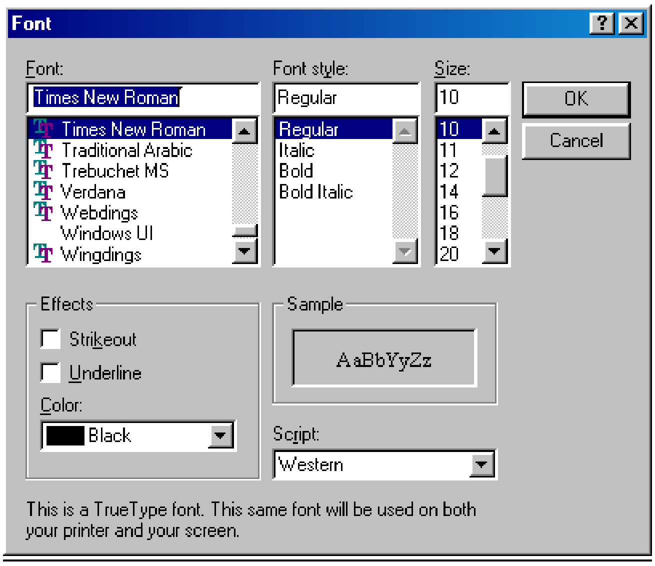


Figure I-8 "Font" Dialog box

7.2 Format Menu –"Paragraph" command

By "Paragraph" command the paragraphs (Figure I-9) for the selected text, if existing, or new text can be formatted.

The following formats can be defined:

Left
Specifies the number of units (inches) to indent text from the left margin

Right
Specifies the number of units (inches) to indent text from the right margin

First line
Specifies the number of units (inches) to indent text from the left margin of the first line

Alignments
The alignments that are available for the selected paragraph can be listed as follows:

Left	Aligns text at the left indent
Right	Aligns text at the right indent
Centered	Centers the text between margins
Justified	Justifies the text between margins

Before
Specifies the number of units (Twips) to leave before the paragraph

After
Specifies the number of units (Twips) to leave after the paragraph

Line spacing
The line spacing that is available for the selected paragraph can be listed as follows:

Single	The line spacing is single space
1.5 single	The line spacing is 1.5 single space
Double	The line spacing is double space

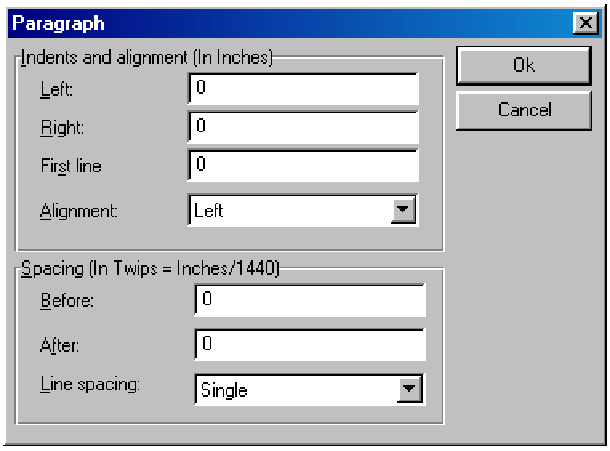


Figure I-9 "Paragraph" Dialog box

7.3 Format Menu –"Numbering" command

By "Numbering" command the selected text can be numbered.

7.4 Format Menu –"Bullets" command

By "Bullets" command the selected text can be bulleted.

7.5 Format Menu –"Protect" command

By "Protect" command the text file can be protected from any editing process. Before applying any editing process a message will appear as shown in Figure I-10.

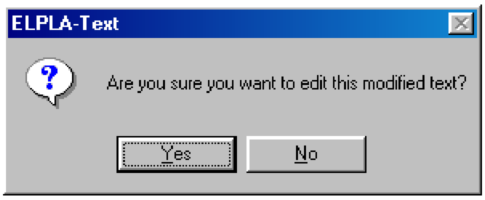


Figure I-10 Edit Message box

8 Window Menu

The Window menu has the following commands:

- Cascade
- Tile horizontally
- Tile vertically
- Arrange icons
- Window 1, 2, 3, …

8.1 Window Menu–"Cascade" command

By "Cascade" command all non-minimized forms are cascaded.

