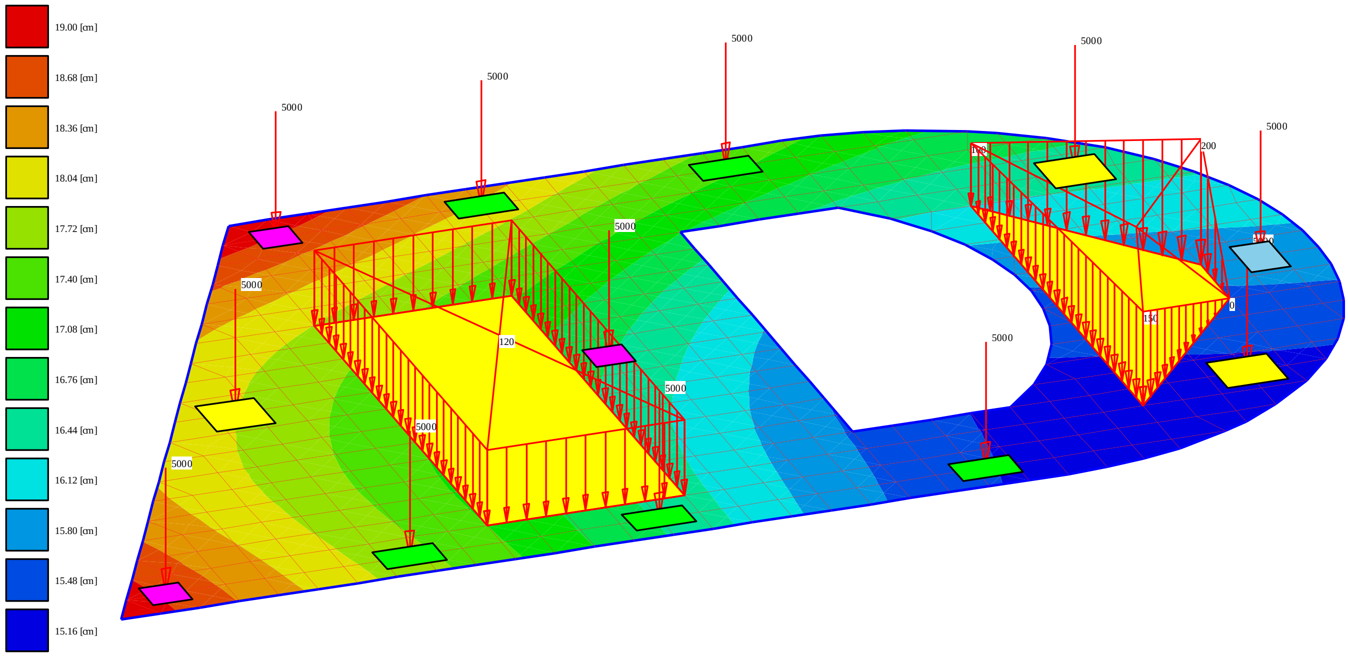
Analysis and Design by the Program *ELPLA*

Description of the program *ELPLA*



Program authors: *M. El Gendy*
A. El Gendy

GEOTEC Software Inc.
PO Box 14001 Richmond Road PO, Calgary AB, Canada T3E 7Y7
Tele.:+1(587) 332-3323
geotec@geotecsoftware.com
www.geotecsoftware.com

2022

Content Page

1 Preface

GEOTEC and Prof. Dr.-Ing. Manfred Kany the founder of Numerical Analysis in Geotechnical Engineering and the father of Elastic Foundations

Prof. Dr.-Ing. Manfred Kany and Prof. M. El Gendy celebrated the first release of ELPLA at Zirndorf, Germany in 1994

In 1961, the Geotechnical Institute LGA Nuremberg in Germany was the first institute to propose the use of computers for settlement calculations. For this purpose, a computer program for a ZUSE Z-23 computer was developed and presented at the German Geotechnical Conference in Berlin in 1964. At that time, Prof. Kany, who was the head of the Geotechnical Institute LGA Nuremberg since 1955, developed computer programs for Geotechnical problems during his lifetime in LGA.

He was the first researcher in Germany who dealt with electronic calculations, with the collaboration of his friend Prof. Konrad Zuse - the inventor of the computer "Zuse 4". The computer accompanied the entire career of Prof. Kany until shortly before his death in 2011. He is considered the founder of Numerical Analysis in Geotechnical Engineering. After retirement in 1987, Prof Kany established his firm, GEOTEC and developed the series of GEOTEC programs.

In 1994, Prof. Mohamed El Gendy finished his Ph.D. under the supervision of Prof. Kany. Based on his Ph.D. research, Prof. El Gendy further developed the program ELPLA, which was initially developed by Prof. Kany for analyzing rafts by the Modulus of Compressibility method. Subsequently, Dr. Amin El Gendy joined the team during the development of the GEOTEC Office suite. Prof. Kany is considered the father of elastic foundations, while Prof. El Gendy is deemed his successor and the son of the father of elastic foundations.

GEOTEC SOFTWARE INC.

In 2014, GEOTEC Software Inc. was re-incorporated in Canada to continue developing GEOTEC Office suite.

2 Description of the program ELPLA

ELPLA (*EL*astic *PLA*te) is a program for analyzing raft/ piled raft of arbitrary shape, variable thickness, and foundation depth with a real subsoil model.

Three known subsoil models for the analysis of raft/ piled raft (standard models) are considered. The subsoil models are the Simple Assumption Model, the Winkler Model, and the Continuum 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 in vertical and horizontal directions.

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 element mesh of the elastic foundation. Therefore, the three analyses can easily be compared and correctly.

ELPLA can also be used for:

- Represent the effect of external loads, neighboring foundations, tunneling, and the influence of the temperature difference on the raft.
- Analysis slab floor, plane frame, plane stress, grid, a system of many slab foundations, beam or grid on elastic foundation, rotational shell, axisymmetric stress, axisymmetric structures, and cylindrical tanks.
- Design the raft, piled raft, and slab floor according to the ACI, EC 2, DIN 1045, and ECP.
- Dynamic analysis of structures.
- Determining stresses, strains, and displacements in soil.

ELPLA is a graphical software product. The common "what you see is what you get" in Windows applications makes it easy to learn how to use *ELPLA*. The new features and enhancements are the result of feedback from users over the last years.

The usage of the program is typically such that data files are created describing a certain problem by Data Tab, and then the project problem is analyzed by using Solver Tab. Finally, the results can be presented as a graphical drawing, graphs, and tables using the six separate command groups: Data, Results, Soil Data, Boring, Section, and List Groups in Results Tab.

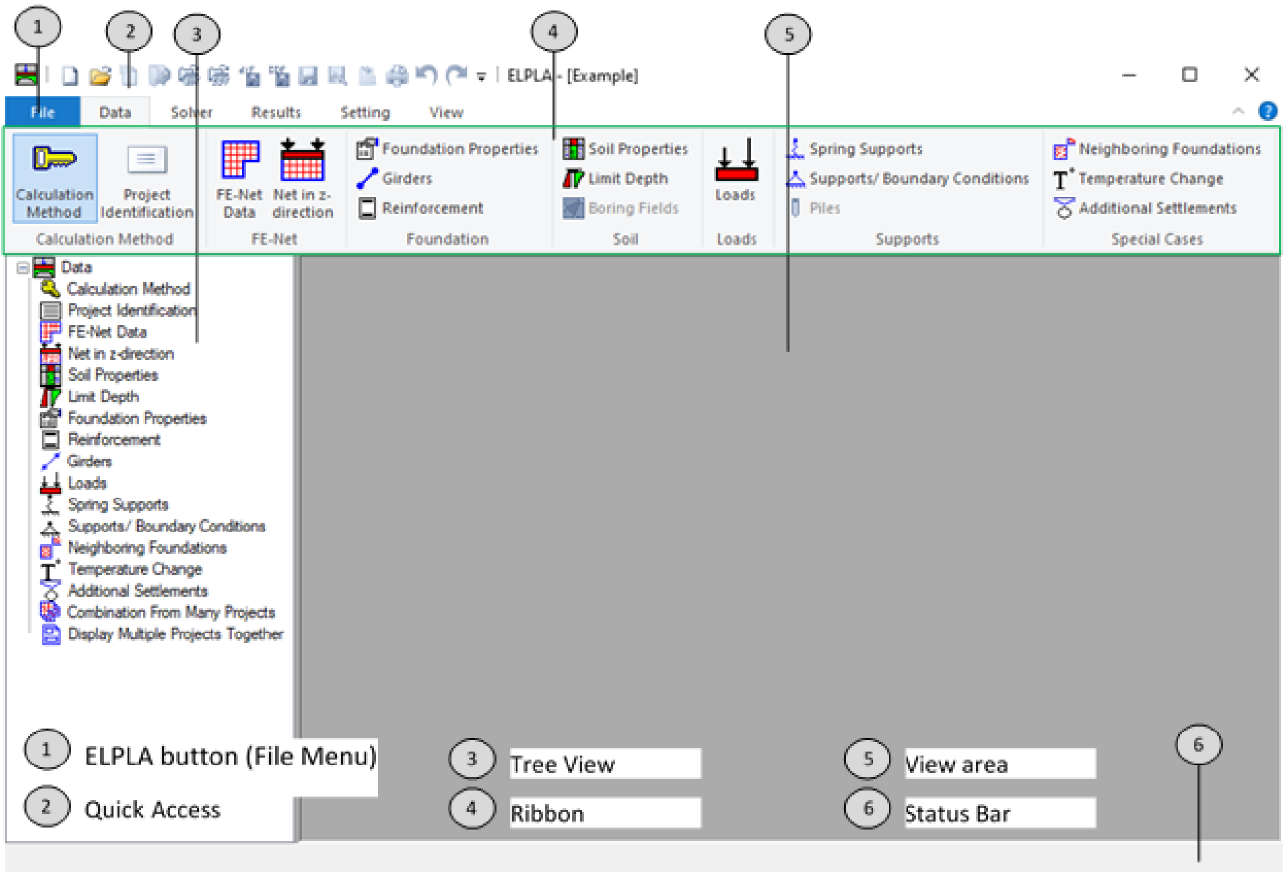


Figure 1 Introduction screen of the program ELPLA

3 Features

- User interface and help system are available in 3 languages: English, German and Arabic
- Analysis of an elastic, a rigid or flexible foundation
- Numerical model of soil-structure interaction is under 9 calculation methods
- Design of the raft according to ACI, EC 2, DIN 1045 and ECP
- Generation of the FE mesh of the raft with different element types
- Automatic generation of the FE mesh of the raft
- Smoothing the FE mesh
- Direct and arrange all elements
- Refining the FE mesh
- Merge two or more nets
- Split the FE-Net into two or more nets
- Powerful mesh generator (for the generation of square, rectangular, circular and annular rafts)
- Beam elements for modeling stiff walls on the raft
- Translational and rotational springs on the raft can be added at nodes
- Elastic or fixed rotations and deflections can be taken into account.
- Determining contact pressures, settlements, internal forces, subgrade reactions and reinforcement of the slab
- Node coordinates and boundary nodes of the FE mesh can be imported from a table via MS Excel
- Arbitrary shape of slabs, holes are also possible
- Variable slab thickness and foundation depth in vertical and horizontal directions
- Consideration of the reduction coefficients α according to DIN 4019 Part 1
- Point loads, line loads, area loads and moments at any position independent of the finite element net
- Polygonal load with variable ordinates and line moment
- Loading and reloading modulus of compressibility are considered
- The soil is defined by a number of borings each boring has multi-layers with different soil material
- Variable thickness and discontinuous soil strata
- Consideration of the variation of the subsoil in the three directions according to three methods
- Drawing soil layers by different symbols and colors according to DIN 4023 for easy identification
- Consideration of groundwater and overburden pressure effects
- Color representation of the dimensions, slab plans and results on the screen or printer
- Presentation of the results as values in the plan, contour lines, circular diagrams
- Drawing results in isometric view
- Distribution of results in plan
- Drawing deformations as deformed mesh
- Principal moments as streaks
- Drawing sections of results from several calculation methods in one view
- Data and results of several projects can be displayed together
- Tabulation of data and final results on the screen or printer
- Results can be saved in an ASCII file
- The drawings can optionally be saved as a WMF file
- There are detailed explanations in the user manual with numerical examples
- Short help information can be requested at any interface location

- Import or export the data to MS Excel
- Export the results and diagrams to MS Excel
- Export the data and results to MS Word
- A group of data with results together in one presentation
- Copying drawings to the clipboard for use in word processors
- Analyzing system of rafts or piled rafts in one mesh
- Analysis of a rigid pile group or free-standing raft on a rigid pile group
- Analysis of a slab floor, plane frame or plane stress, grid
- Analysis of system of many slab foundations
- Analysis of rotational shell, axisymmetric stress or axisymmetric structures
- Analysis of cylindrical tanks
- Analysis of beam or grid on elastic foundation
- Dynamic analysis of structures
- Determining stresses, strains, and displacements in soil
- Importing references from a DXF-file into *ELPLA*
- Importing FE-Net as “3DFACE” AutoCAD type into *ELPLA*
- Create a DXF-File from the finite element mesh or any graphic in the FE-Net mode
- Create a 3DFACE-File from the finite element mesh of rectangular elements
- Reduce computational time and computer storage by using the system symmetry

4 Calculation methods

ELPLA can be used to analyze raft/piled raft or any other structural problems such as slab floors, grids, plane frame, plane stress, a system of many slab foundations, rotational shell, axisymmetric stress, and axisymmetric structures (Figure 2)

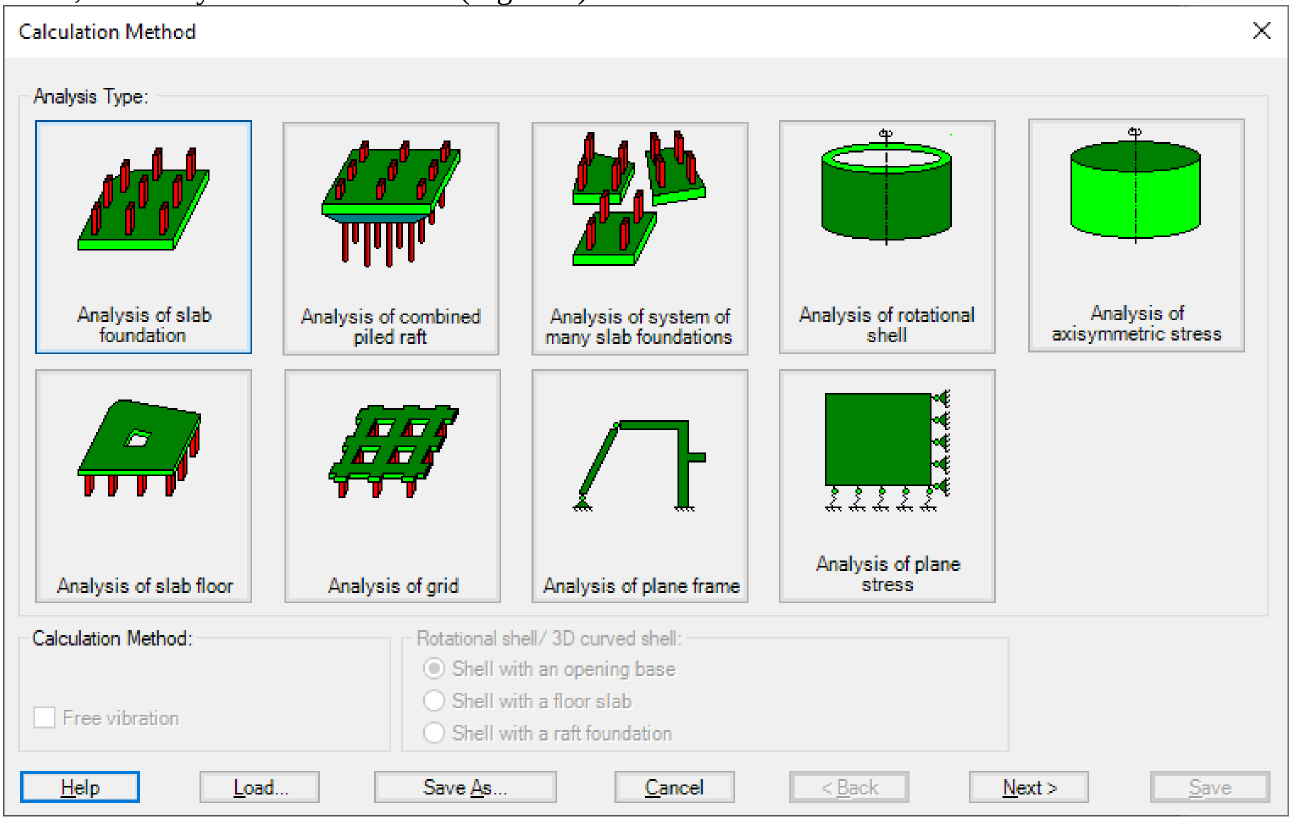


Figure 2 Calculation Methods

4.1 Analysis of slab foundation

The analysis of slab foundation problems is available in *ELPLA* (Figure 3 and Figure 4).

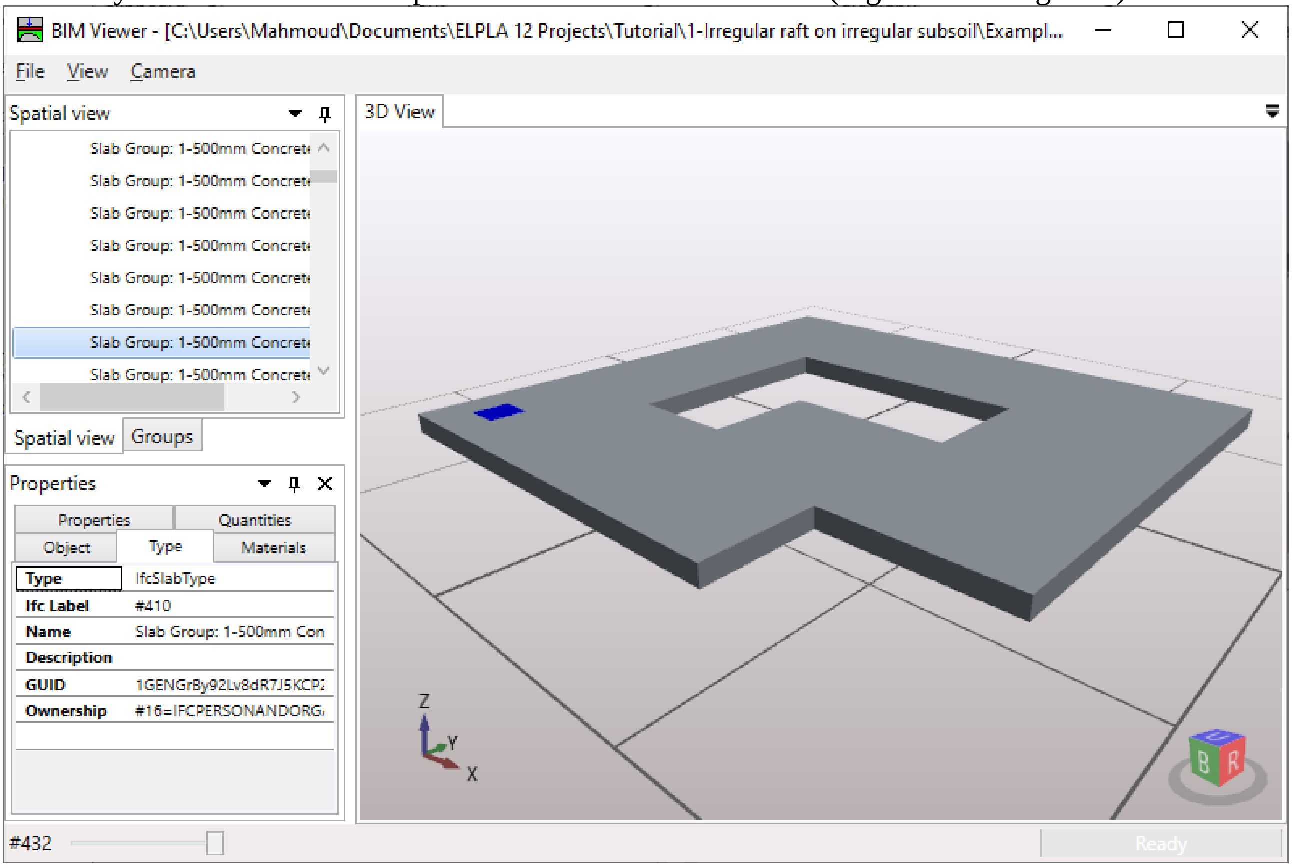


Figure 3 "Analysis of slab foundation"

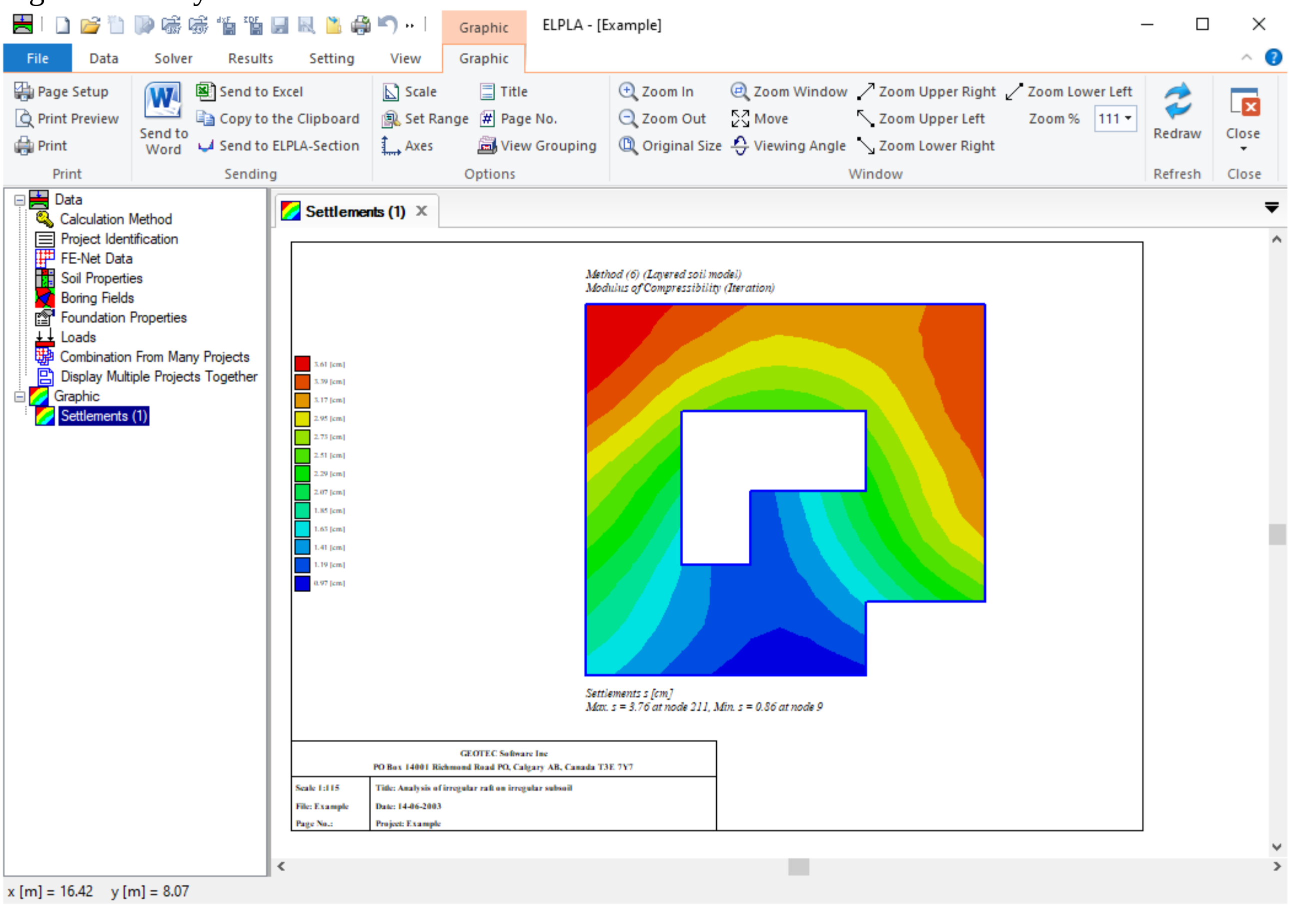


Figure 4 "Analysis of slab foundation" displacements

4.2 Analysis of slab floor

The analysis of slab floor problems is available in *ELPLA* (Figure 5 and Figure 6).