8.2 Window Menu–"Tile horizontally" command

By "Tile horizontally" command all non-minimized forms are tiled horizontally.

8.3 Window Menu–"Tile vertically" command

By "Tile vertically" command all non-minimized forms are tiled vertically.

8.4 Window Menu–"Arrange icons" command

When choosing "Arrange icons" command, the icons of minimized forms are arranged.

8.5 Window Menu–"Window 1, 2, 3, …" command

By this command the user can display one list of the pervious loaded data or results.

9 Help Menu

The Help Menu commands are:

- Contents
- Short description of ELPLA
- New in ELPLA
- About GEOTEC-Editor

9.1 Help Menu–"Contents" command

"Contents" command displays a help file in HTML-Format containing the complete ELPLA User's Guide .

9.2 Help Menu–"Short description of ELPLA" command

"Short description of ELPLA" command gives a short description of ELPLA package.

9.3 Help Menu–"New in ELPLA" command

"New in ELPLA" command summarizes the new features and enhancements in ELPLA.

9.4 Help Menu–"About GEOTEC-Editor" command

Clicking the command "About GEOTEC-Editor" displays the information form of GEOTEC-Editor, which gives information about GEOTEC-Editor.

10 Tips and Tricks

10.1 Keyboard

The user can obtain all menu titles and commands also through Shortcut keys. The action of the Shortcut keys is listed in Tables I-2 to I-8:

Table I-2 Shortcut keys of menu head

Shortcut keys	Action	
[Alt+f] [Alt+e]	Calling menu head	"File" "Edit"
[Alt+v]		"View"
[Alt+f]		"Format"
[Alt+w]		"Window"
[Alt+h]		"Help"

Table I-3 Shortcut keys of File-Command

Shortcut keys	Action	
[Ctrl +n] or [Alt+f] then [n]	Calling command	"New"
[Ctrl +o] or [Alt+f] then [o]		"Open"
[Alt+f] then [c]		"Close"
[Ctrl +s] or [Alt+f] then [s]		"Save"
[Alt+f] then [a]		"Save as"
[Ctrl +p] or [Alt+f] then [p]		"Print"
[Alt+f] then [u]		"Page setup"
[Alt+f] then [1]	Calling the first file from the last four loaded files	
[Alt+f] then [2]	Calling the second file from the last four loaded files	
[Alt+f] then [3]	Calling the third file from the last four loaded files	
[Alt+f] then [4]	Calling the fourth file from the last four loaded files	
[Ctrl+q] or [Alt+f] then [x]	Calling command	"Exit"

Table I-4 Shortcut keys of Edit-Command

Shortcut keys	Action
[Ctrl+z] or [Alt+e] then [u]	Calling command "Undo"
[Ctrl+y] or [Alt+e] then [r]	"Redo"
[Ctrl+x] or [Alt+e] then [t]	"Cut"
[Ctrl+c] or [Alt+e] then [c]	"Copy"
[Ctrl+v] or [Alt+e] then [p]	"Paste"
[Del] or [Alt+e] then [c]	"Clear"
[Alt+e] then [a]	"Select all"
[Ctrl+f] or [Alt+e] then [f]	"Find"
[F3] or [Alt+e] then [n]	"Find next"
[Ctrl+h] or [Alt+e] then [r]	"Replace"

Table I-5 Shortcut keys of View-Command

Shortcut keys	Action
[Alt+v] then [b]	Calling command "Status bar"
[Alt+v] then [t]	"Tool bars"
[Alt+v] then [t], then [f]	"Tool bars-File"
[Alt+v] then [t], then [e]	"Tool bars-Edit"
[Alt+v] then [t], then [t]	"Tool bars-Format"
[Alt+v] then [t], then [w]	"Tool bars-Window"
[Alt+v] then [t], then [h]	"Tool bars-Help"
[Alt+v] then [t], then [s]	"Tool bars-Style-Transparent"