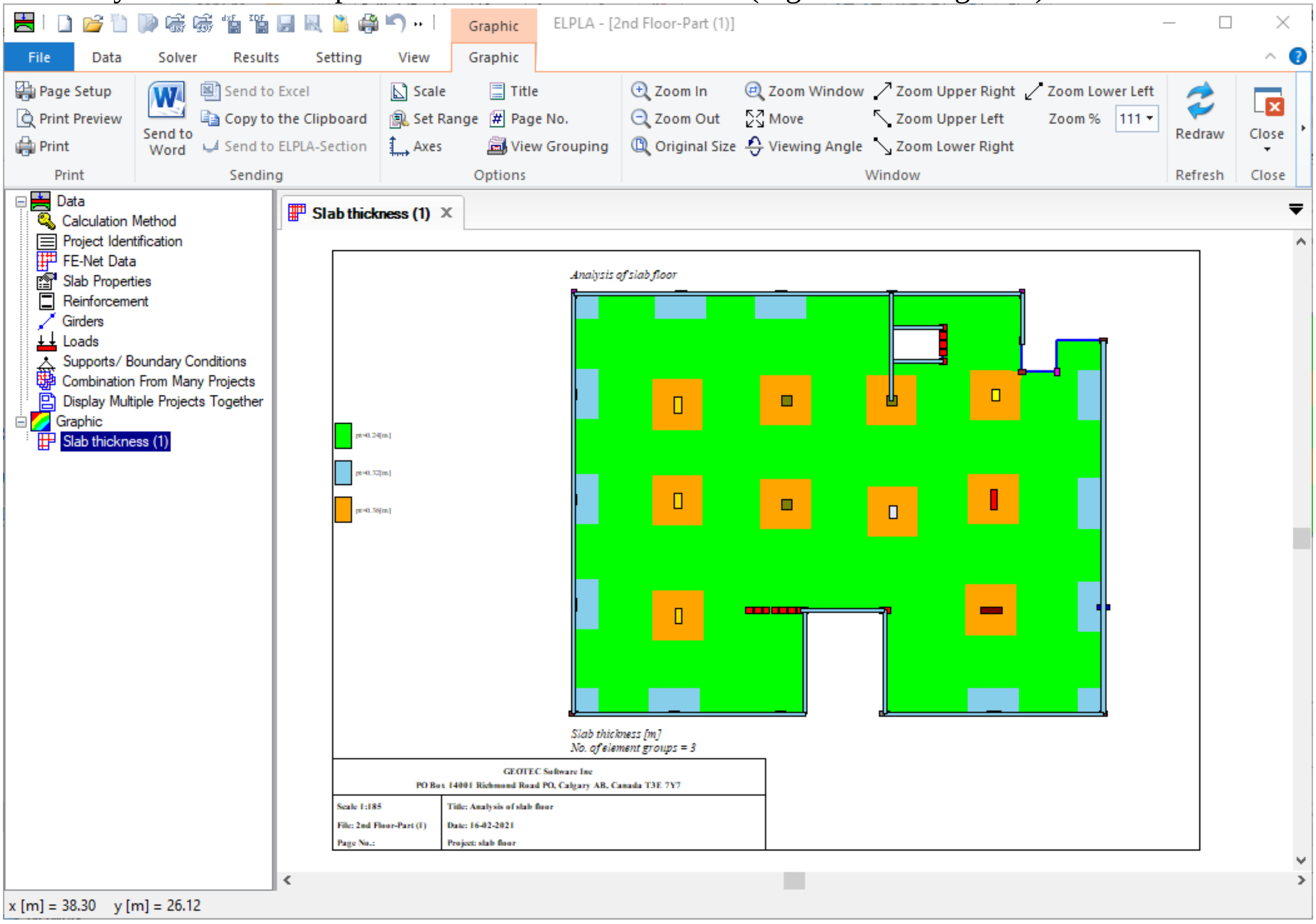


Figure 5 "Analysis of slab floor" slab thickness

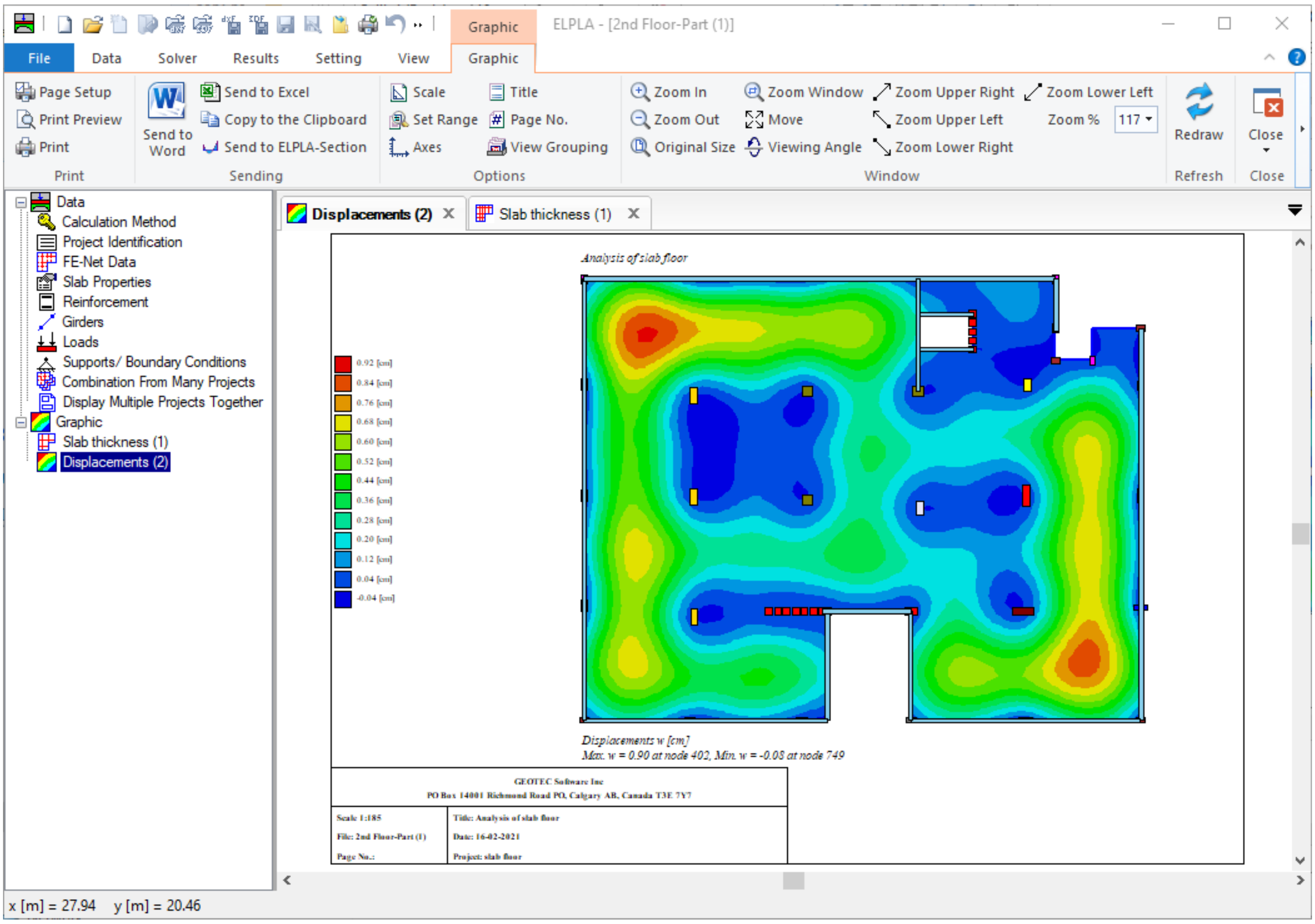


Figure 6 "Analysis of slab floor" displacements

4.3 Analysis of combined piled raft

The analysis of piled raft problems is available in *ELPLA* (Figure 7 and Figure 8).

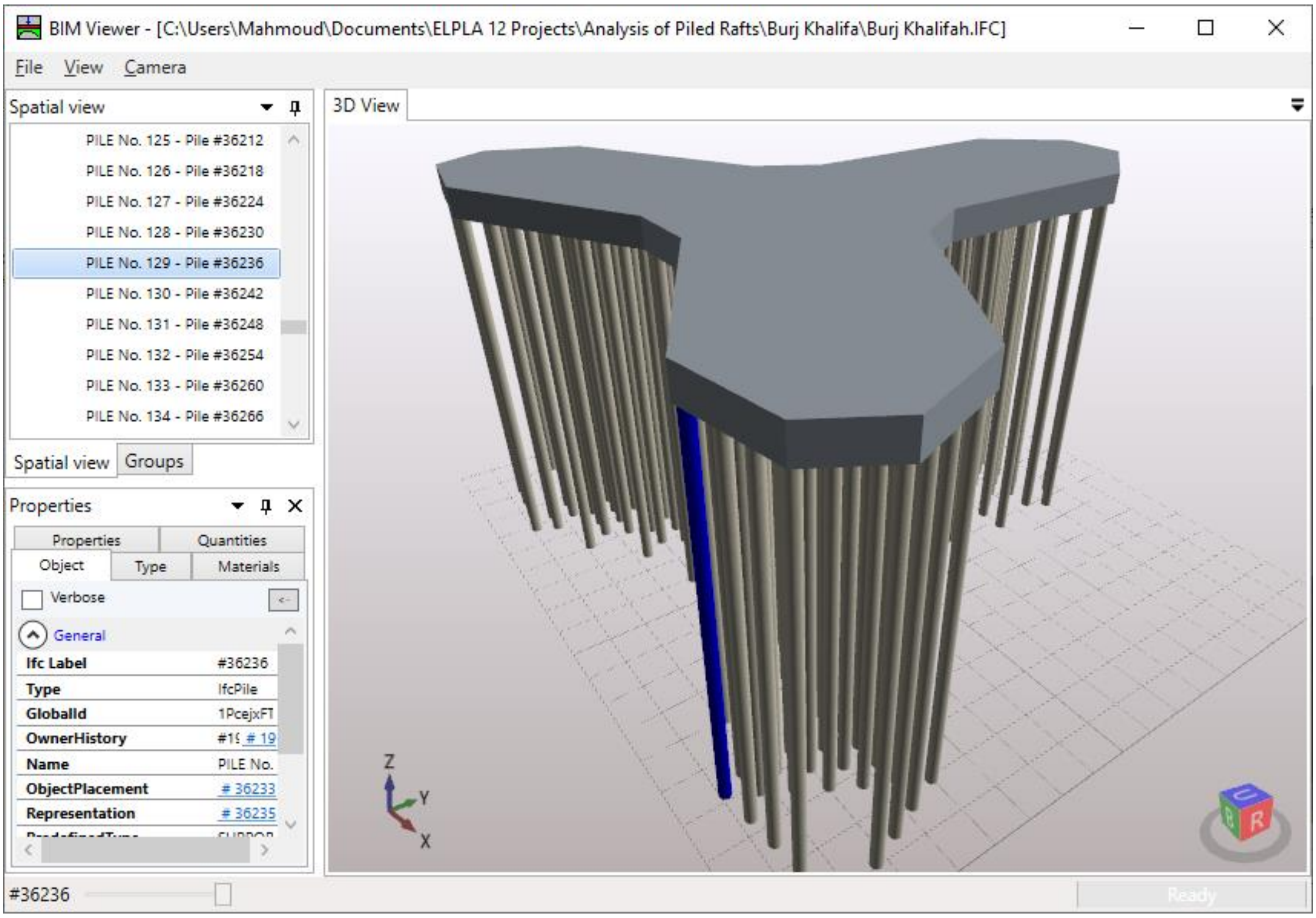


Figure 7 "Analysis of Burj Khalifah foundation"

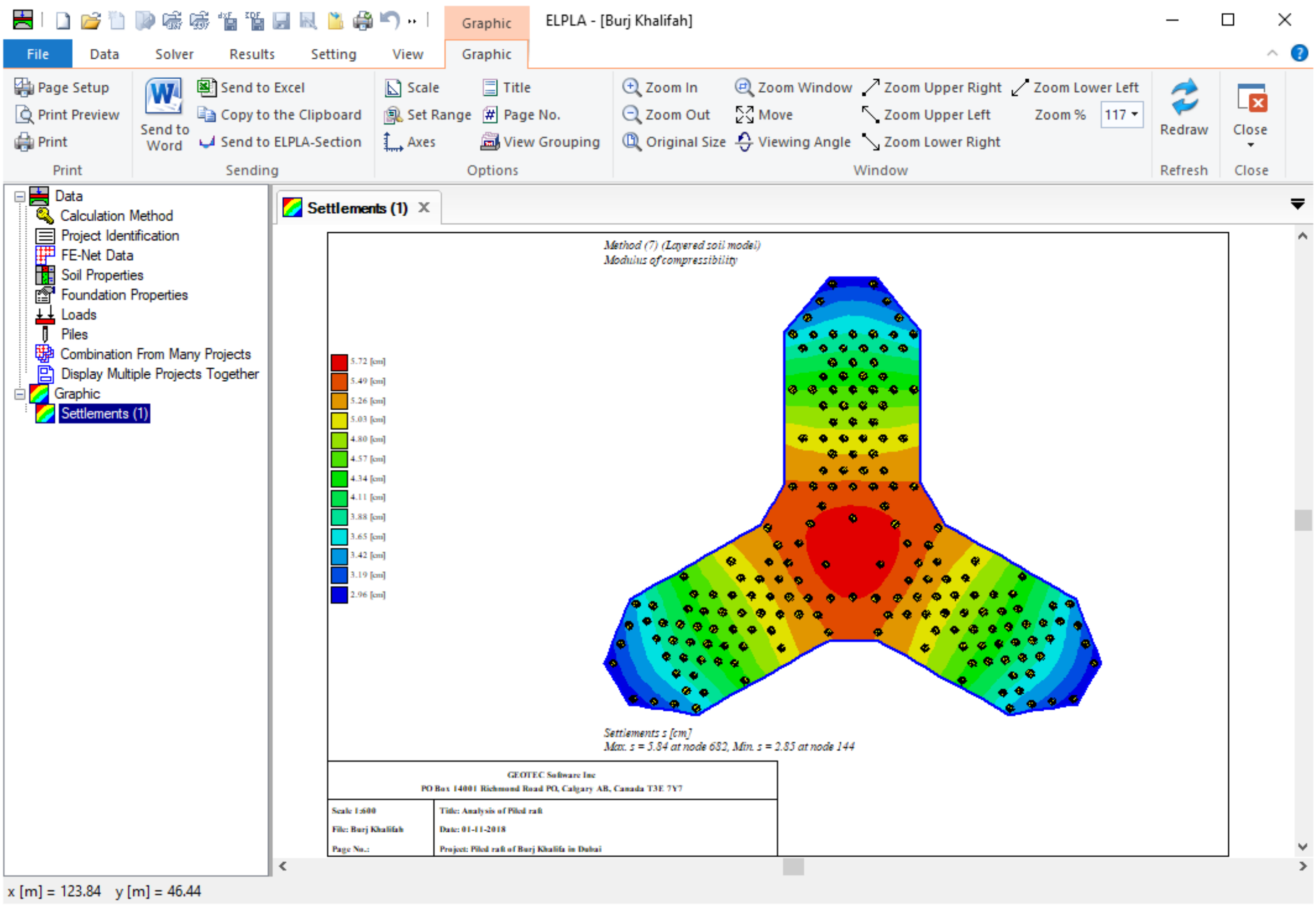


Figure 8 "Analysis of Burj Khalifah foundation" displacements

4.4 Analysis of grid

The analysis of grid problems is available in *ELPLA* (Figure 9).

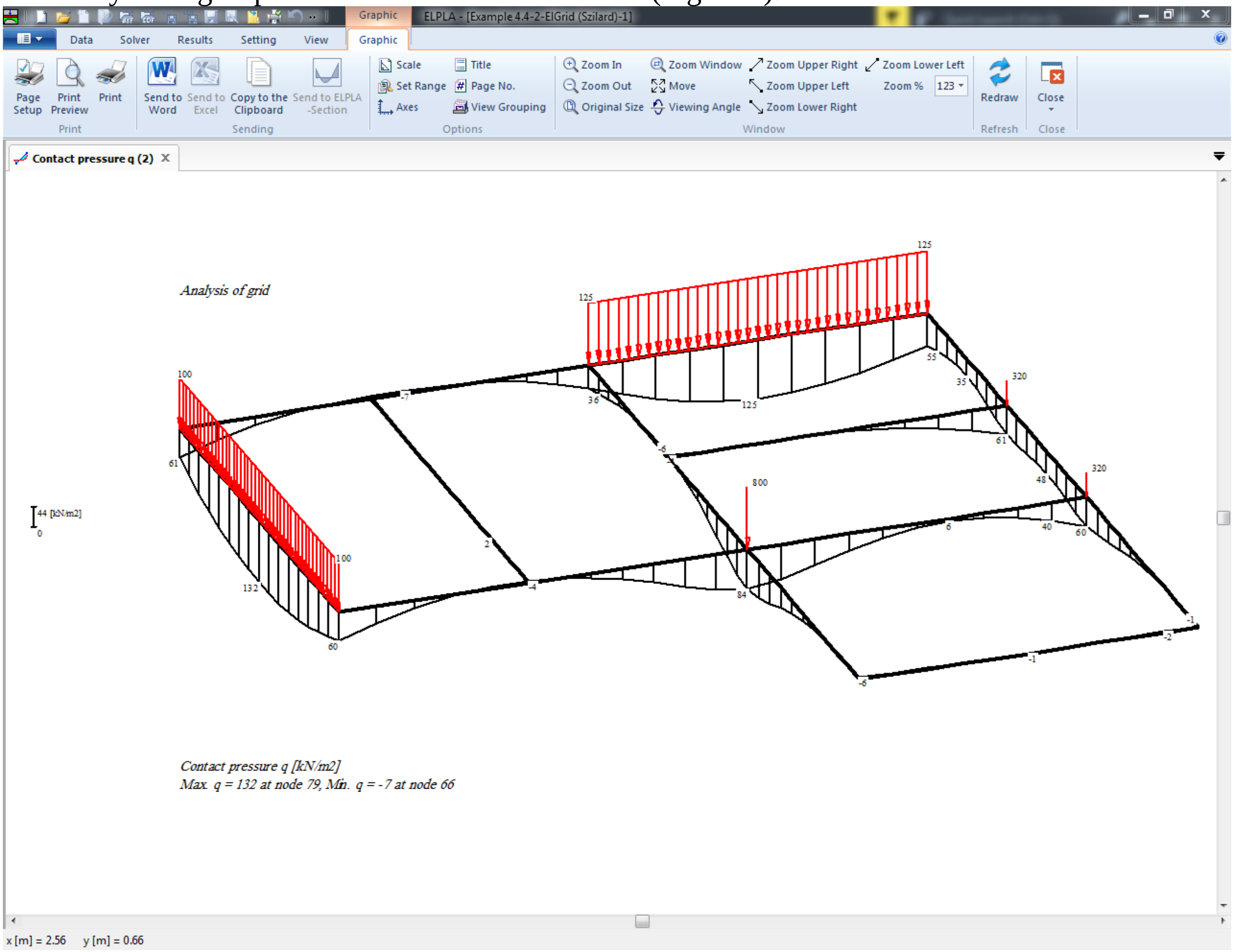


Figure 9 "Analysis of grid"

4.5 Analysis of system of many slab foundations

In the "Analysis type" Form, if the option "Analysis of system of many slab foundations" is chosen, the following Dialog box in Figure 10 appears. Three different numerical calculation methods are considered for the analysis of a system of slab foundations, flexible, elastic, or rigid. For the analysis of a system of many slab foundations, the project filenames (slab foundations) are required. Figure 11.

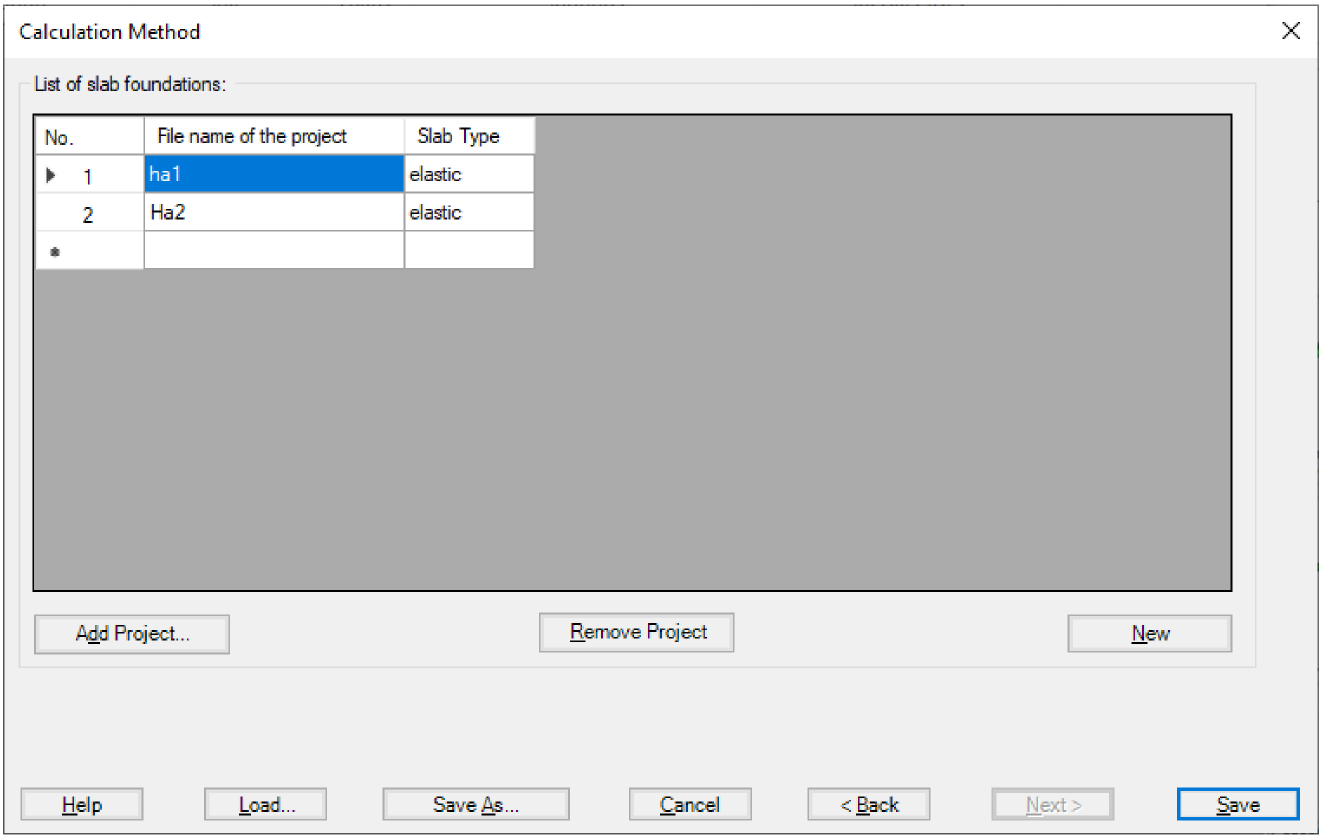


Figure 10 "Analysis of system of many slab foundations" Dialog box

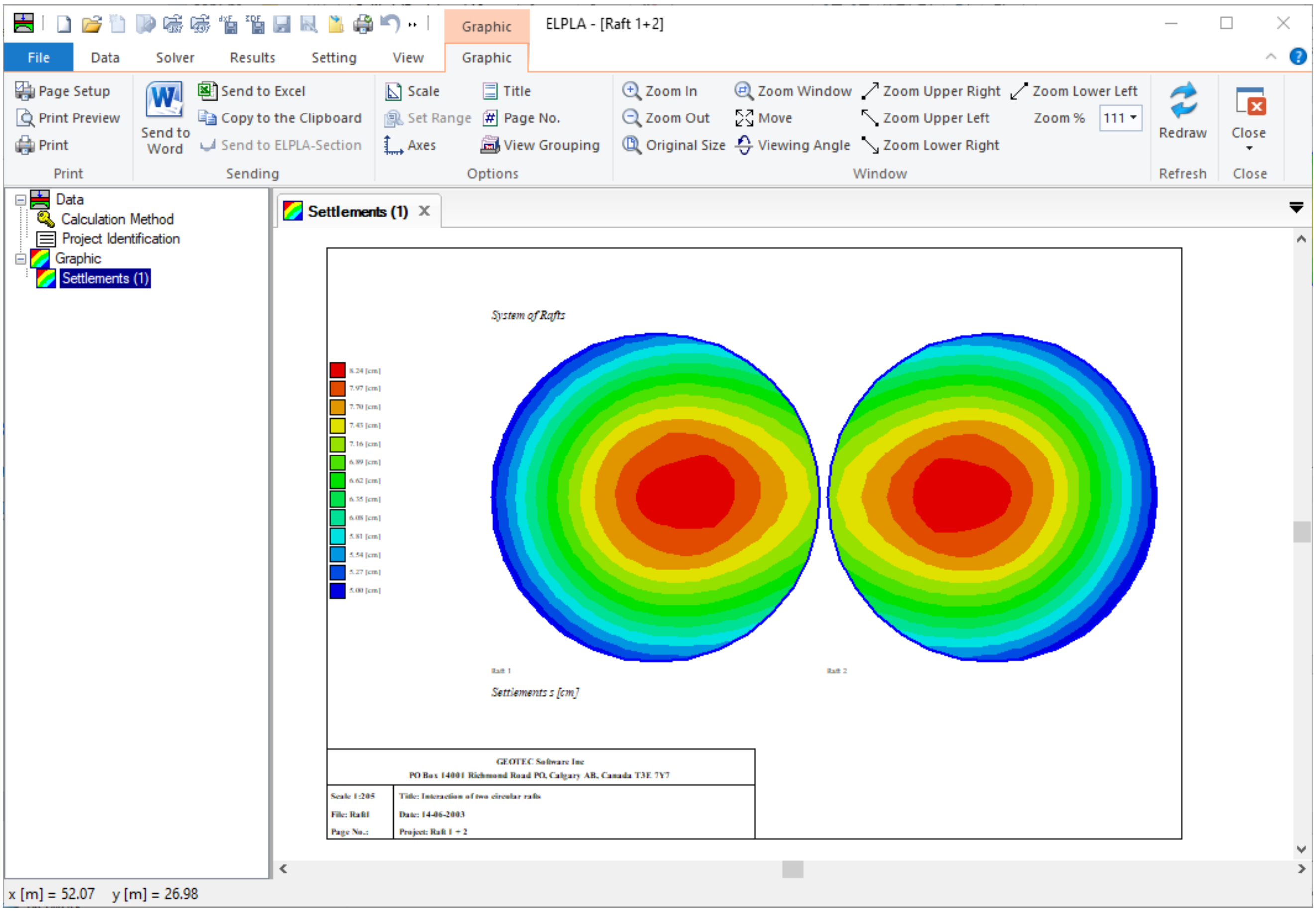


Figure 11 "Analysis of system of many slab foundations" displacements

4.6 Analysis of plane stress

The analysis of plane stress problems is available in *ELPLA* (Figure 12).

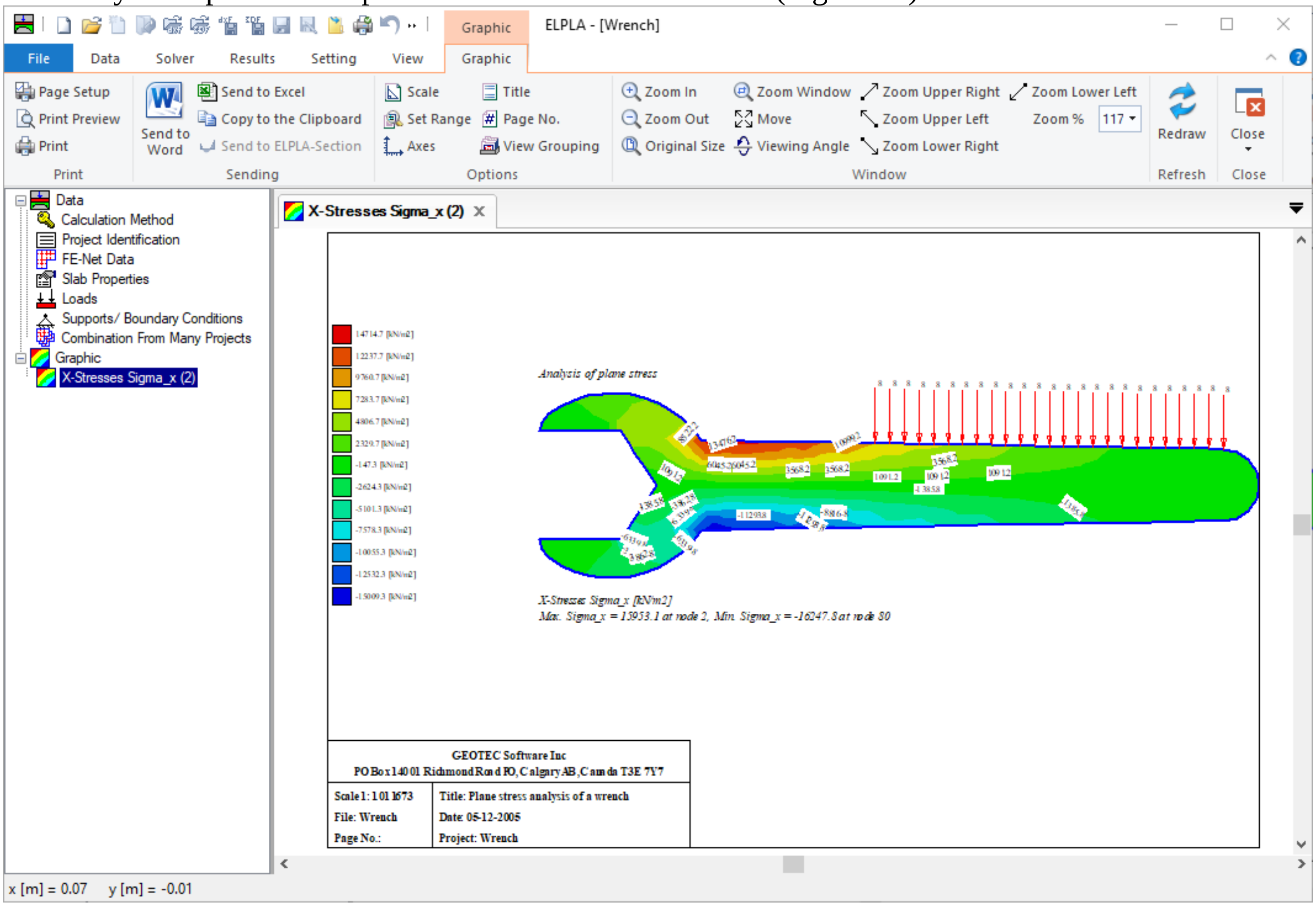


Figure 12 Analysis of plane stress

4.7 Two-Dimensional Frame Problems

The analysis of Two-Dimensional frame problems is available in *ELPLA* (Figure 13).

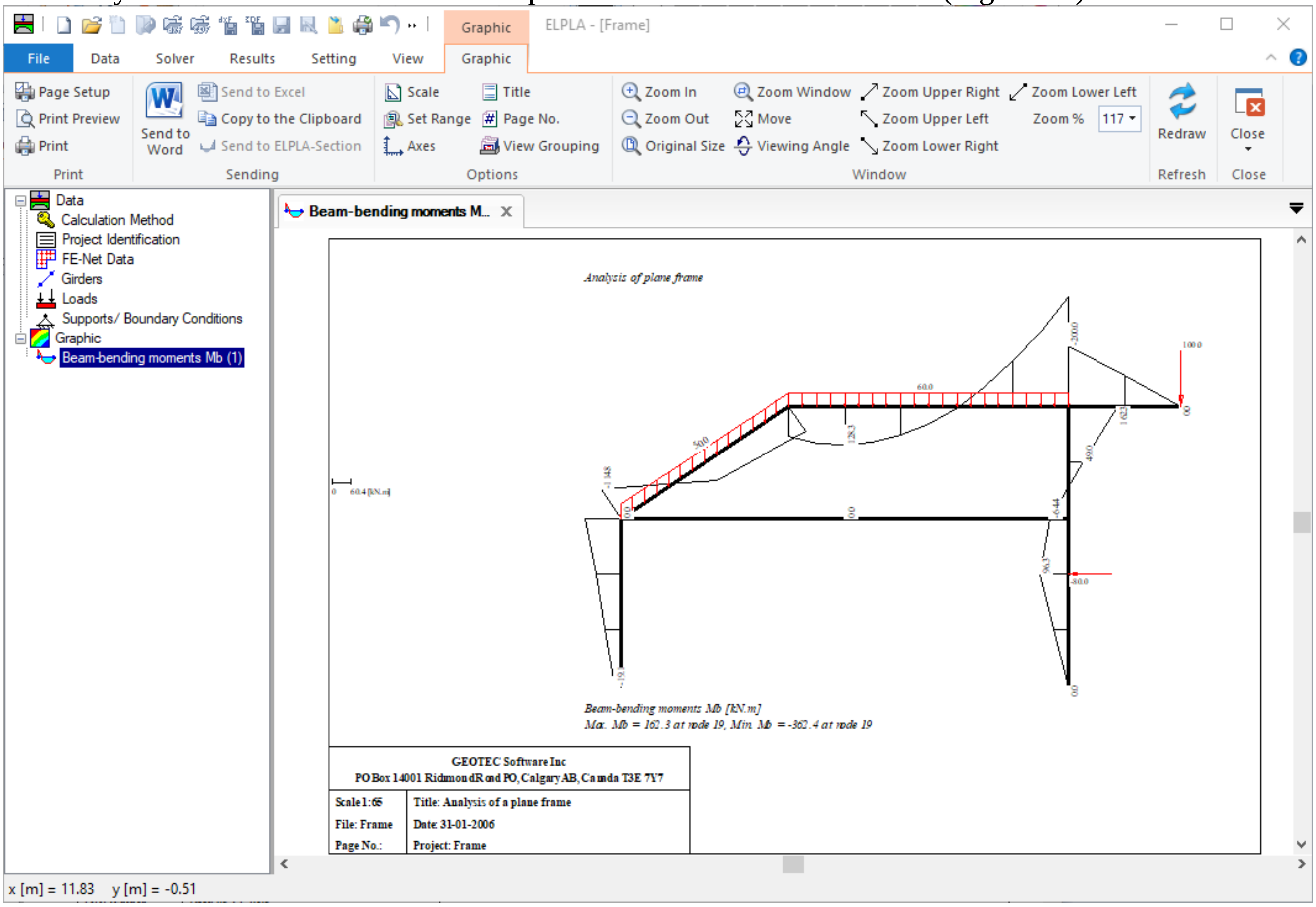


Figure 13 Analysis of plane frame

4.8 Dynamic analysis of structures

It is possible to determine Eigenmodes and Eigenvectors due to free vibration for the following structures:

1. Beams
2. Trusses
3. Grids
4. Space frames
5. Shear walls
6. Floor slabs
7. Axisymmetric solids

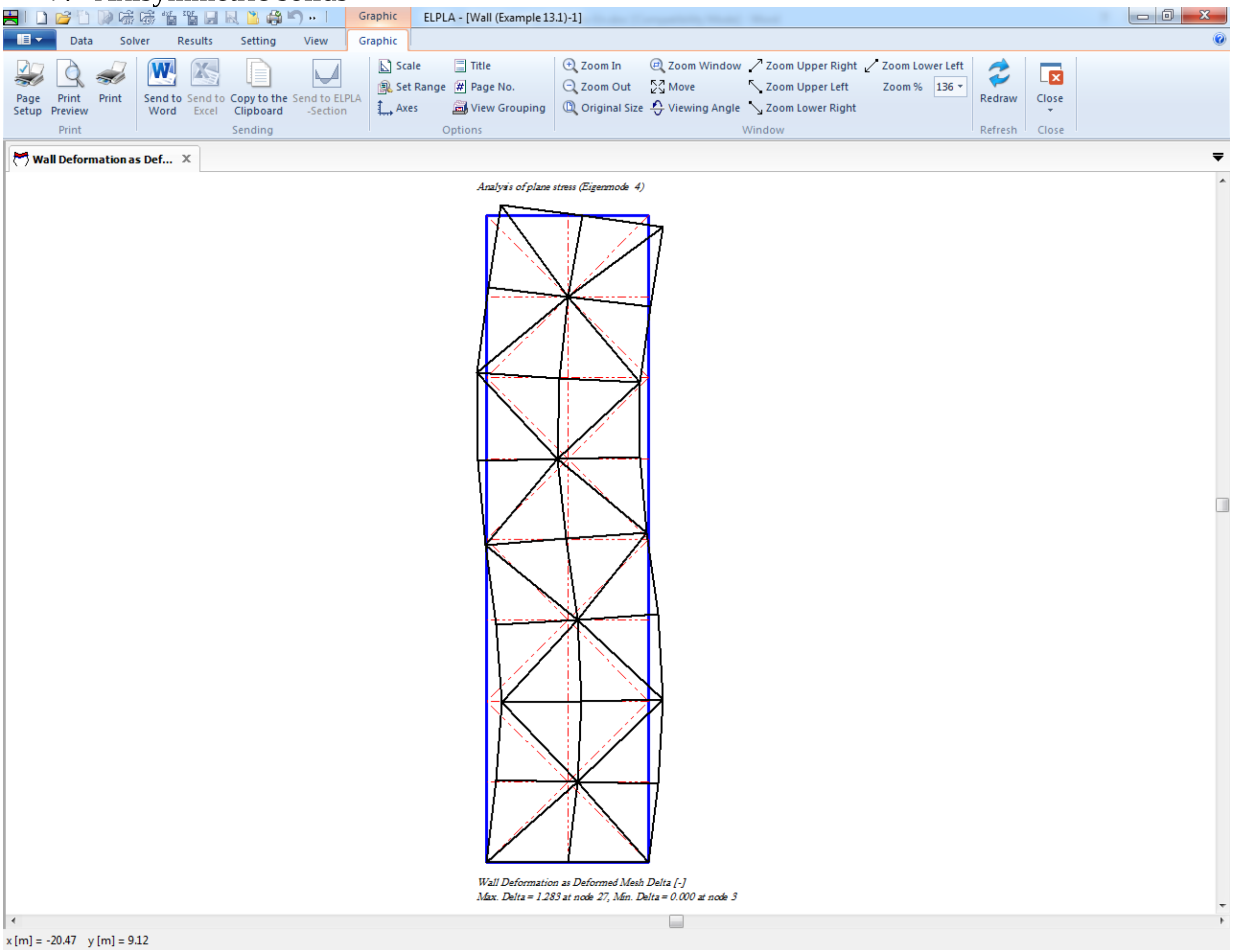


Figure 14 Dynamic analysis of shear wall

4.9 System symmetry

The next step is to define the "System symmetry", Figure 15. In this step, select system symmetry and click "Next" button to go to the next step.

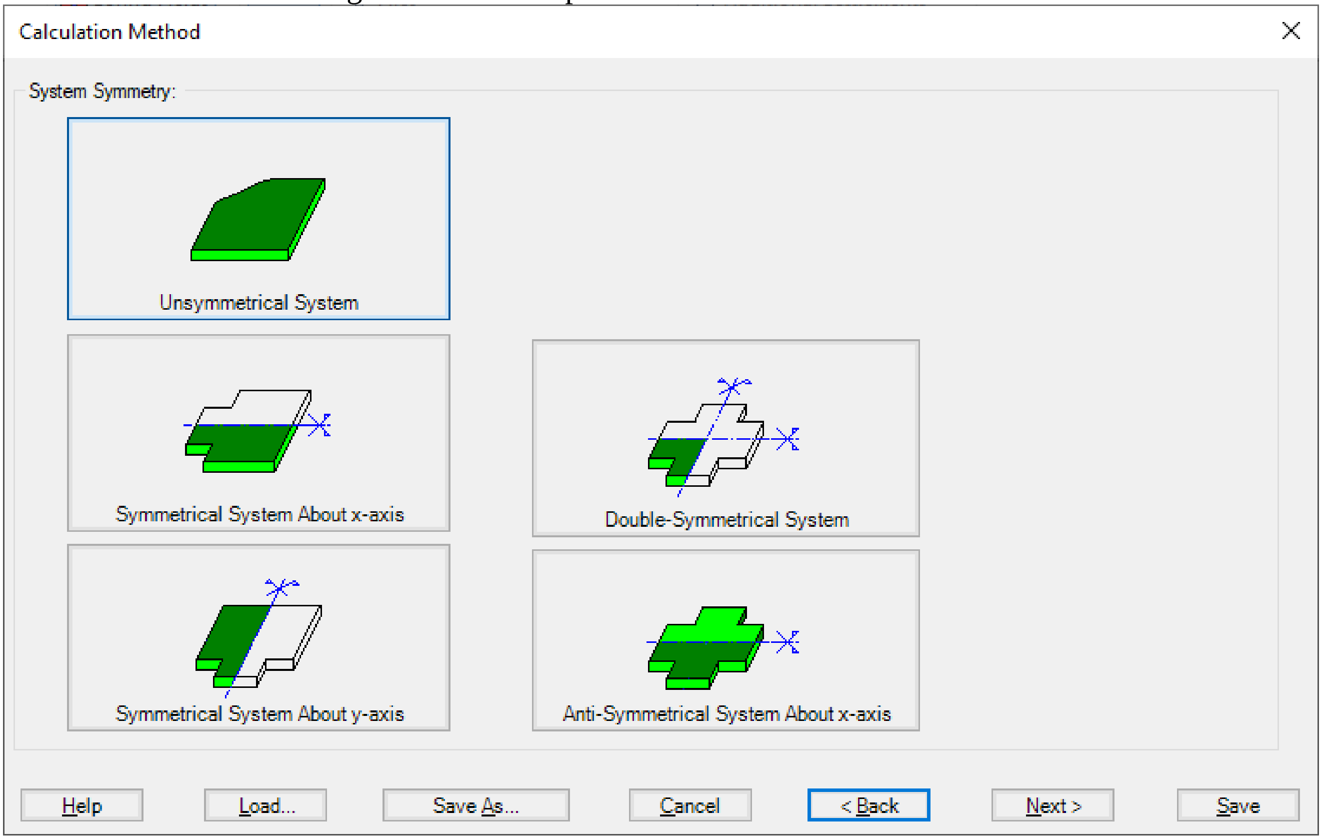


Figure 15 "System Symmetry" Form

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 a simple symmetrical or anti-symmetric slab system, the data are defined according to Figure 16, in which only the lower half slab is considered for symmetry about the x-axis while only the left half slab is considered for symmetry about the y-axis.

Figure 16 Simple symmetrical slab system

By defining the project data for a double symmetrical slab system, the data are defined according to Figure 17. Only the left lower quarter of the slab is considered.

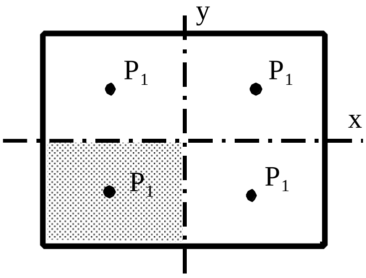


Figure 17 Double symmetrical loaded slab

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 anti-symmetrical loading, Figure 18. The symmetrical cases are available for all calculation methods 1 to 9. The anti-symmetric case is only possible for calculation methods 4 to 8.

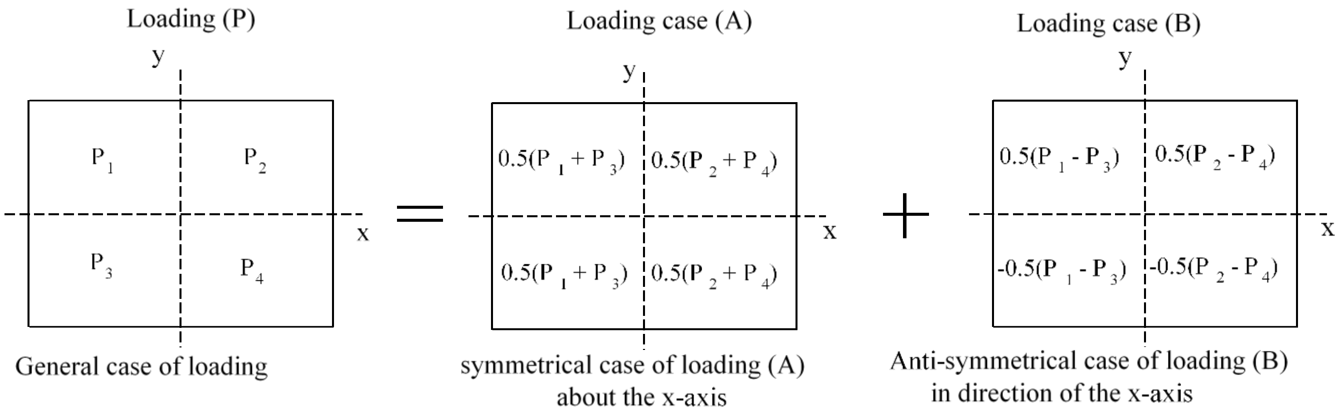


Figure 18 General case of loading by symmetrical and anti-symmetric loading

4.10 Options

Some options are available in *ELPLA* such as the concrete design of sections, additional springs, supports, girders, piles, limit depth, nonlinear subsoil model, determining displacements, stresses, and strains 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 19 check the options that you want to consider in the analysis.

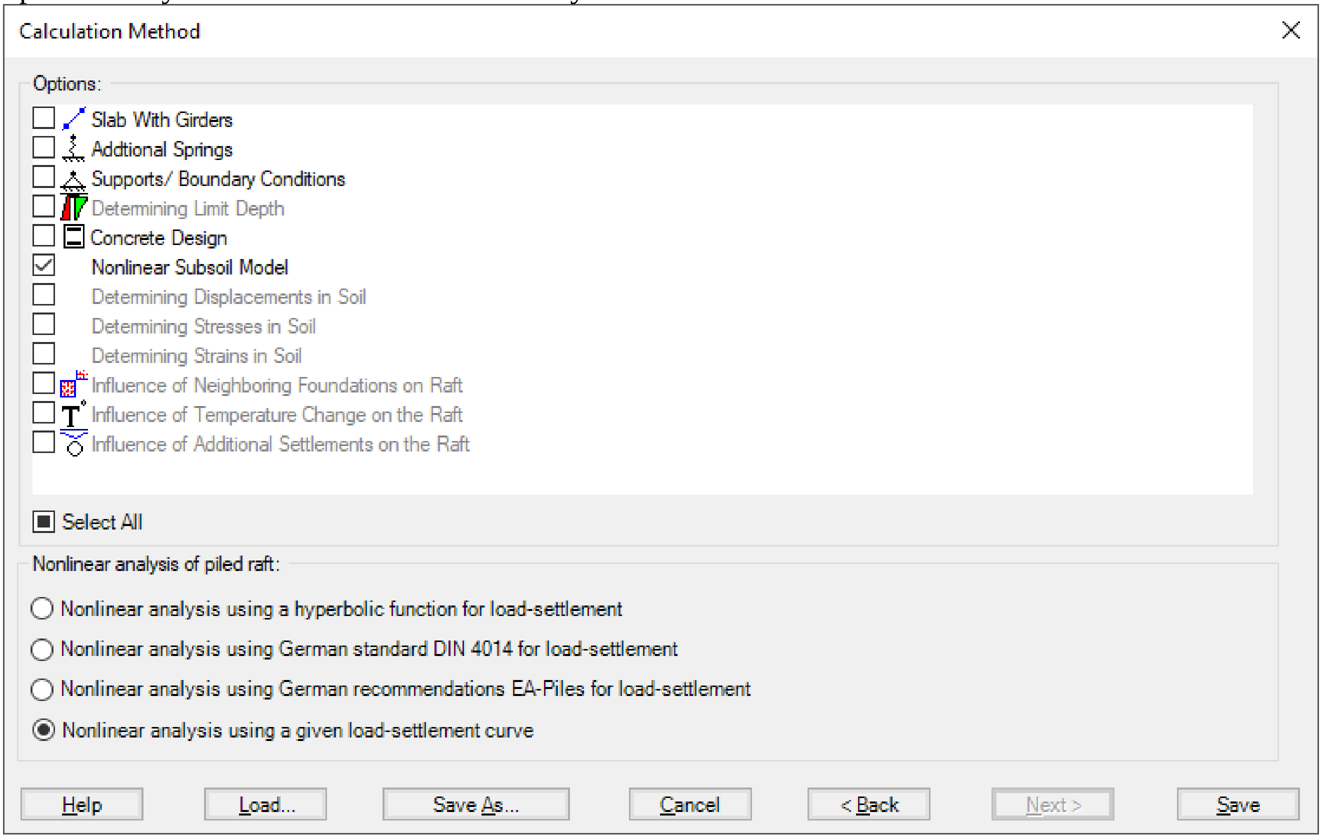


Figure 19 "Options" Check box

5 Soil models

In ELPLA, different numerical methods with 3 soil models are considered for analyzing raft/piled raft 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)

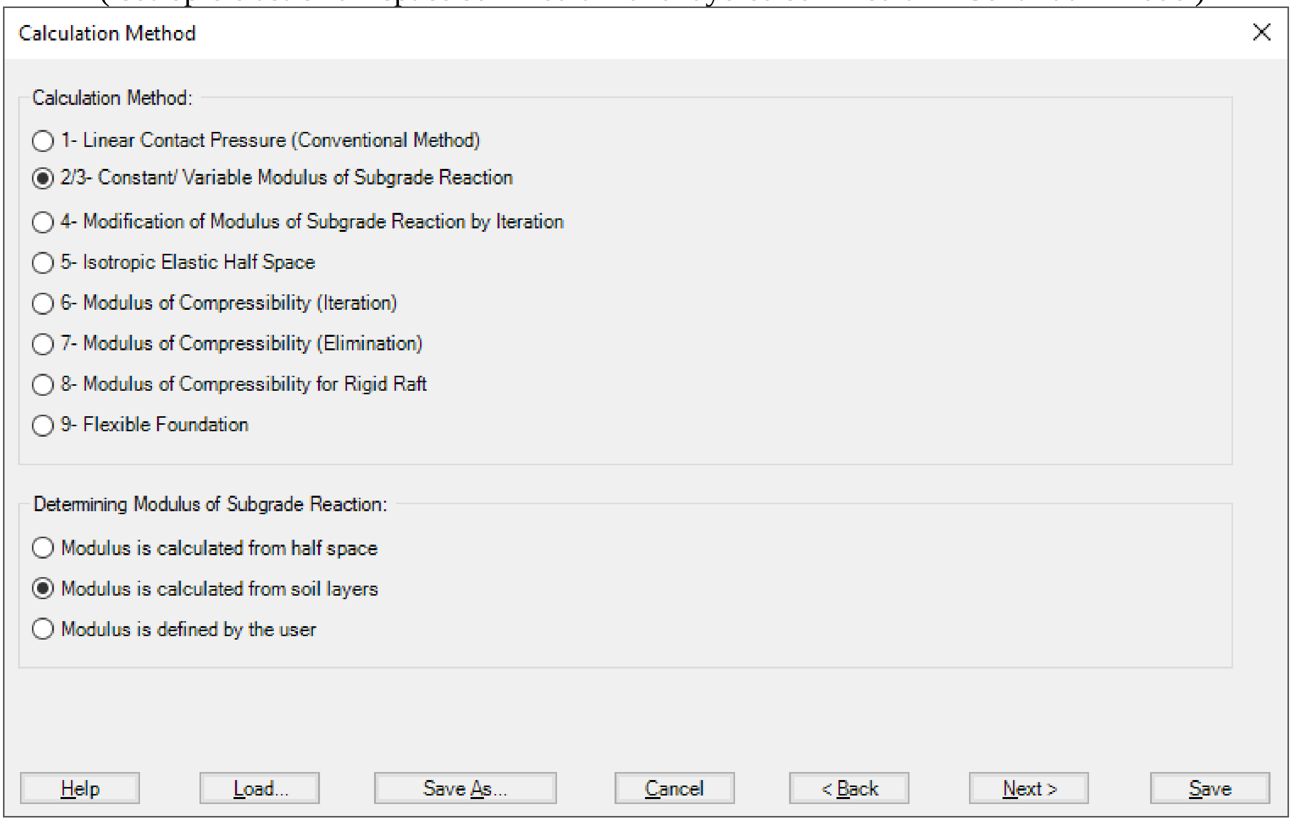


Figure 20 "Calculation methods" Dialog box

5.1 Determination of modulus of subgrade reactions

In *ELPLA*, it is possible to analyze the raft by the modulus of subgrade reaction method in which the modulus can be determined in 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.

It is possible to perform linear and nonlinear analyses of the soil models.

5.2 Simple Assumption Model

There is no interaction between the subsoil and the foundation for the 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 be defined for calculation method 1 (Linear Contact Pressure method), the following Dialog box of Figure 21 appears.

If the water table is located above the foundation, the foundation will be exposed to additional negative pressure. In the Dialog box of Figure 21 define the groundwater depth under the ground surface G_w in order to take the effect of groundwater pressure in the analysis.

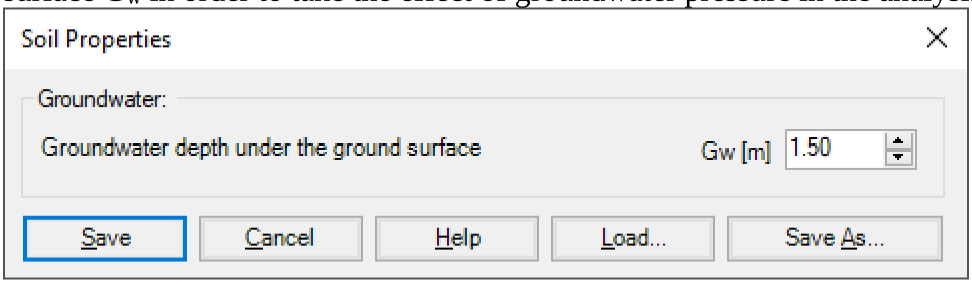


Figure 21 "Soil properties" Dialog box (method 1)

5.3 *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 be defined 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 22).

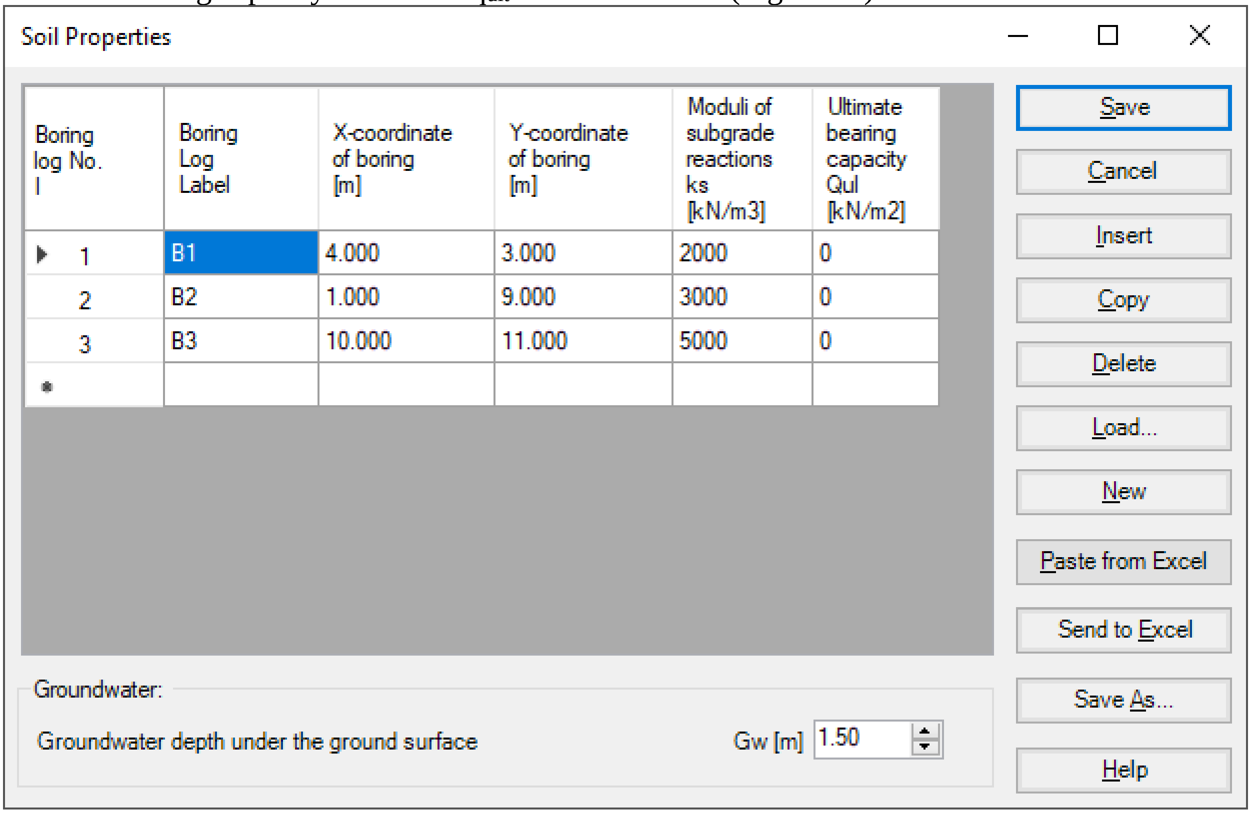


Figure 22 "Soil properties" Dialog box (methods 2 and 3)

5.4 Isotropic Elastic Half-Space Model

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

In the Dialog box, 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.