Table I-6 Shortcut keys of Format-Command

Shortcut keys	Action
[Alt+t] then [f]	Calling command "Font"
[Alt+t] then [p]	"Paragraph"
[Alt+t] then [n]	"Numbering"
[Alt+t] then [b]	"Bullets"
[Alt+t] then [r]	"Protect"

Table I-7 Shortcut keys of Window-Command

Shortcut keys	Action
[Alt+w] then [c]	Calling command "Cascade"
[Alt+w] then [h]	"Tile horizontally"
[Alt+w] then [v]	"Tile vertically"
[Alt+w] then [a]	"Arrange icons"
[Alt+w] then [1, 2, 3, …]	"Window 1, 2, 3, …"

Table I-8 Shortcut keys of Help-Command

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

10.2 Mouse

By clicking the right mouse Button on the screen the user can also obtain the Popup-Format-Menu, Figure I-13.

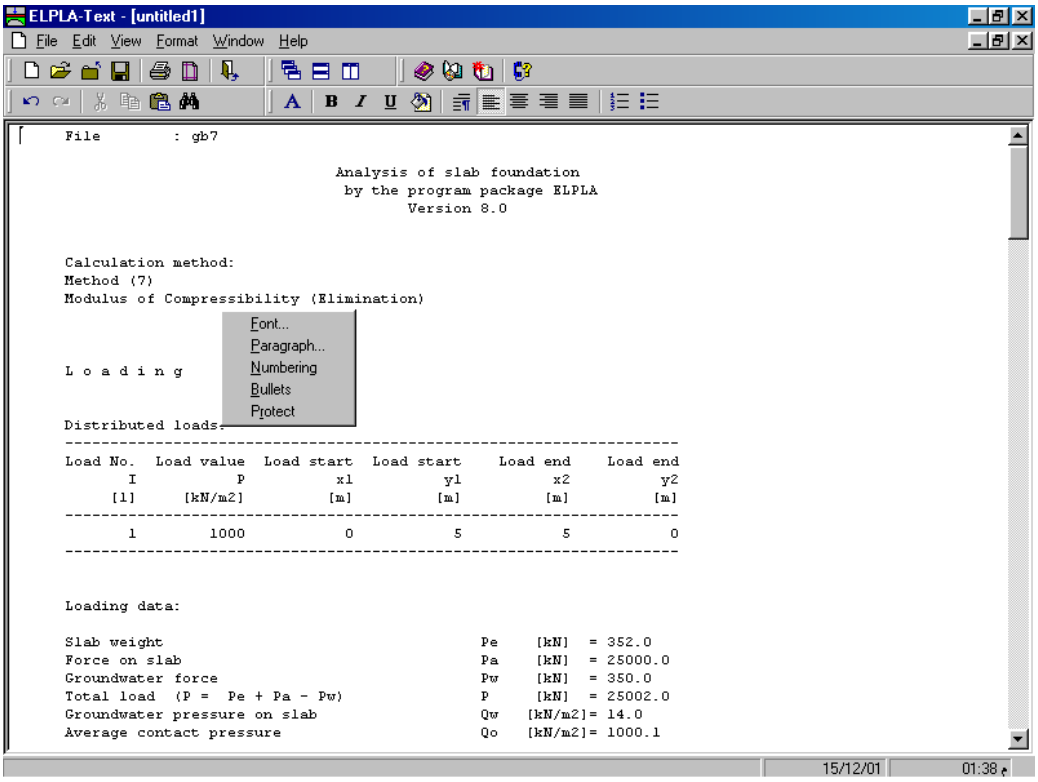


Figure I-13 Menu "Popup-Format"

11 Index