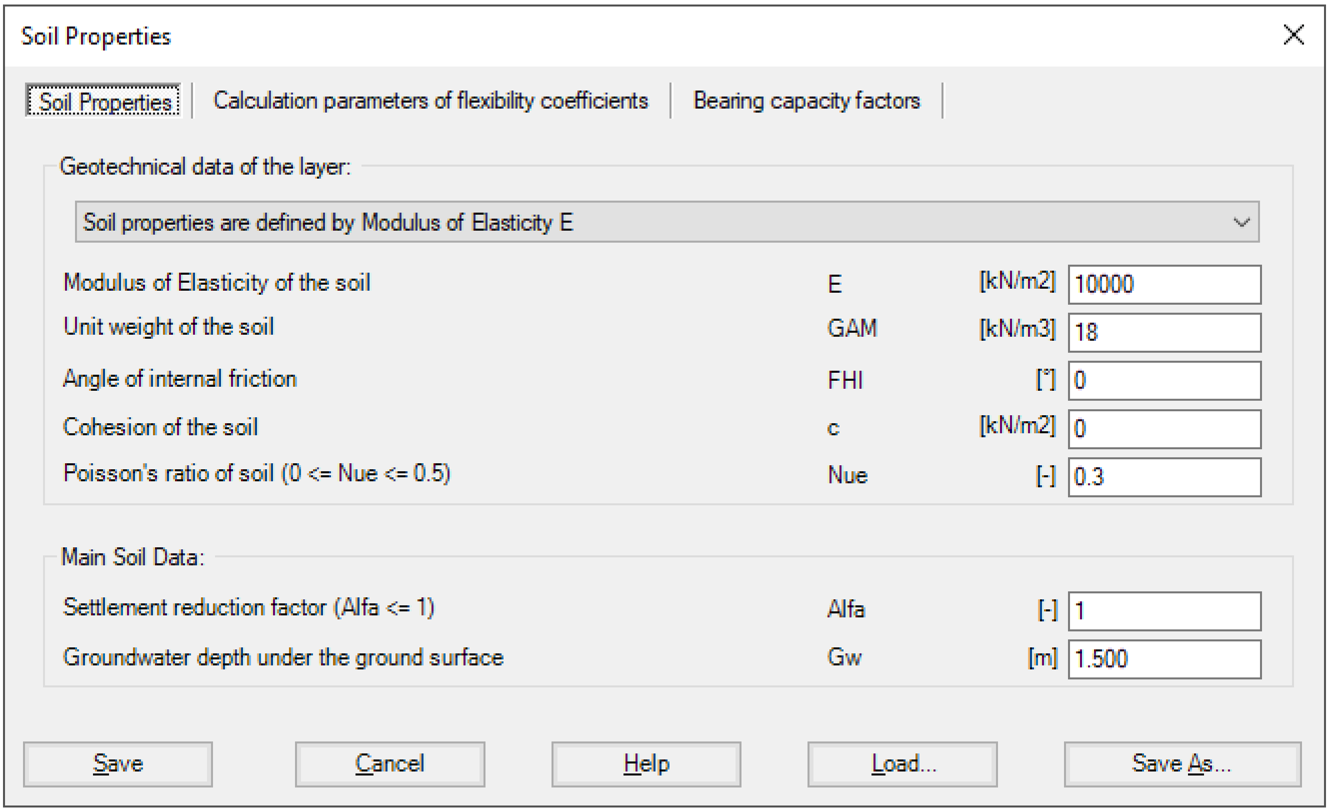


Figure 23 "Soil properties" Dialog box (methods 2 and 5)

Settlement reduction factor

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

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

Flexibility coefficients for interior nodes

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

It is possible to determine the flexibility coefficient of the interior node due to a uniform load at that node (Figure 24):

- at the characteristic point of the loaded area, where rigid settlement is equal to the 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

The earlier version of ELPLA determines flexibility coefficients for both interior and exterior nodes by assuming uniform loaded areas on these nodes. This assumption requires 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 24. In this way, the program does not follow the principle of superposition in the analysis, making it much faster than the old analysis. The new method 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.

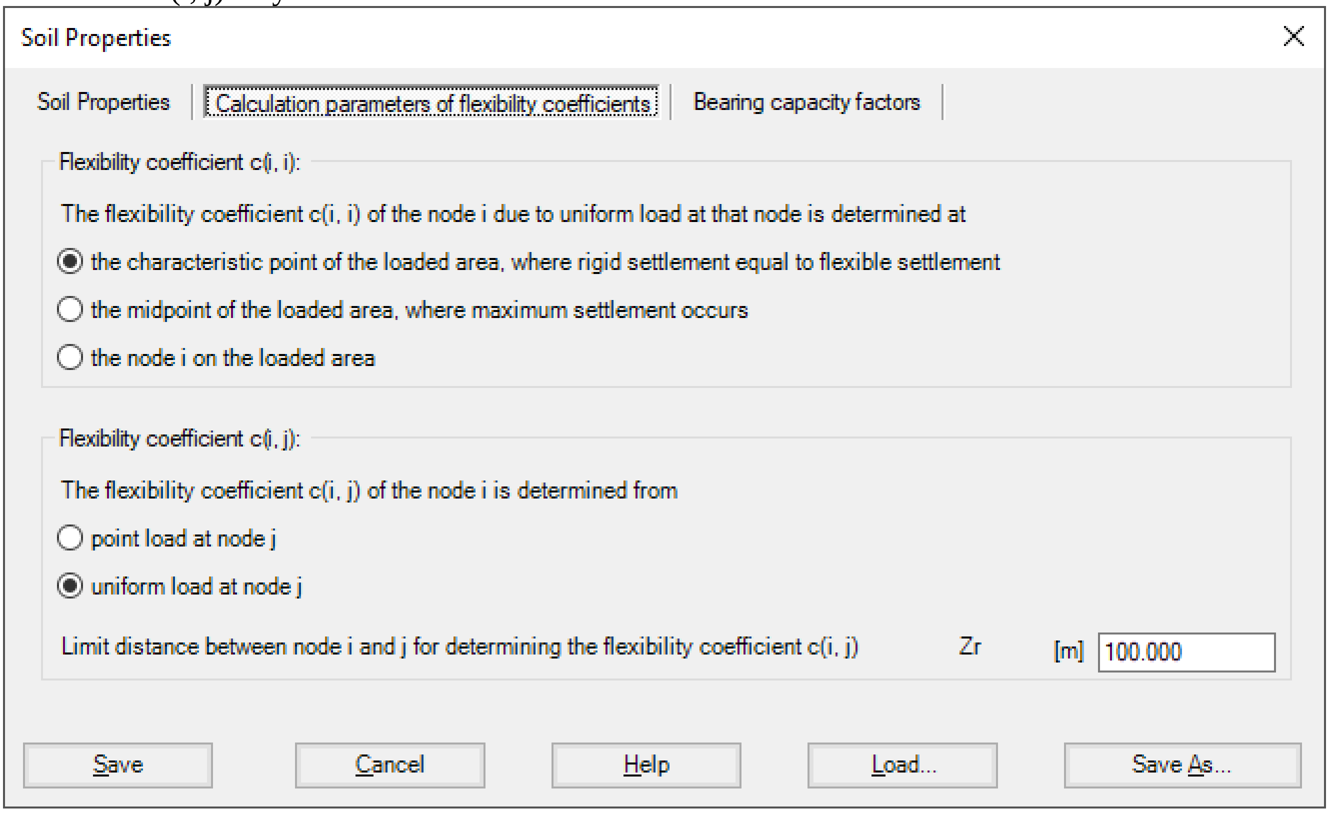


Figure 24 "Calculations parameters of flexibility coefficients" Dialog box (methods 2 and 5)

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 25):

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

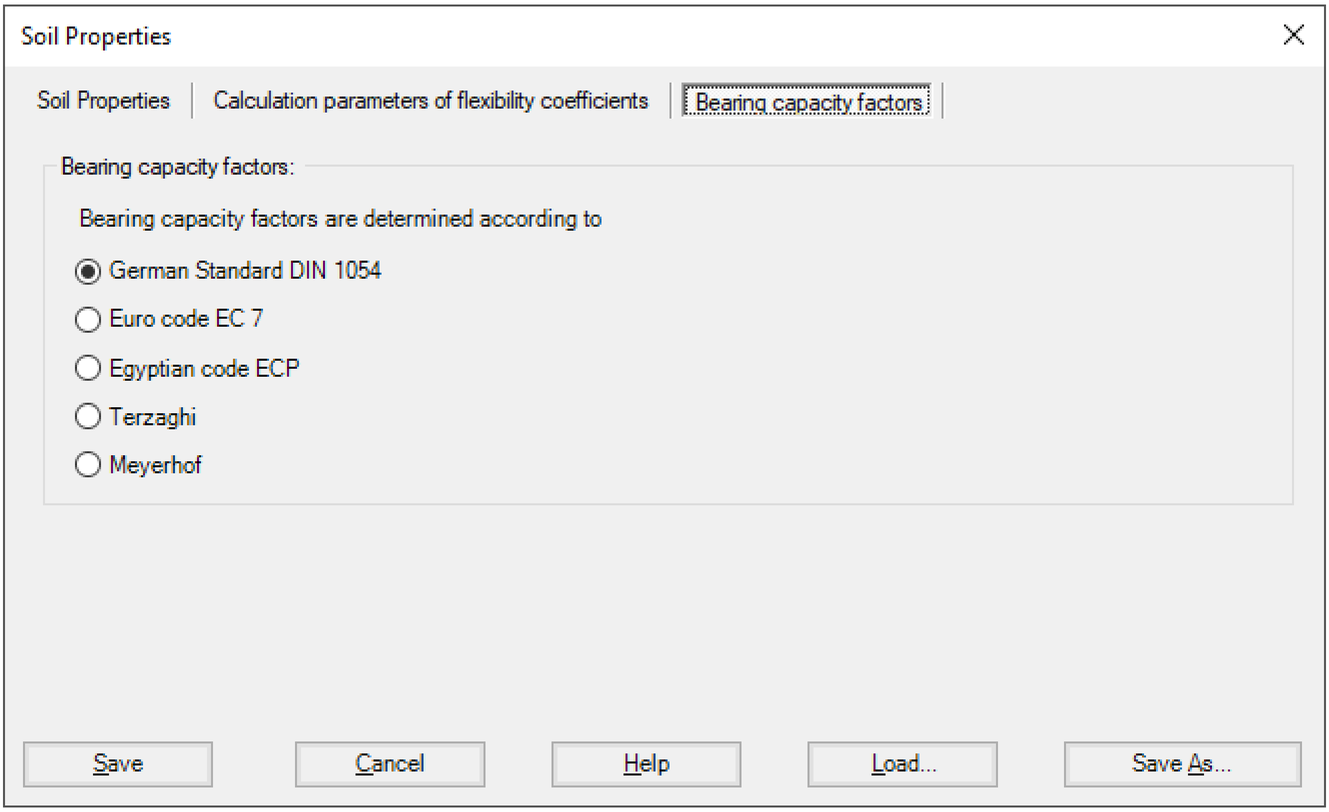


Figure 25 "Bearing Capacity factors" Dialog box (methods 2 and 5)

5.5 Layered Soil Model

A layered soil model is used in the analysis of the calculation methods shown in Table 1. When soil properties are required for one of the calculation methods shown in this Table, the following Soil Properties Tab View appears (Figure 26).

Table 1 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 an elastic raft on layered soil medium Solving system of linear equations by iteration, Layered soil medium - Continuum model
7	Modulus of compressibility method for an elastic raft on layered soil medium Solving system of linear equations by elimination, Layered soil medium - Continuum model
8	Modulus of compressibility method for a rigid raft on layered soil medium, Layered soil medium - Continuum model
9	Modulus of compressibility method for a flexible foundation on layered soil medium, Layered soil medium - Continuum model

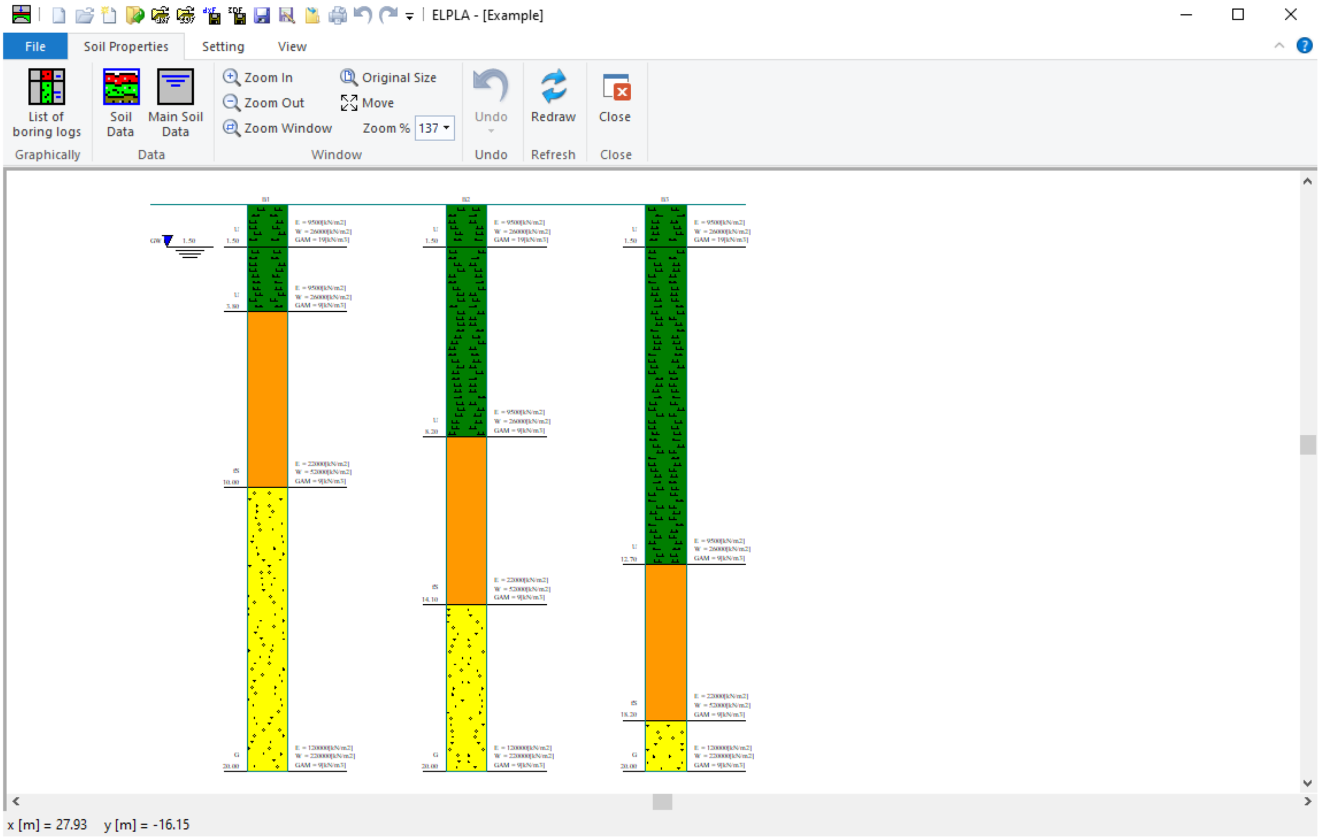
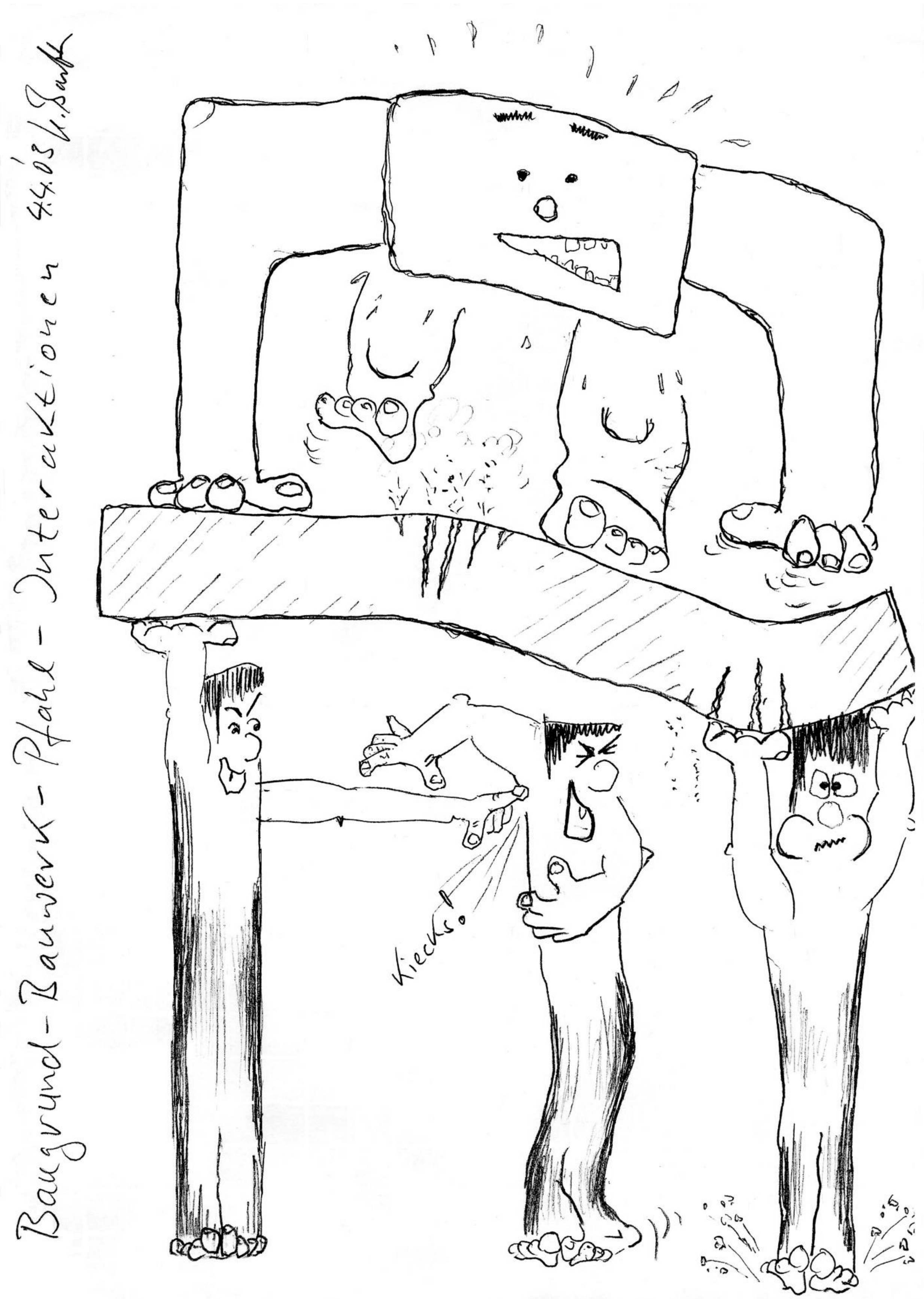


Figure 26 "Soil properties" Tab view

6 Piled raft



ELPLA is a powerful tool for analyzing piled-raft foundations. Today, nearly every engineering office has its own computer programs for the analysis and design of piled rafts. Furthermore, most of the available programs under Windows are user-friendly and give very excellent output graphics with colors. Consequently, theoretically a secretary not an engineer can use them. But the problem here is how can man control the data and check the results.

Many practical problems are available which are analyzed in detail by using the program *ELPLA*. It is important for the engineer to be familiar with this information when carrying out a computer analysis of piled rafts. An understanding of these concepts will be of great benefit in carrying out the computer analysis, resolving difficulties, and judging the acceptability of the results. Three familiar types of subsoil models (standard models) for piled raft analyses are considered. The models are the Simple Assumption Model, *Winkler*'s Model, and Continuum Model. In the analysis, rafts are treated as elastic or rigid. The Finite Element Method was used to analyze the raft, in which plate bending elements represent the raft according to the two-dimensional nature of the foundation.

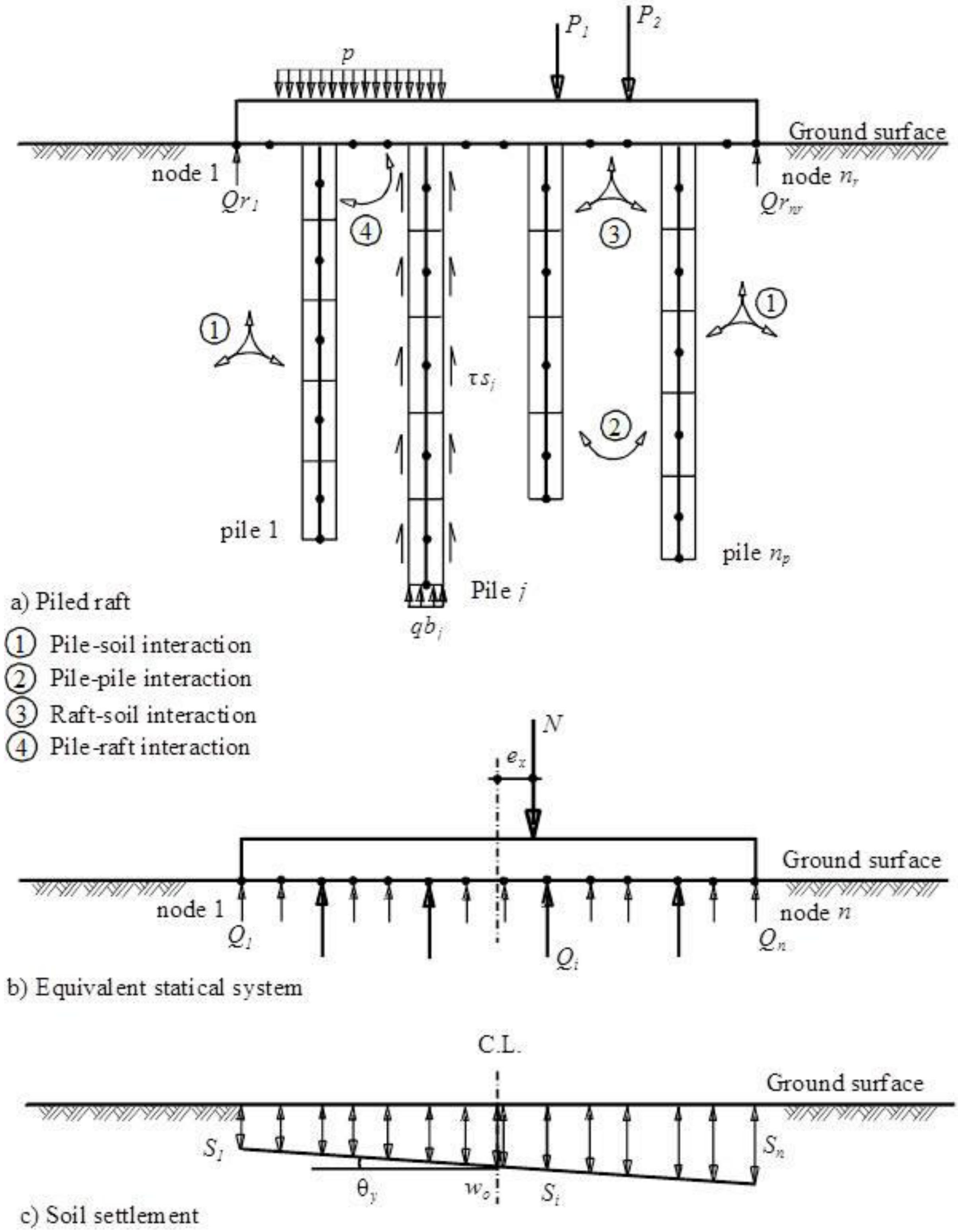
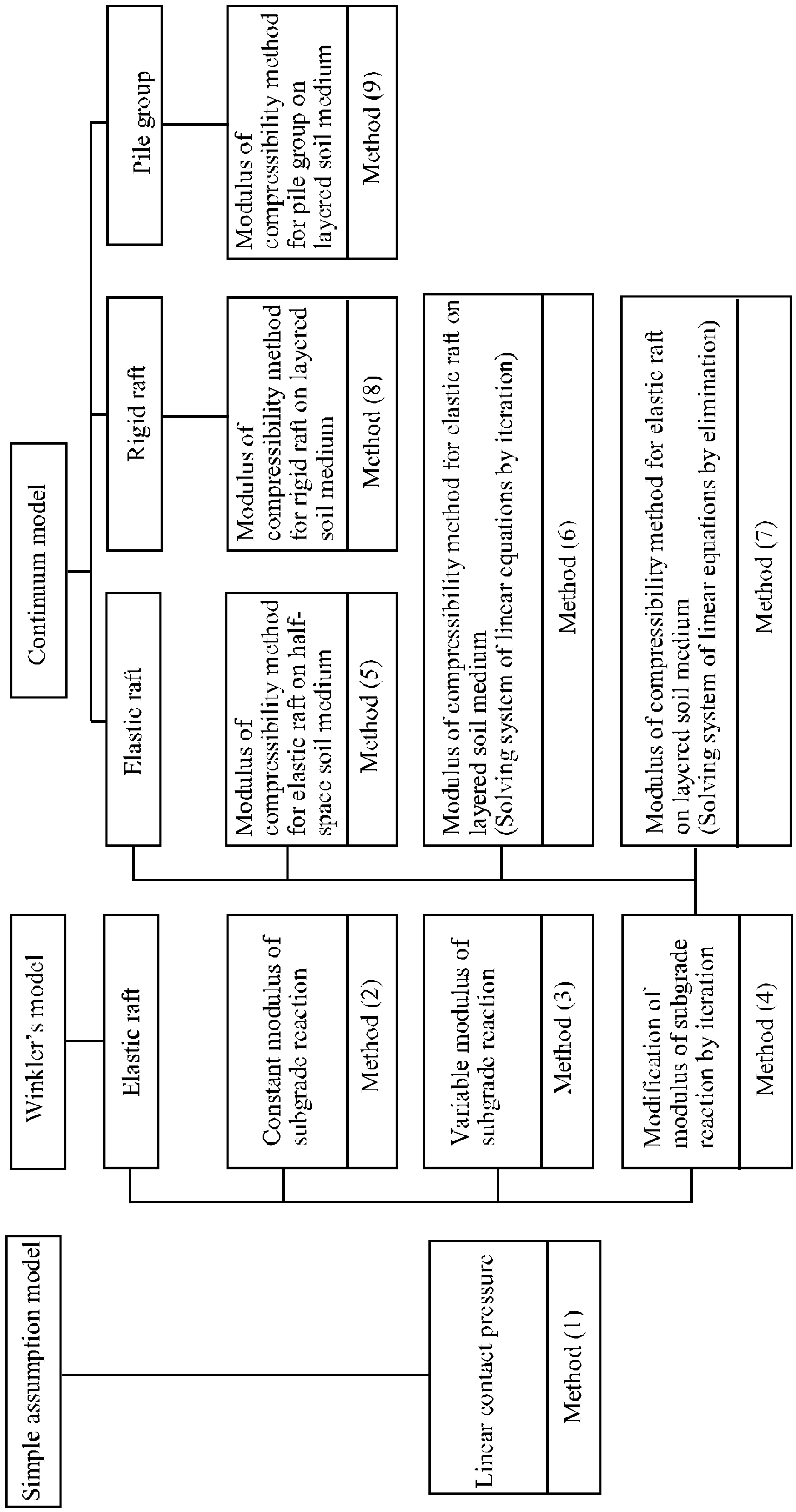


Figure 27 Pile-raft-soil interactions



Simple assumption model
Linear contact pressure
Method (1)
Winkler's model
Elastic raft
Constant modulus of subgrade reaction
Method (2)
Variable modulus of subgrade reaction
Method (3)
Modification of modulus of subgrade reaction by iteration
Method (4)
Continuum model
Elastic raft
Modulus of compressibility method for elastic raft on half-space soil medium
Method (5)
Modulus of compressibility method for elastic raft on layered soil medium (Solving system of linear equations by iteration)
Method (6)
Modulus of compressibility method for elastic raft on layered soil medium (Solving system of linear equations by elimination)
Method (7)
Rigid raft
Modulus of compressibility method for rigid raft on layered soil medium
Method (8)
Pile group
Modulus of compressibility method for pile group on layered soil medium
Method (9)

6.1 Simple Assumption Model

The simple assumption model does not consider the interaction between the foundation and the soil. The model assumes a linear distribution of contact pressures beneath the foundation.

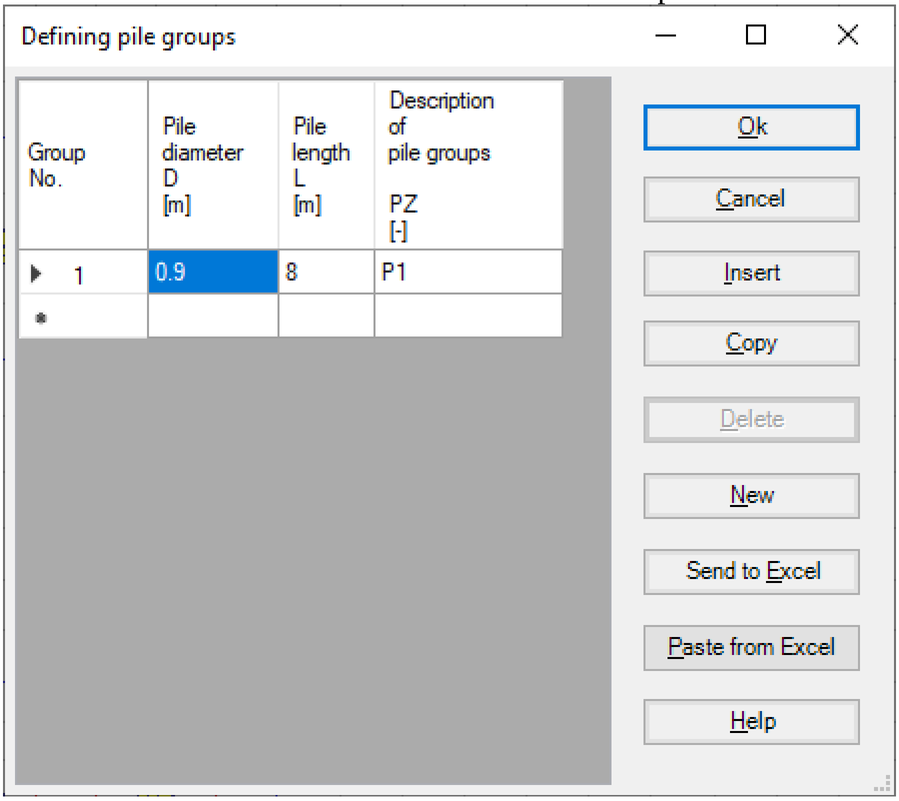


Figure 28 Defining pile groups for Simple Assumption Model

6.2 *Winkler's* Model

Winkler's model is the oldest and simplest one that considers the interaction between the foundation and the soil. The model represents the soil or piles as elastic springs. It is available in the two methods of Constant and Variable Modulus of Subgrade Reaction (methods 2 and 3).

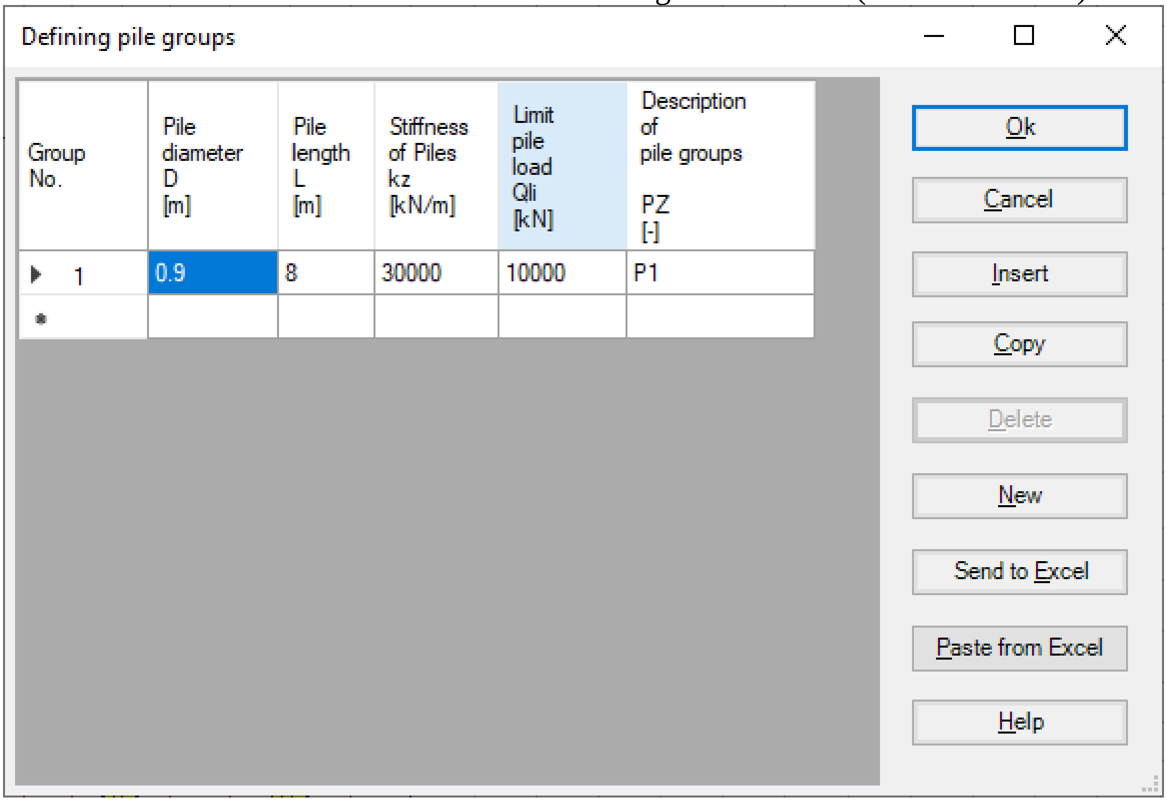


Figure 29 Defining pile groups for Winkler's Model

6.3 Continuum Model

The continuum model is the complicated one. The model considers also the interaction between all foundation elements and soil. It represents the soil as a layered continuum medium or isotropic elastic half-space soil medium.

Although the continuum model provides a better physical representation of the supporting soil, it has remained unfamiliar, because of its mathematical difficulties where an application of this model requires extensive calculations. Practical application for this model is only possible if a computer program or appropriate tables or charts are available. These tables and charts are limited to certain problems.

A general computerized mathematical solution based on Finite elements-method was developed to represent an analysis for pile foundations on the real subsoil model. The solution can analyze foundations of any shape considering holes within the foundation and the interaction of external foundations.

6.3.1 Nonlinear analysis using hyperbolic function Model

In this model, all forces acting on the raft will be transmitted nonlinearly on the piles using the hyperbolic function for Load-settlement.

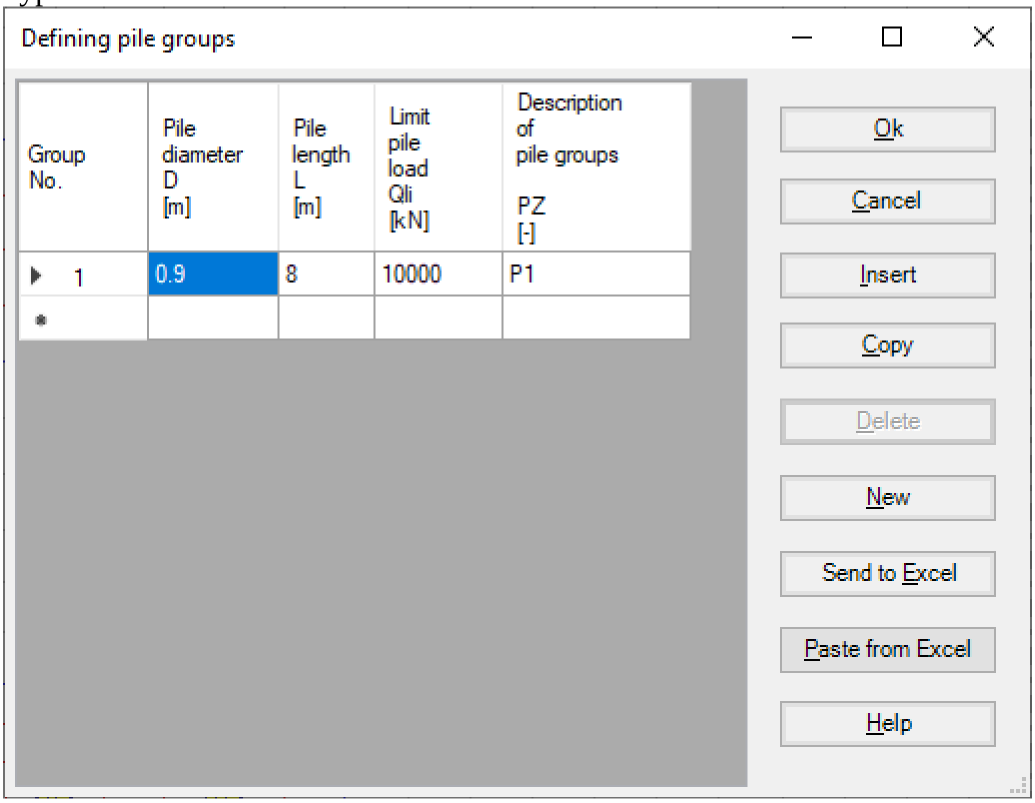


Figure 30 "Defining pile groups" Nonlinear analysis using the hyperbolic function for load-settlement

6.3.2 Nonlinear analysis using German standard DIN-4014

In this model, all forces acting on the raft will be transmitted nonlinearly on the piles using German standard DIN-4014.

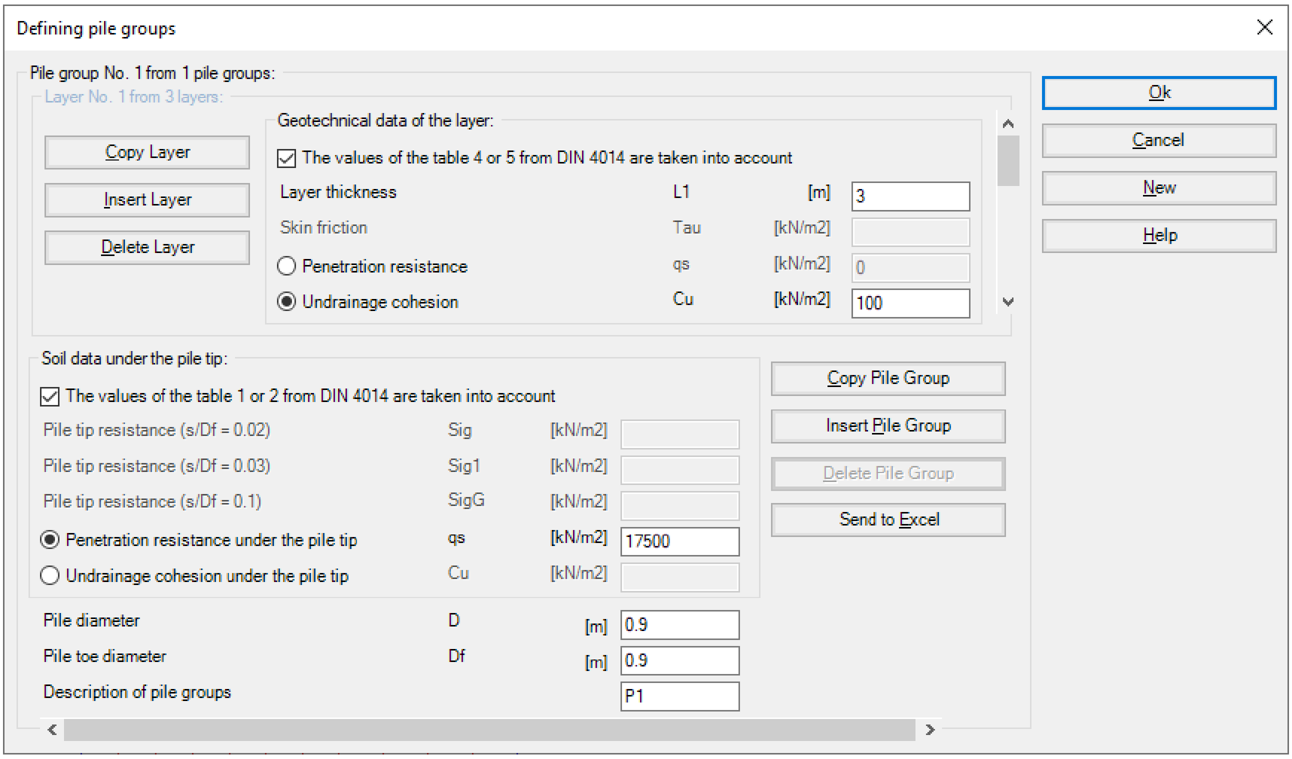


Figure 31 "Defining pile groups" Nonlinear analysis using German standard DIN-4014

6.3.3 Nonlinear analysis using German recommendations EA-Piles

In this model, all forces acting on the raft will be transmitted nonlinearly on the piles using German recommendations EA-Piles.

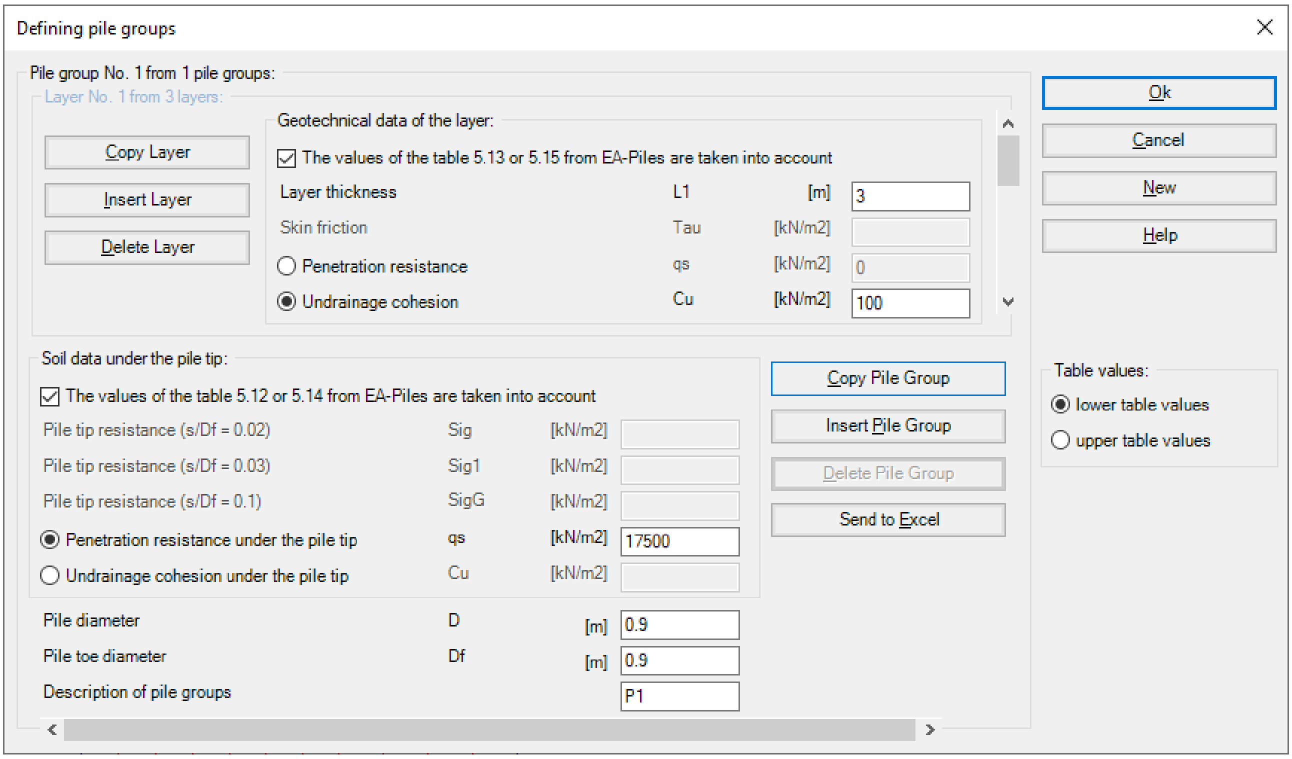


Figure 32 "Defining pile groups" Nonlinear analysis using German recommendations EA-Piles

6.3.4 Nonlinear analysis using a given load-settlement curve

In this model, all forces acting on the raft will be transmitted nonlinearly on the piles using a specified Load-settlement curve. The load-settlement curve for pile group and piled raft can be obtained from two resistance-settlement relations as follows:

a) Skin resistance with tip resistance-settlement, Figure 34.

b) Pile resistance –settlement, Figure 35.

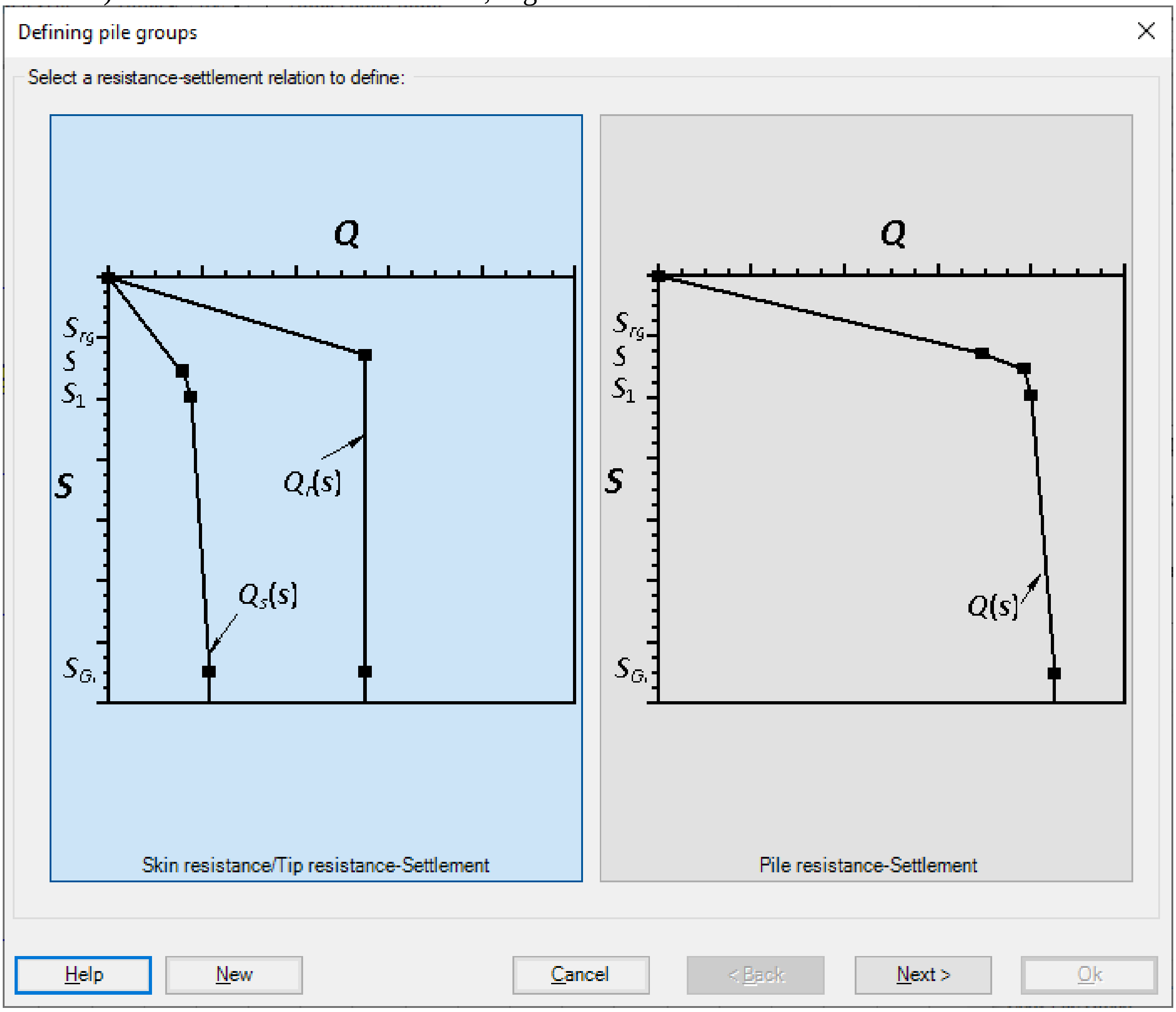


Figure 33 "Defining pile groups" Nonlinear analysis using a given load-settlement curve

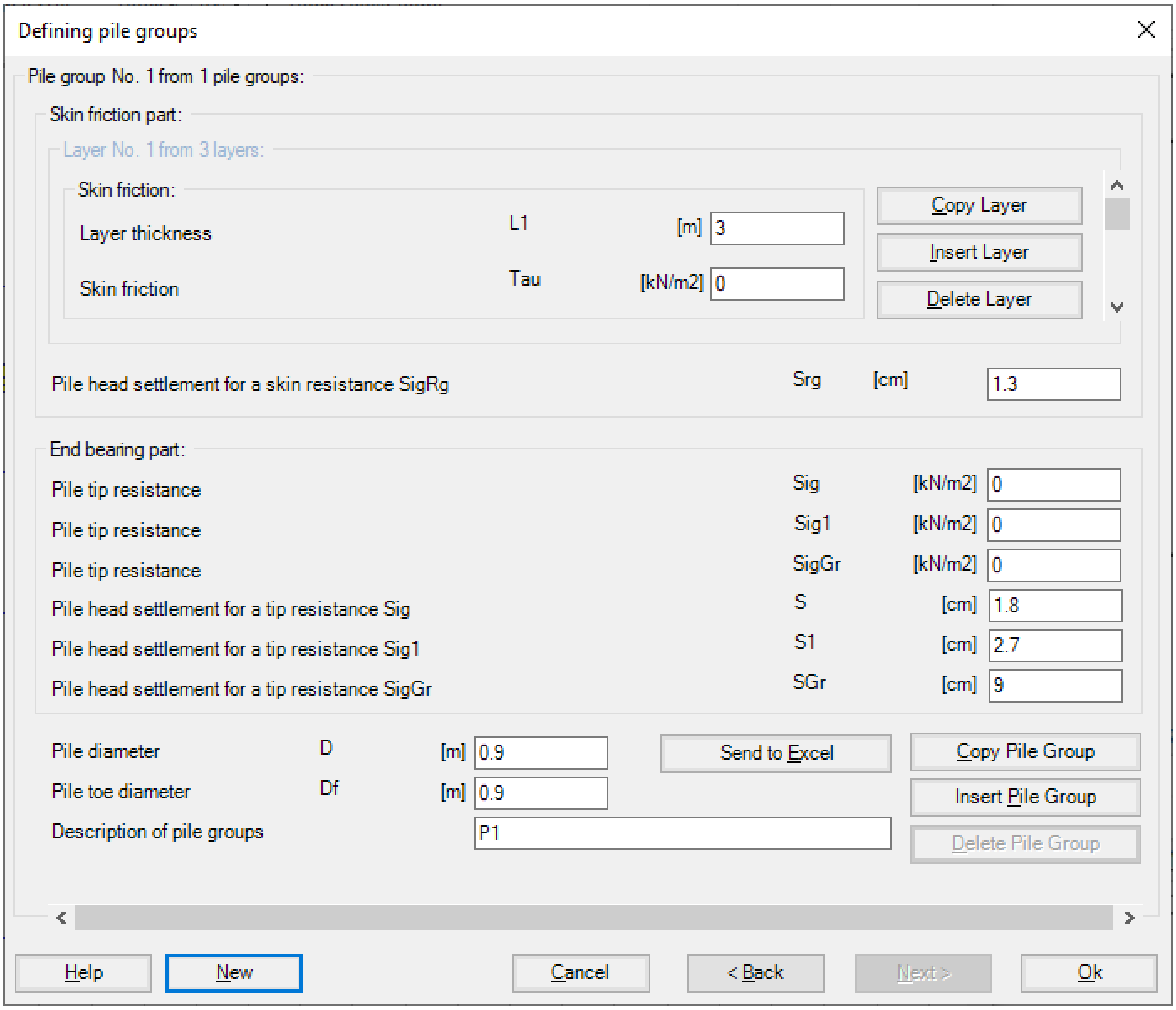


Figure 34 "Defining pile groups" Skin resistance with tip resistance-settlement

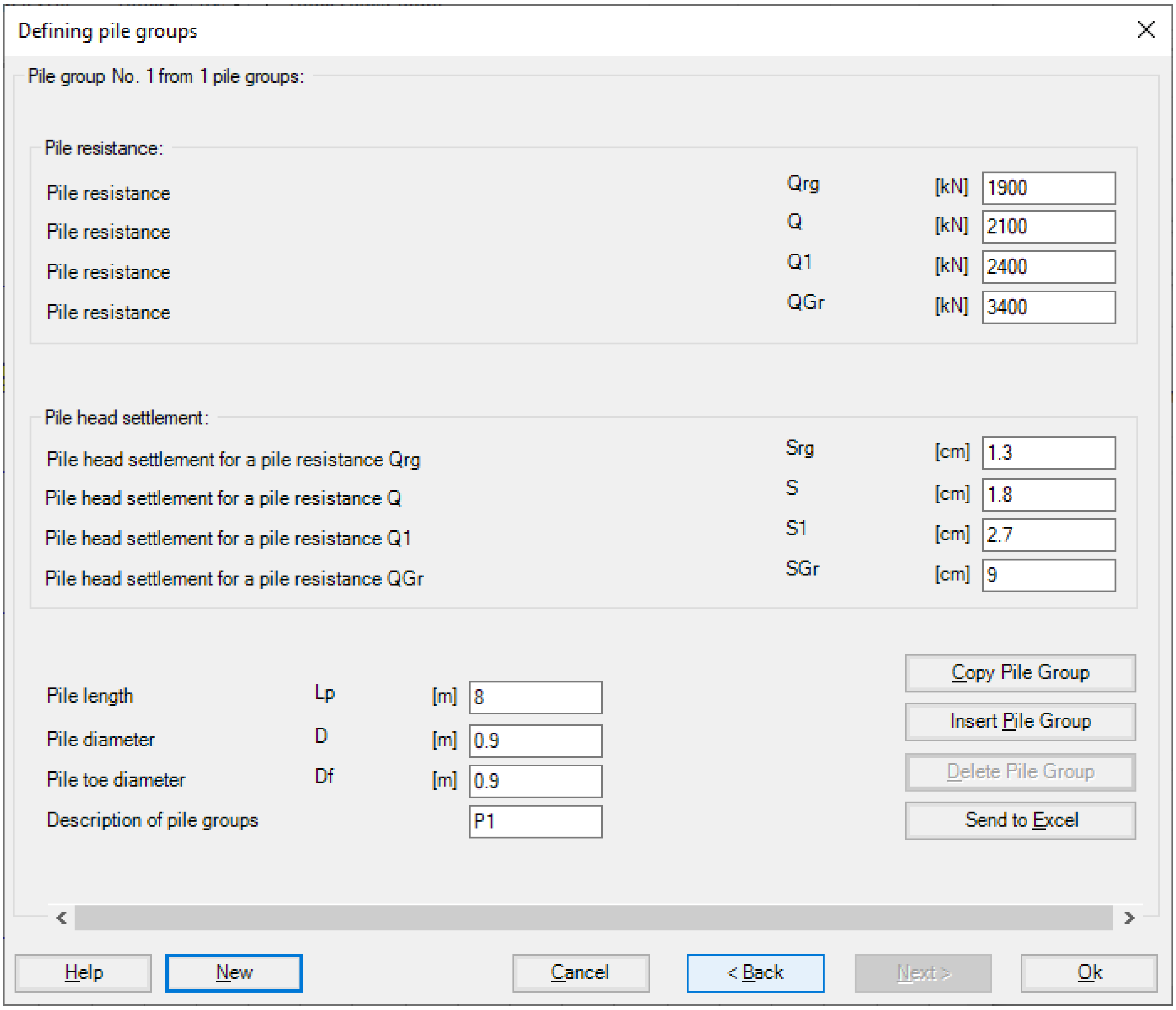


Figure 35 "Defining pile groups" Pile resistance -settlement

7 Geometry and loads

It is possible to consider raft with any arbitrary shape including holes, Figure 36. It is also possible to consider rafts with variable thickness, Figure 37. 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 38.

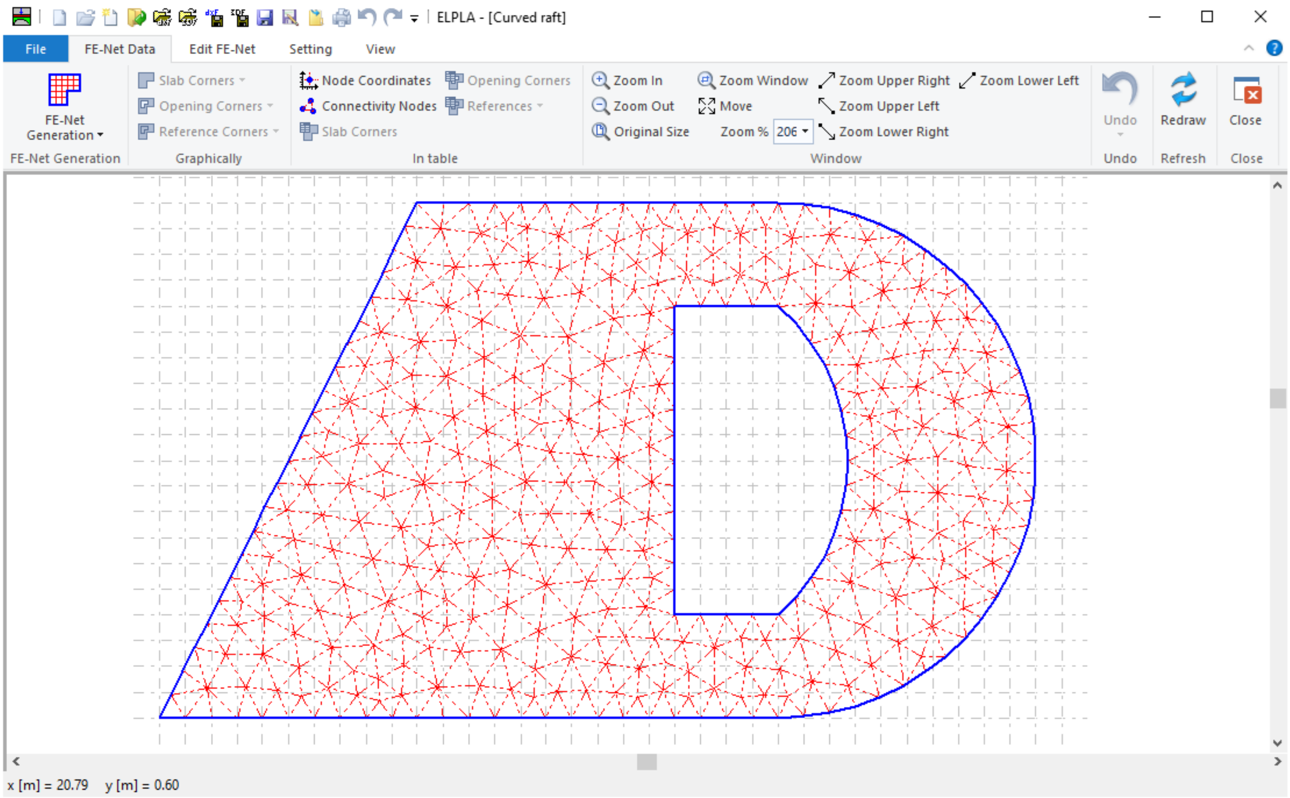


Figure 36 Arbitrary shape of raft with hole

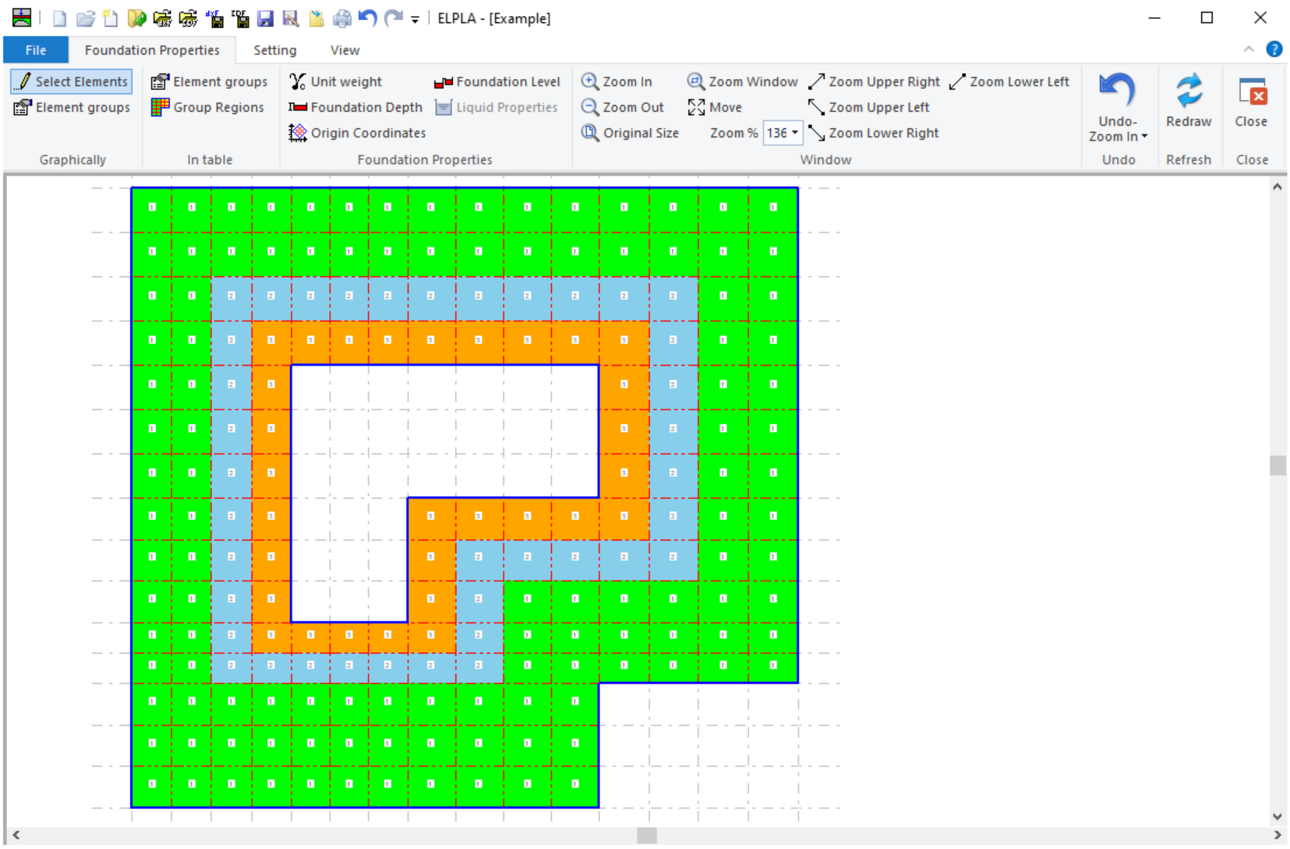


Figure 37 Variable slab thickness

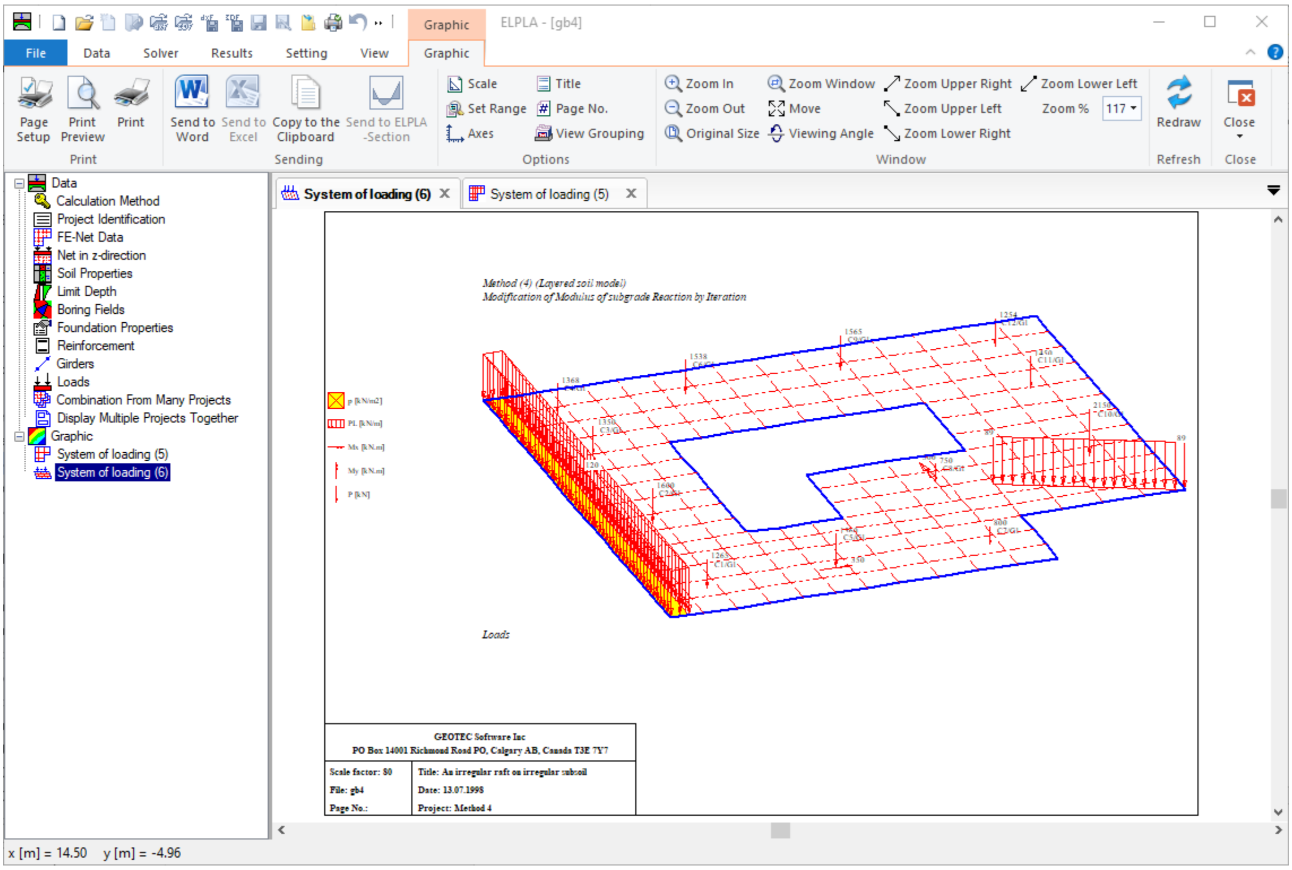


Figure 38 Arbitrary type of loads

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

A new FE-Net is created. When choosing the File - New command and the following templates for different types of FE-Net appear (Figure 39). Using these templates, FE-Net can be generated according to the following features:

- Generating the FE-Net for square, rectangular, and irregular slabs using 6 different types of nets, Figure 40.
- Generating the FE-Net for circular and ring slabs using 15 different types of nets, Figure 41.
- Generating an irregular slab with openings and arc boundaries using a refined mesh, Figure 42.
- It is possible to use combined rectangular, quadratic, and triangular finite elements at the same time for the slab, Figure 43.
- It is possible to create slab corners, opening corners, and reference corners graphically using CAD-style, which lets the definition of the net quick and easy, Figure 44.
- In ELPLA, there are two alternative methods to define slab corners, opening, and references either by Mouse or by editing in a table, Figure 45.
- It is possible to optimize the dimension of FE-Net by making all elements having nearly the same area as possible as by the option "Smoothing the mesh", Figure 46.
- It is possible to direct and arrange all elements on the slab borders (Figure 47). This option is useful to present contact pressures at raft edges in a good manner when analyzing the raft by Continuum model, in which the contact pressures at raft edges are higher than that at the middle.
- 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 48).

- By selecting the "Merge Nets" command from the "Arrange" group, two or more nets (Figure 50) can be combined into a net, Figure 49.
- By selecting the "Split Net" command from the "Arrange" group, the FE-Net (Figure 49) can be divided into two or more nets, Figure 50.
- Reference points, reference lines, reference corners, Reference polylines, and curves (Figure 51) can be imported into *ELPLA* from a DXF file (AutoCAD Drawing Exchange File), Figure 52.
- The user can generate a finite element mesh using the AutoCAD command as "3DFACE", Figure 53. 3D Face can be imported into *ELPLA*, Figure 54.
- An existed FE-Net can be added to the current "FE-Net". This option enables the user to analyze a system of many foundations together in one combined FE-Net, Figure 55.

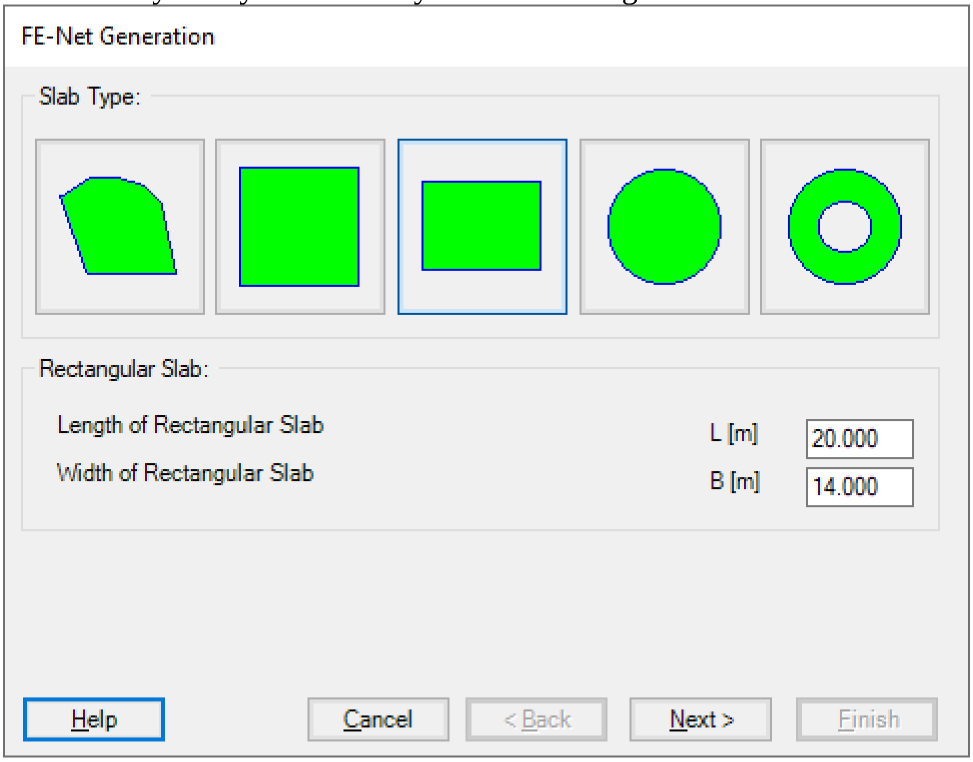


Figure 39 Templates for different types of FE-Net

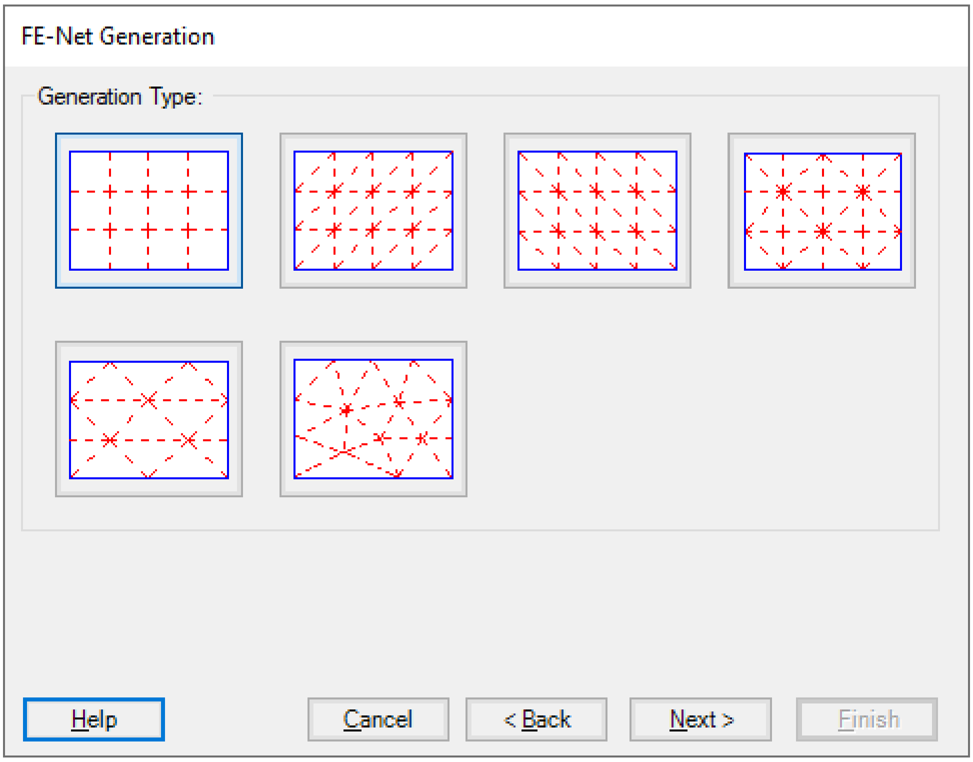


Figure 40 Generation type for square, rectangular and irregular slabs

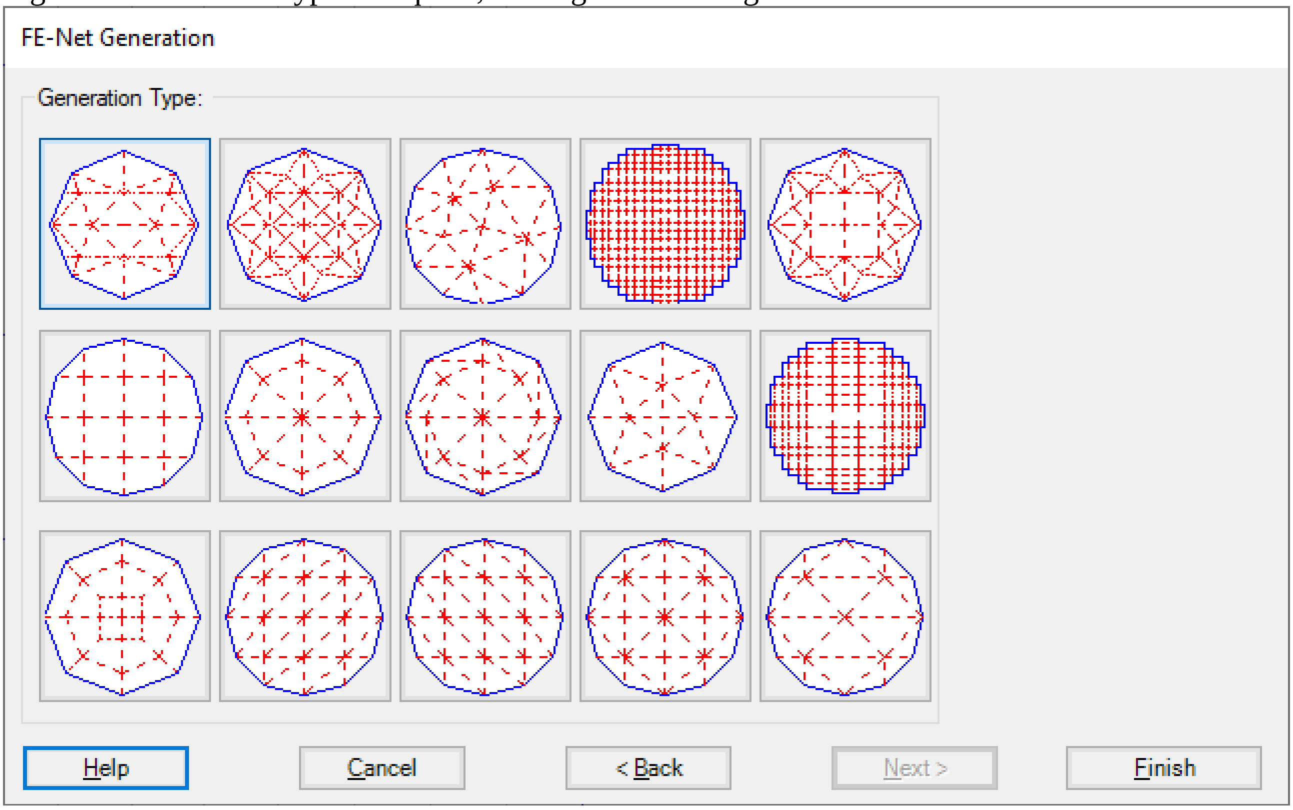


Figure 41 Generation type for circular and ring slabs

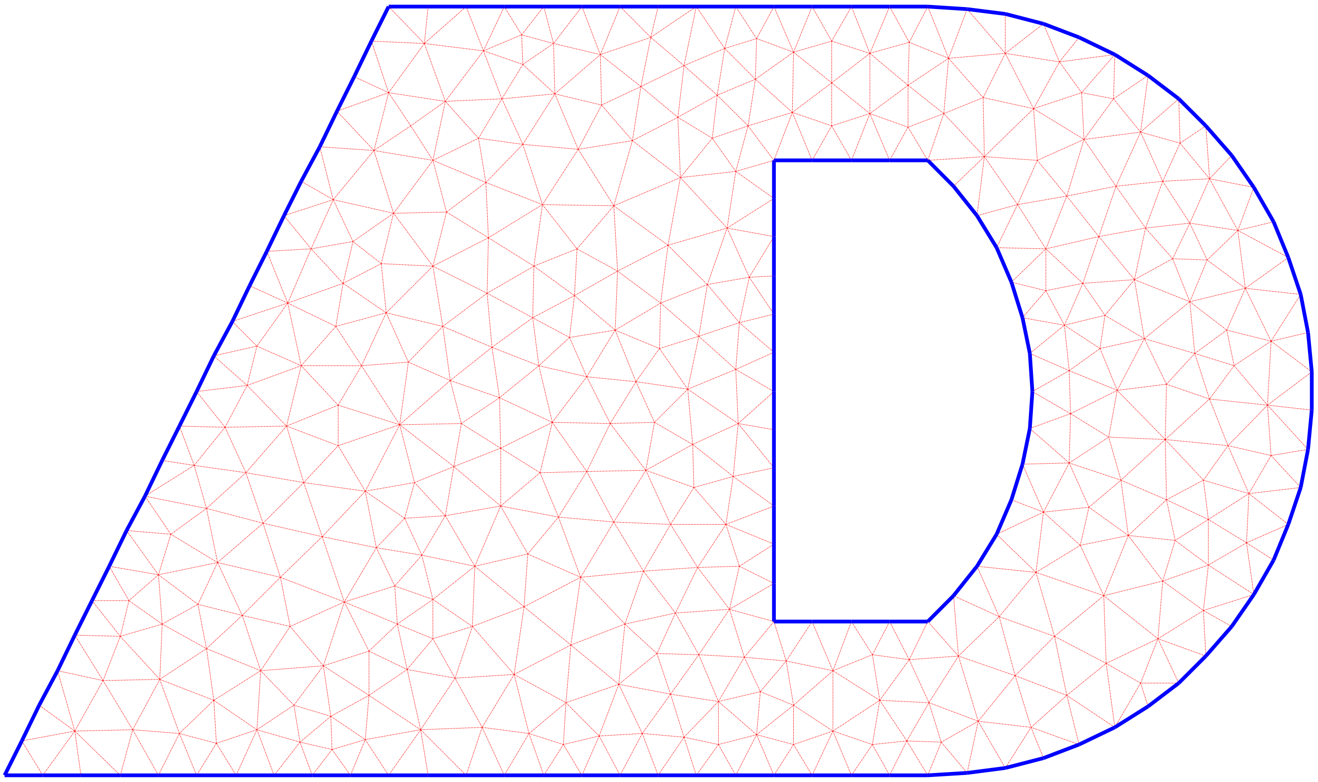


Figure 42 Irregular slab with triangular finite elements

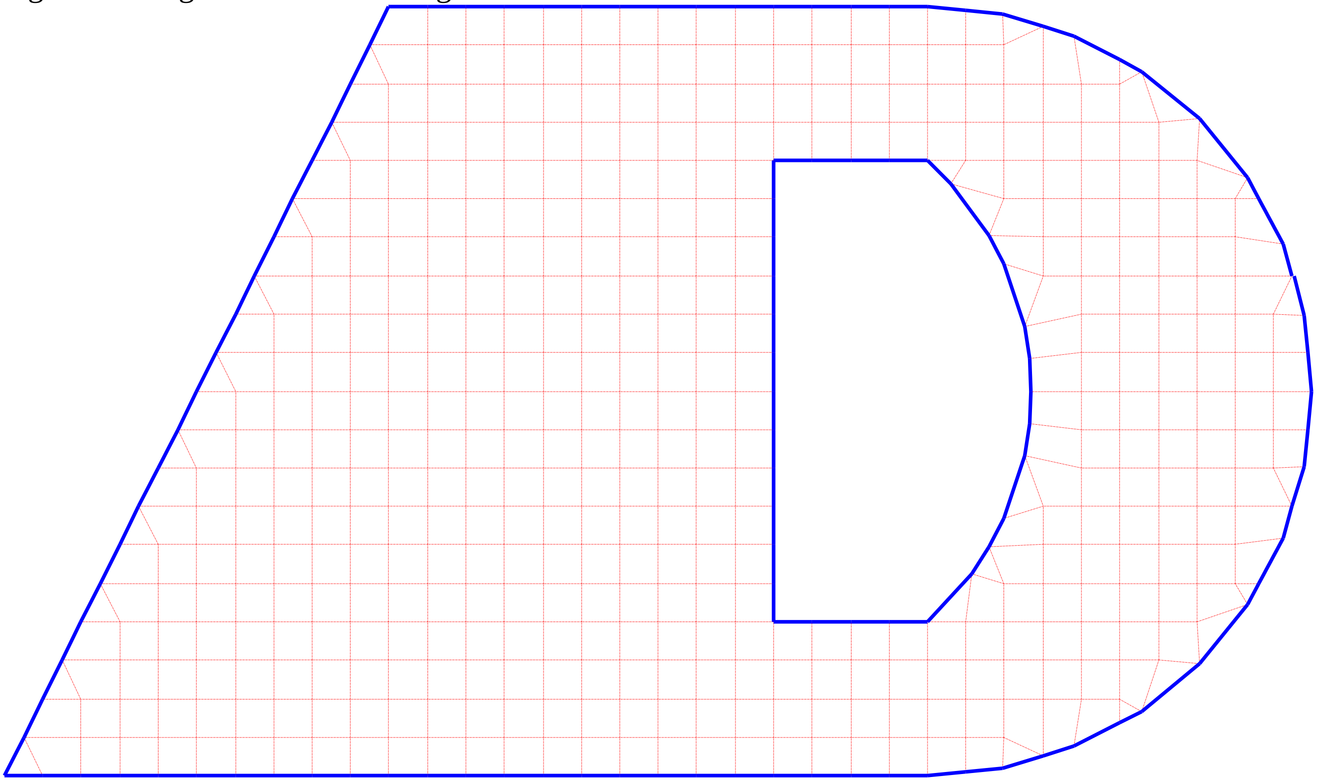


Figure 43 Irregular slab with combined rectangular, quadratic and triangular finite elements

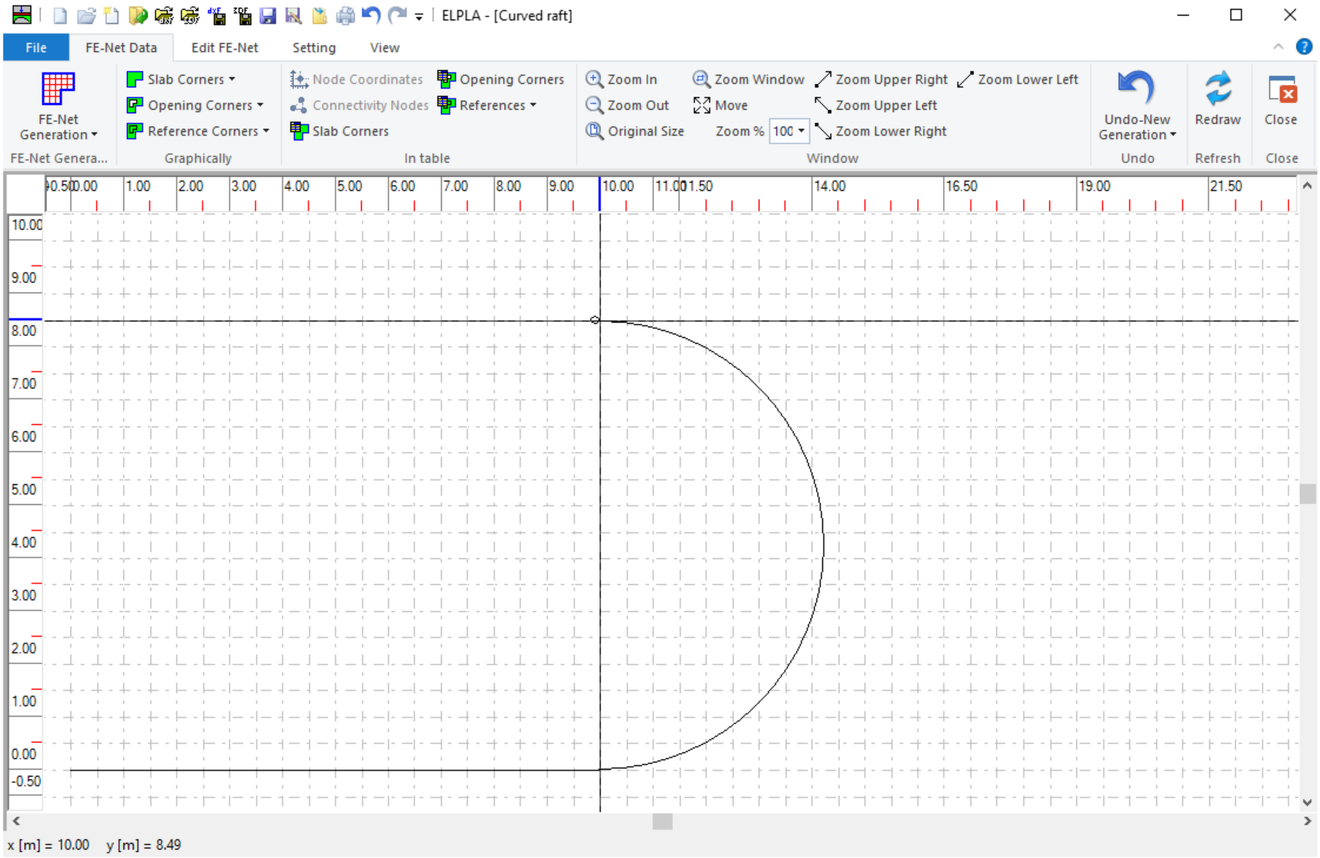


Figure 44 Drawing slab corners graphically by mouse

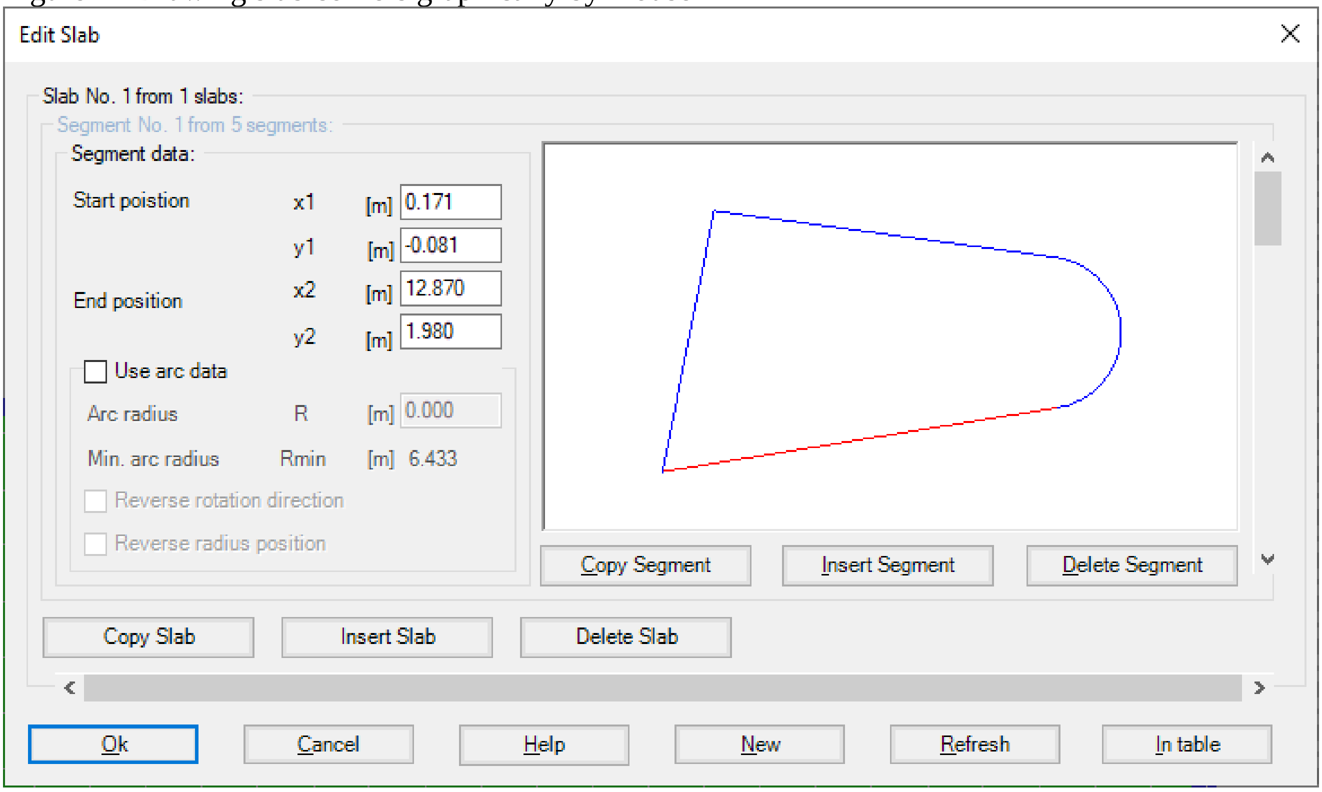


Figure 45 Defining slab corners in a table

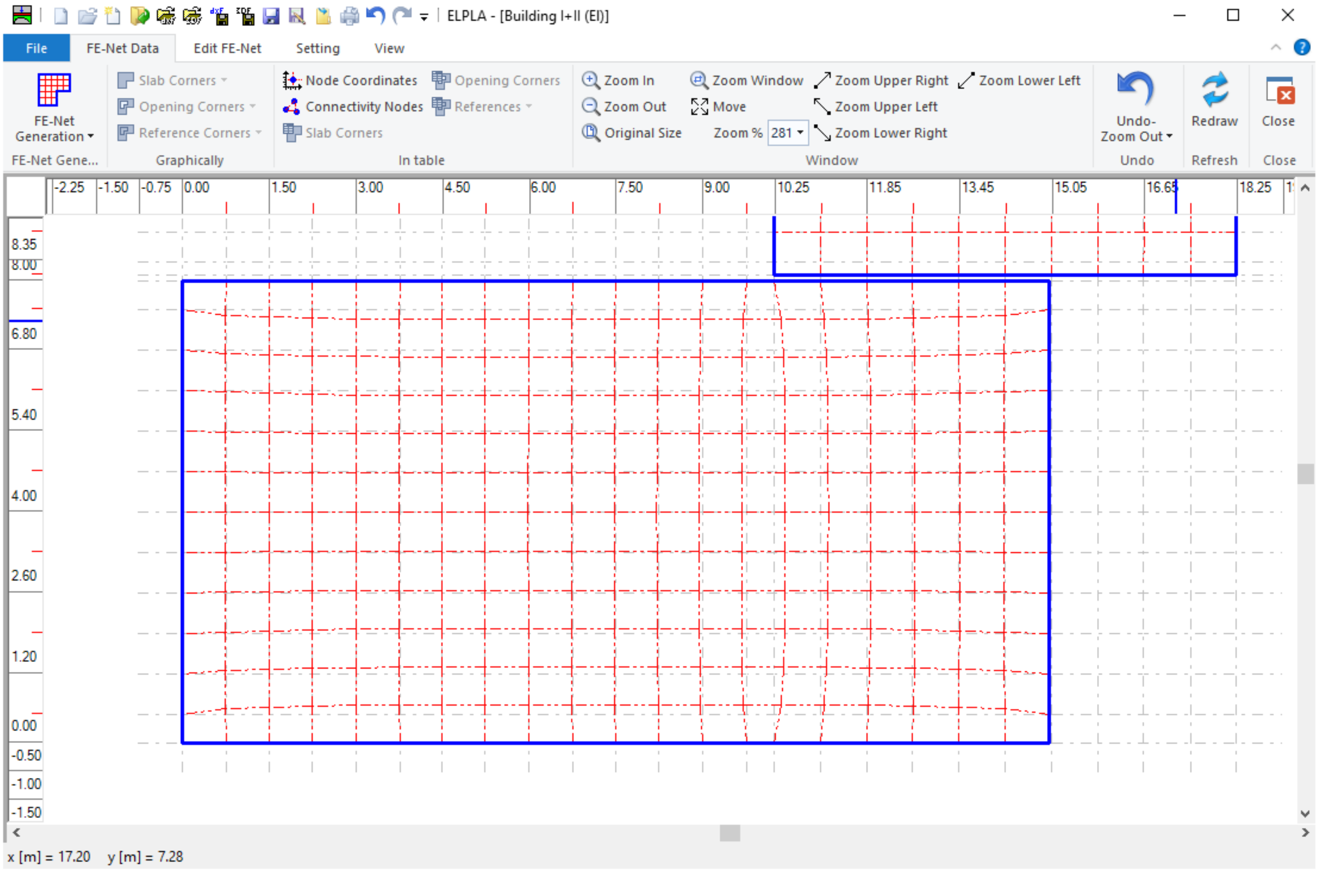


Figure 46 Smoothing the mesh

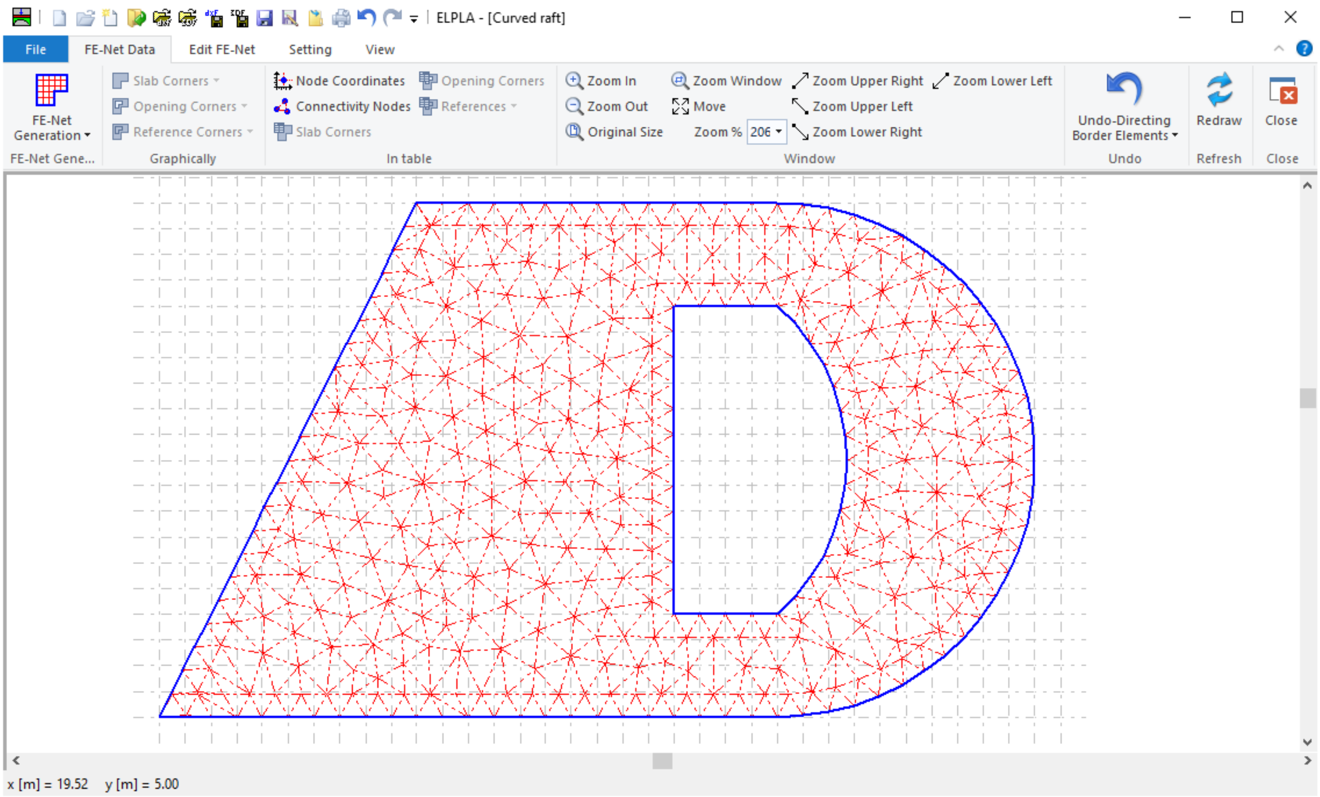


Figure 47 Directing border elements

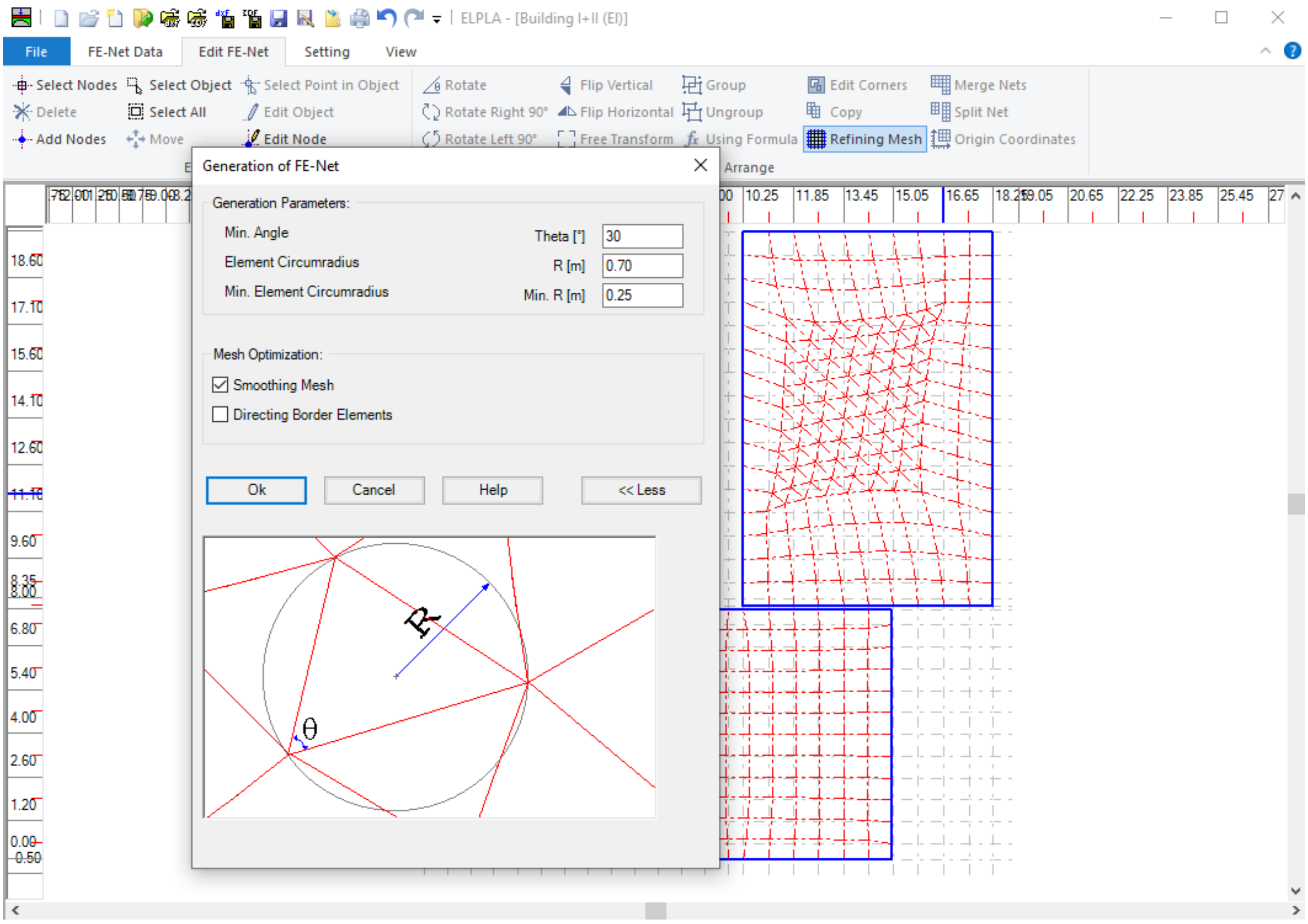


Figure 48 Refining the mesh around specified nodes

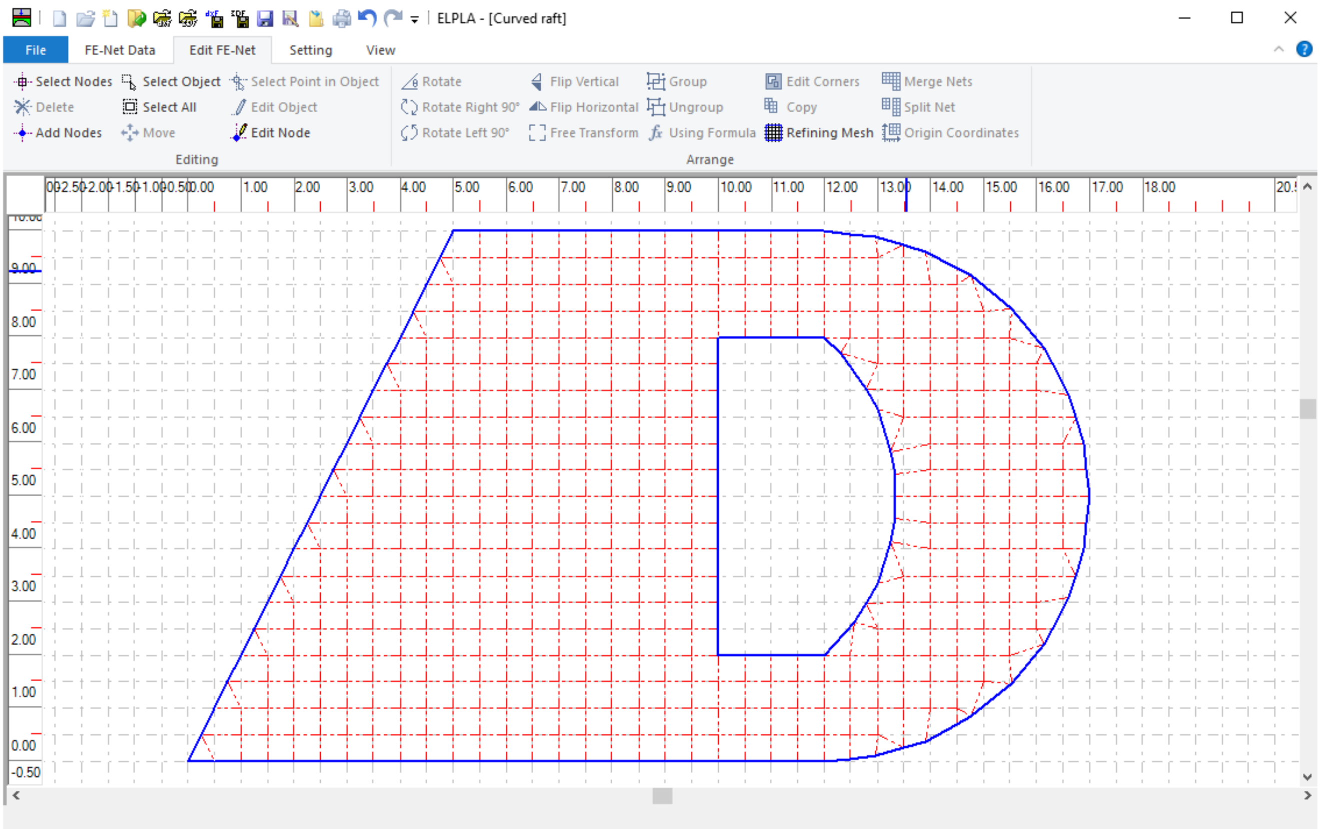


Figure 49 the merged mesh

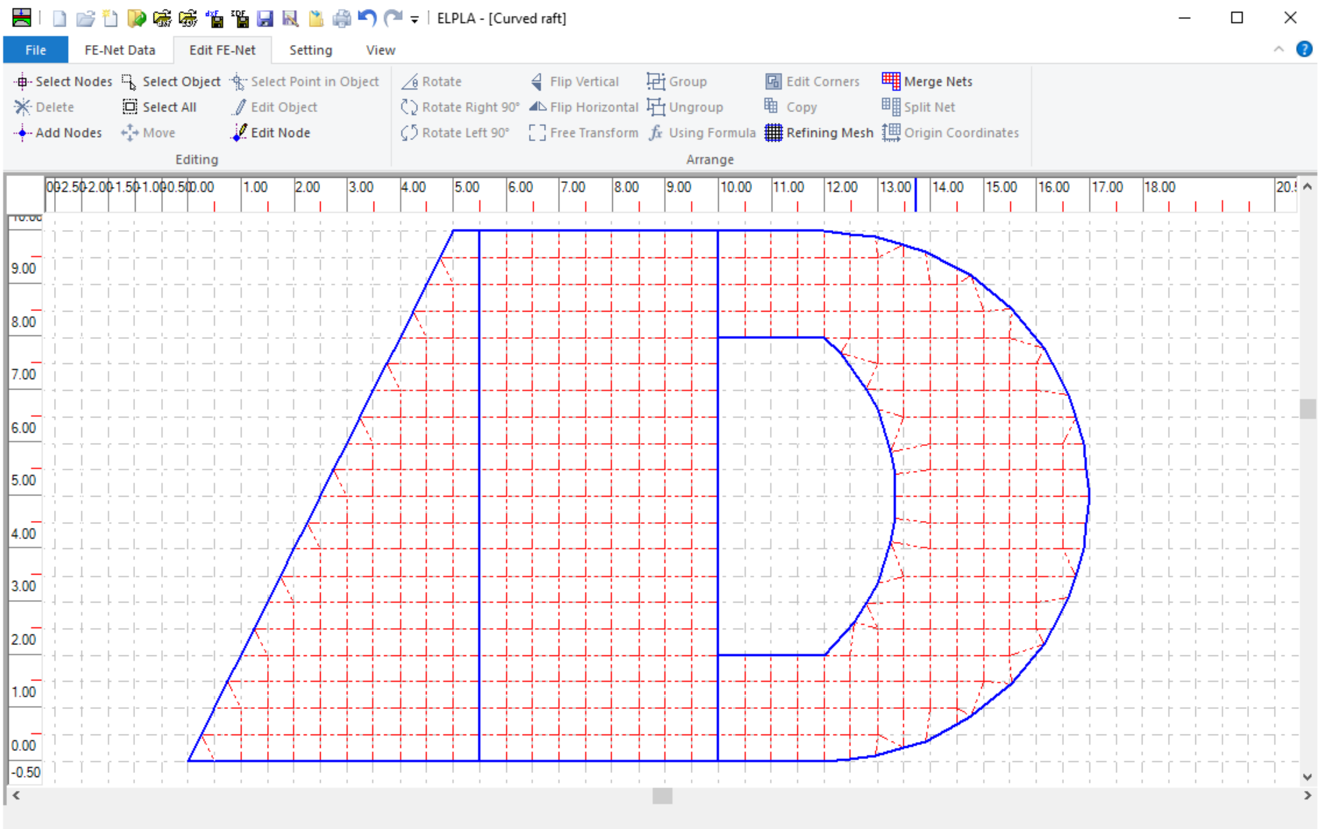


Figure 50 divided net

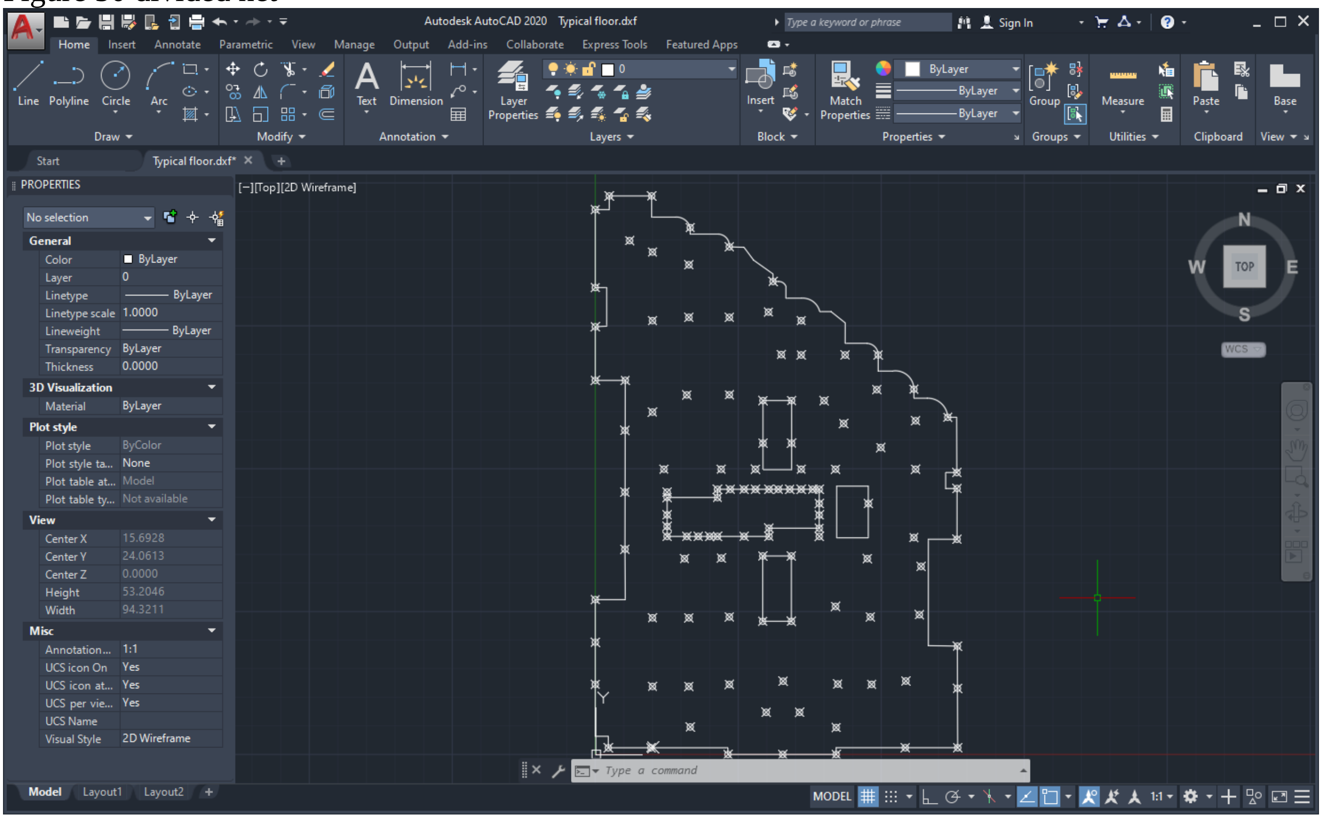


Figure 51 DXF references to be imported into ELPLA

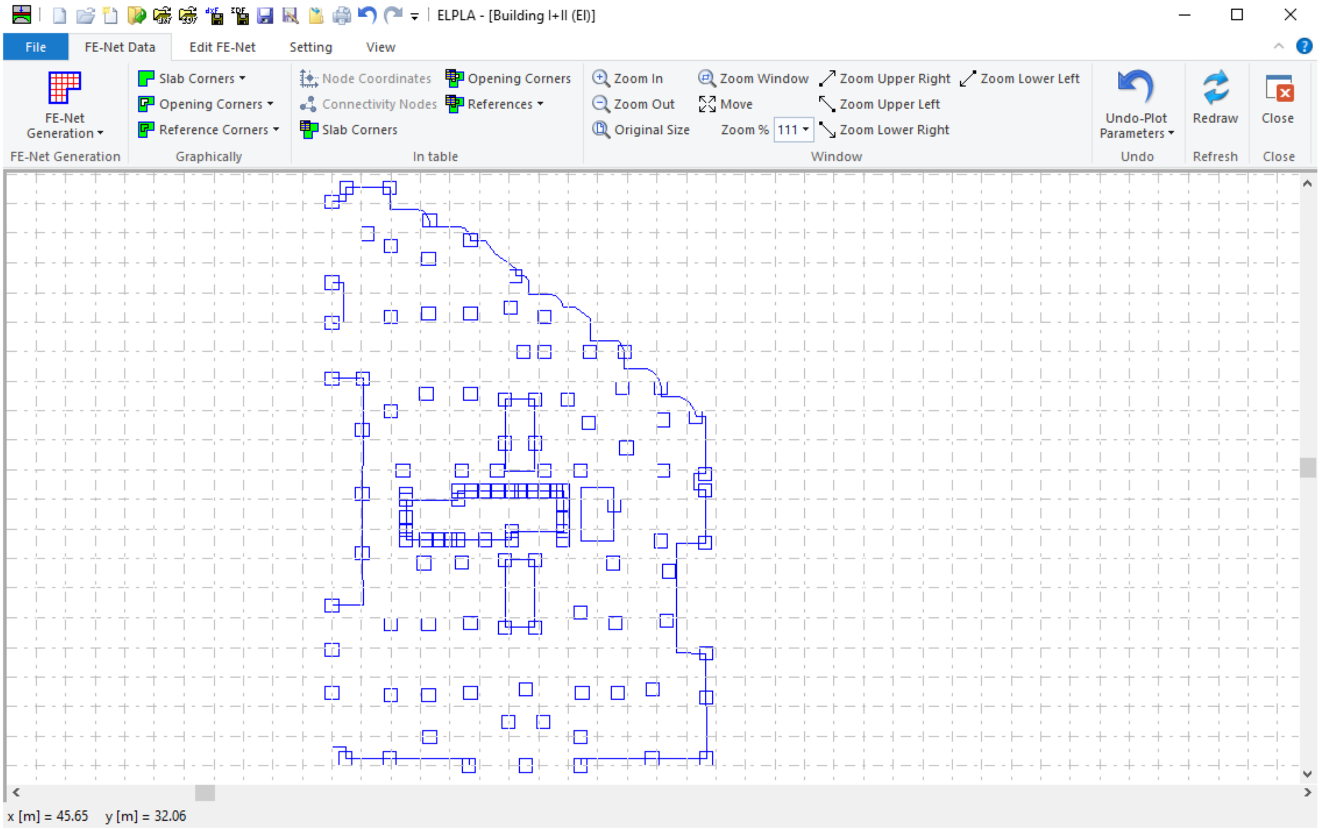


Figure 52 DXF references as ELPLA references

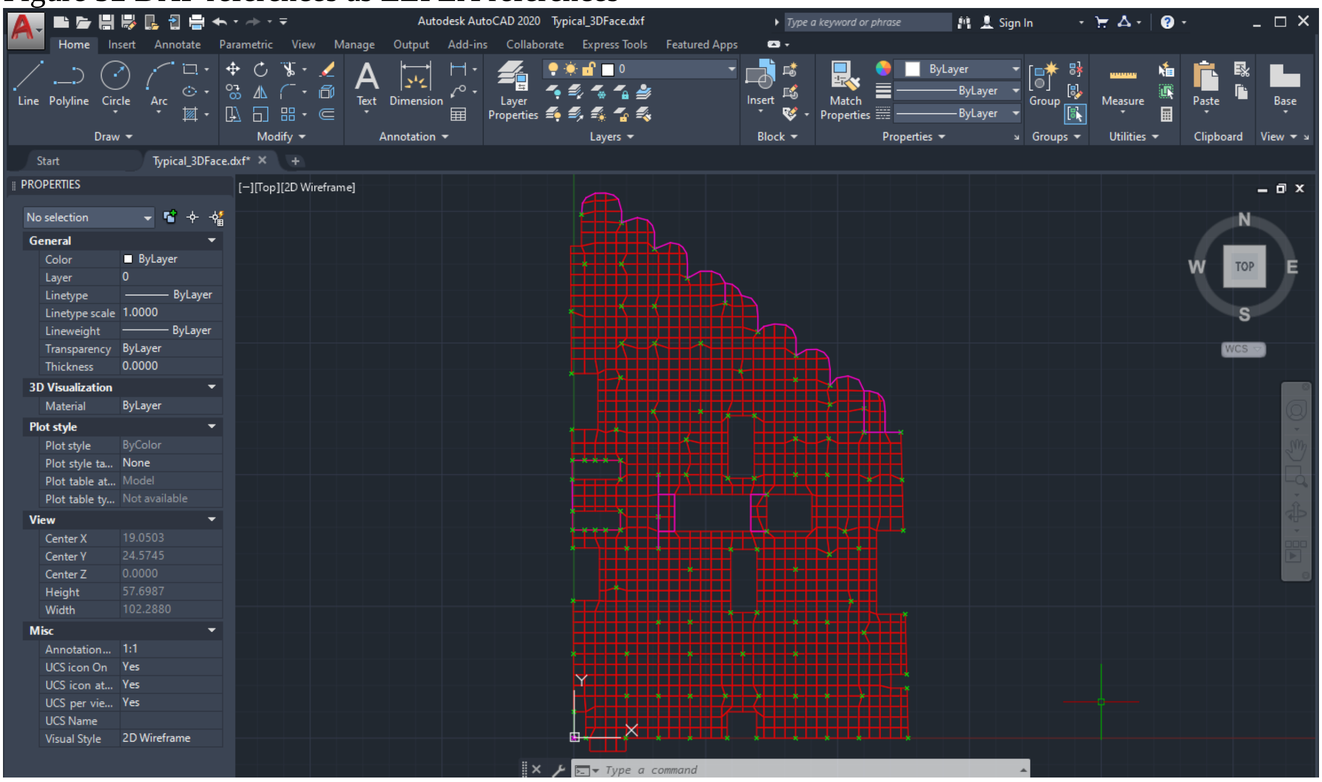


Figure 53 Finite element mesh created by command "3DFACE" in AutoCAD

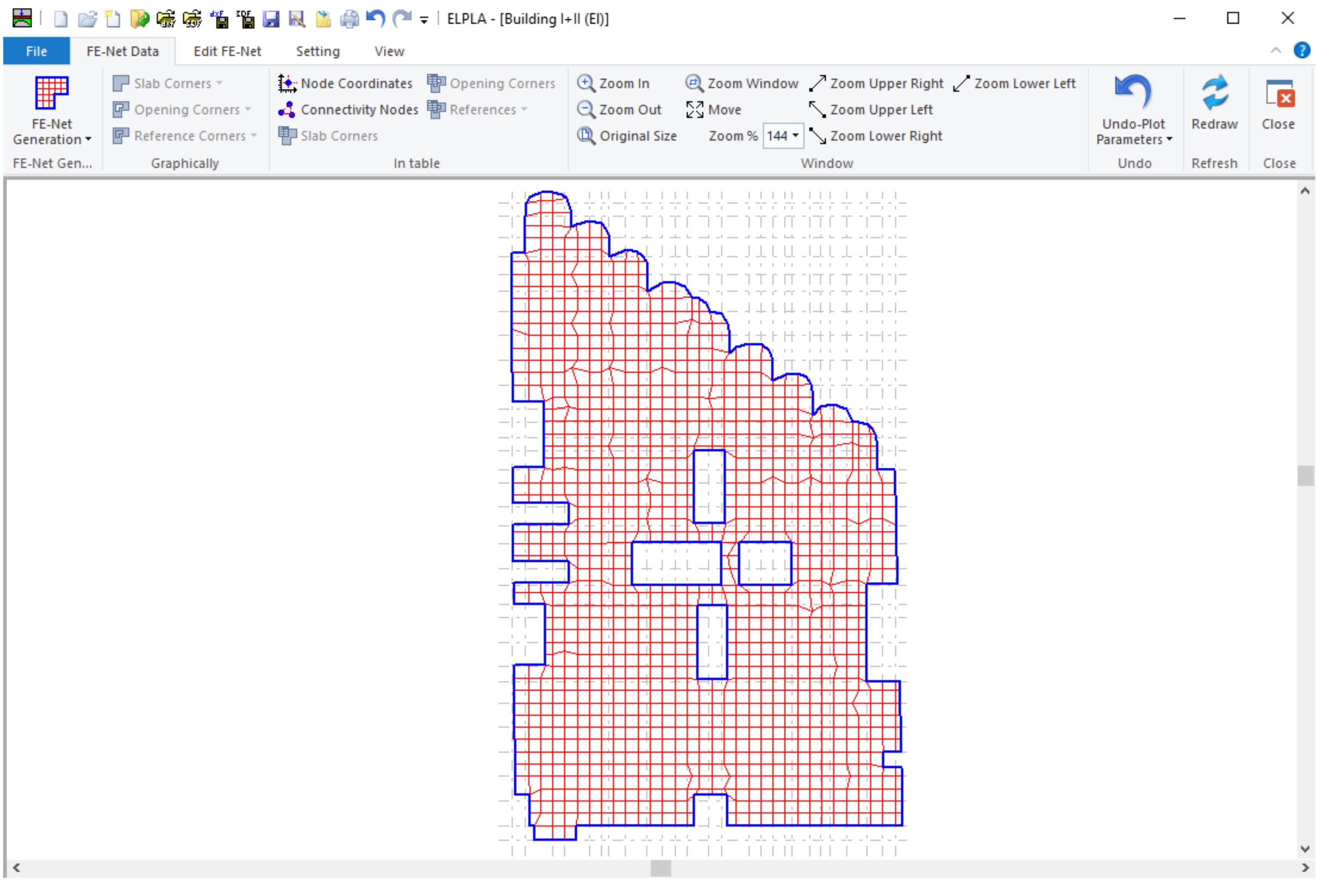


Figure 54 FE-Net as "3DFACE" AutoCAD type is imported into ELPLA from a DXF-file

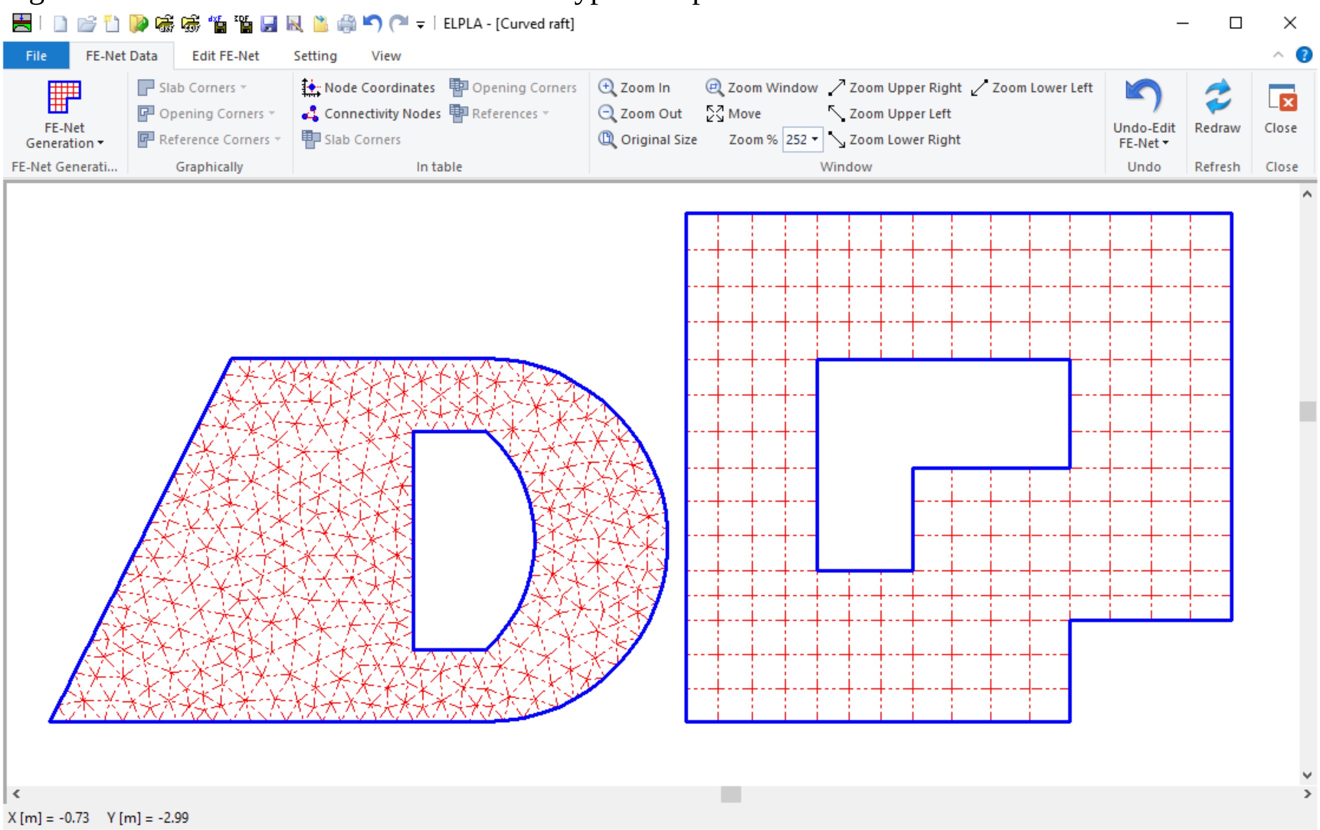


Figure 55 Two FE-Nets together

8 Boundary conditions

It is possible to define elastic or fixed rotations and displacements on the raft, Figure 56. Also, translational or rotational springs may be defined.

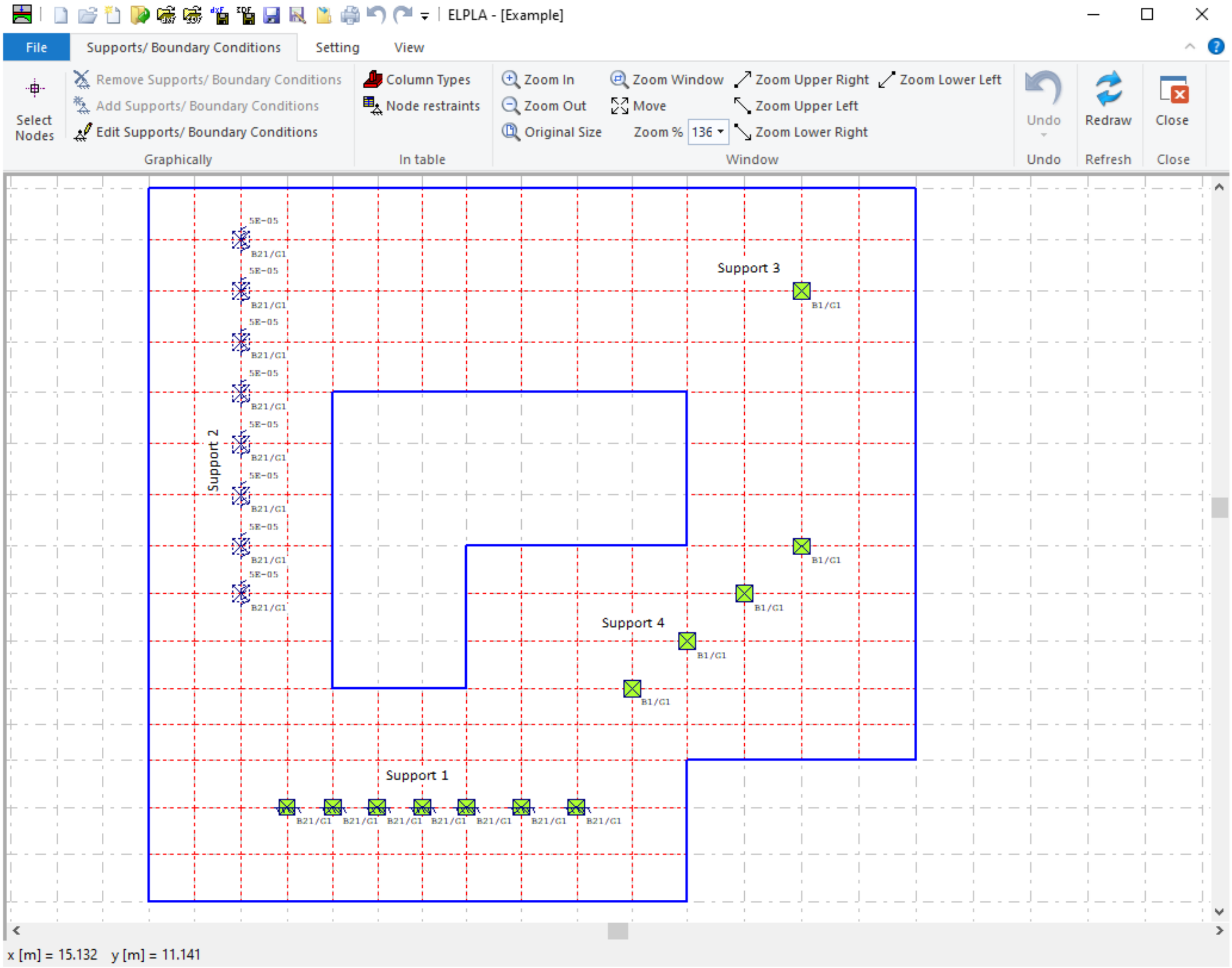


Figure 56 Elastic and fixed rotations and displacements

9 Soil

The soil is defined by a number of borings. Each one has multi-layers with different soil materials, Figure 57. Variable thickness and discontinuous soil strata can be considered, Figure 60. Loading and reloading of the soil modulus can be taken into account by the analysis, Figure 61. 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 60).

Three different methods are used to determine the flexibility coefficients or the modulus of subgrade reaction:

1. Interpolation method (Figure 57)
2. Subareas method (Figure 58)
3. Hand-Division of boring logs to nodes (Figure 59)

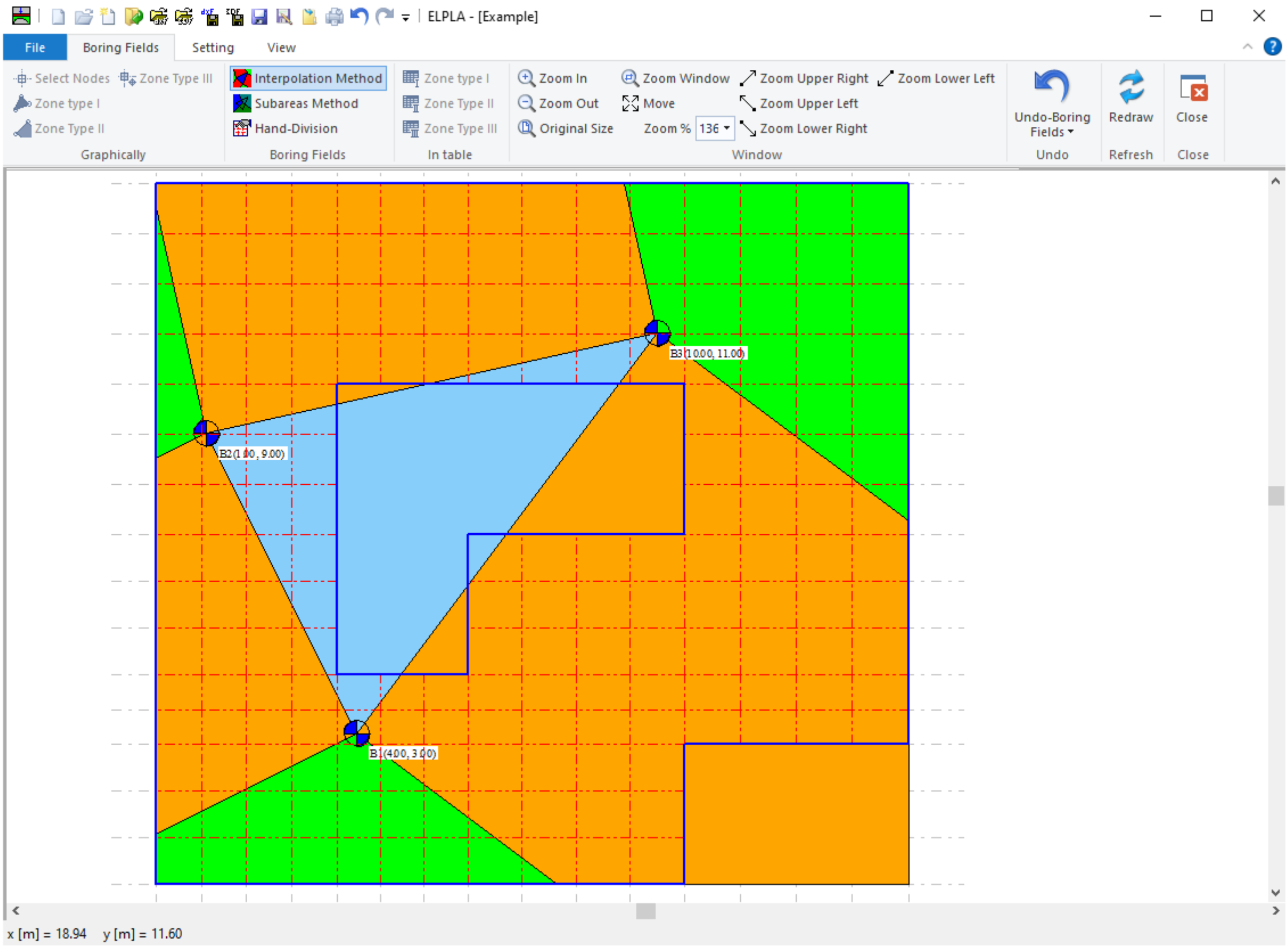


Figure 57 The soil is defined by a number of borings (Interpolation method)

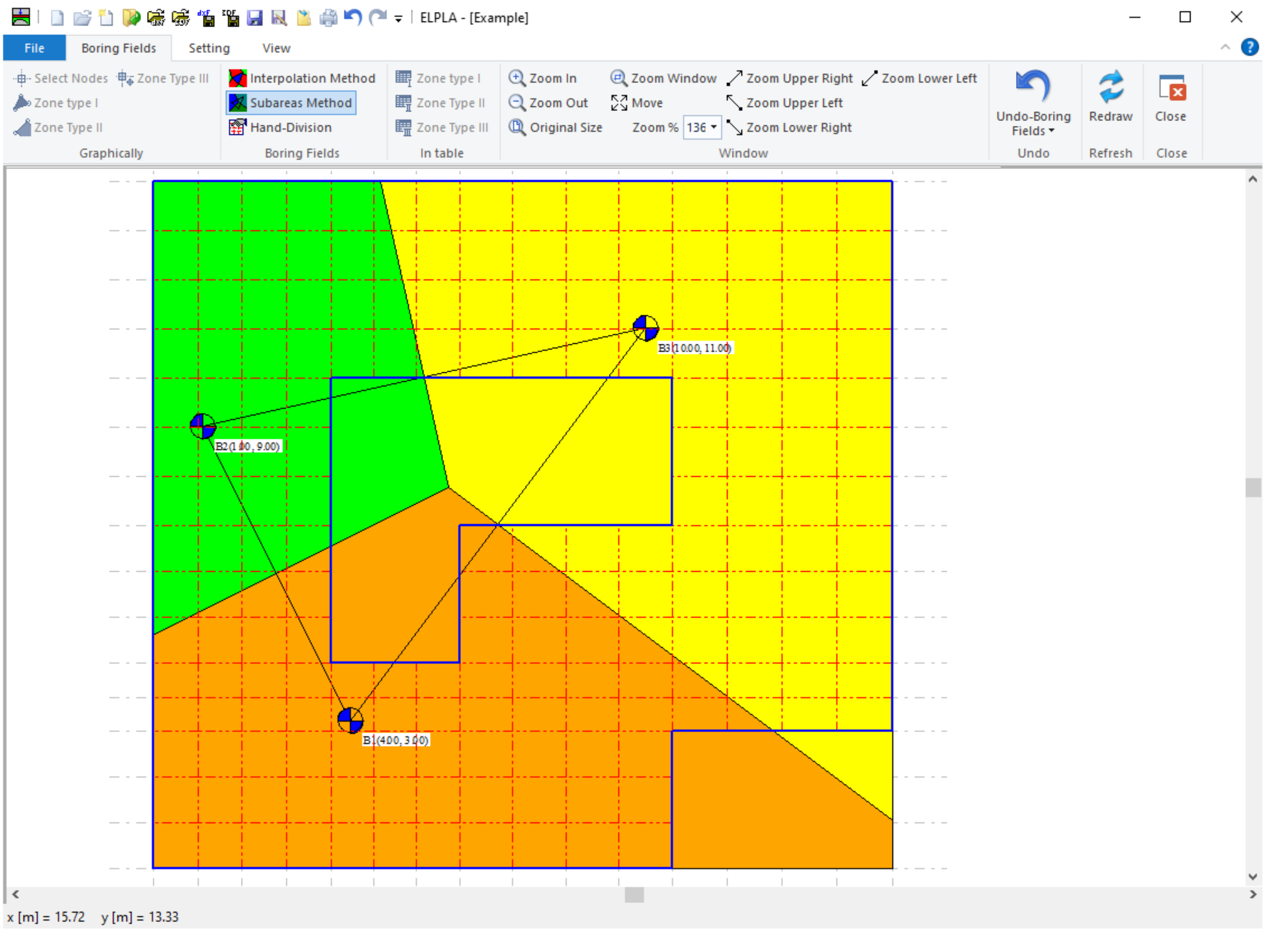


Figure 58 The soil is defined by a number of borings (Subareas method)

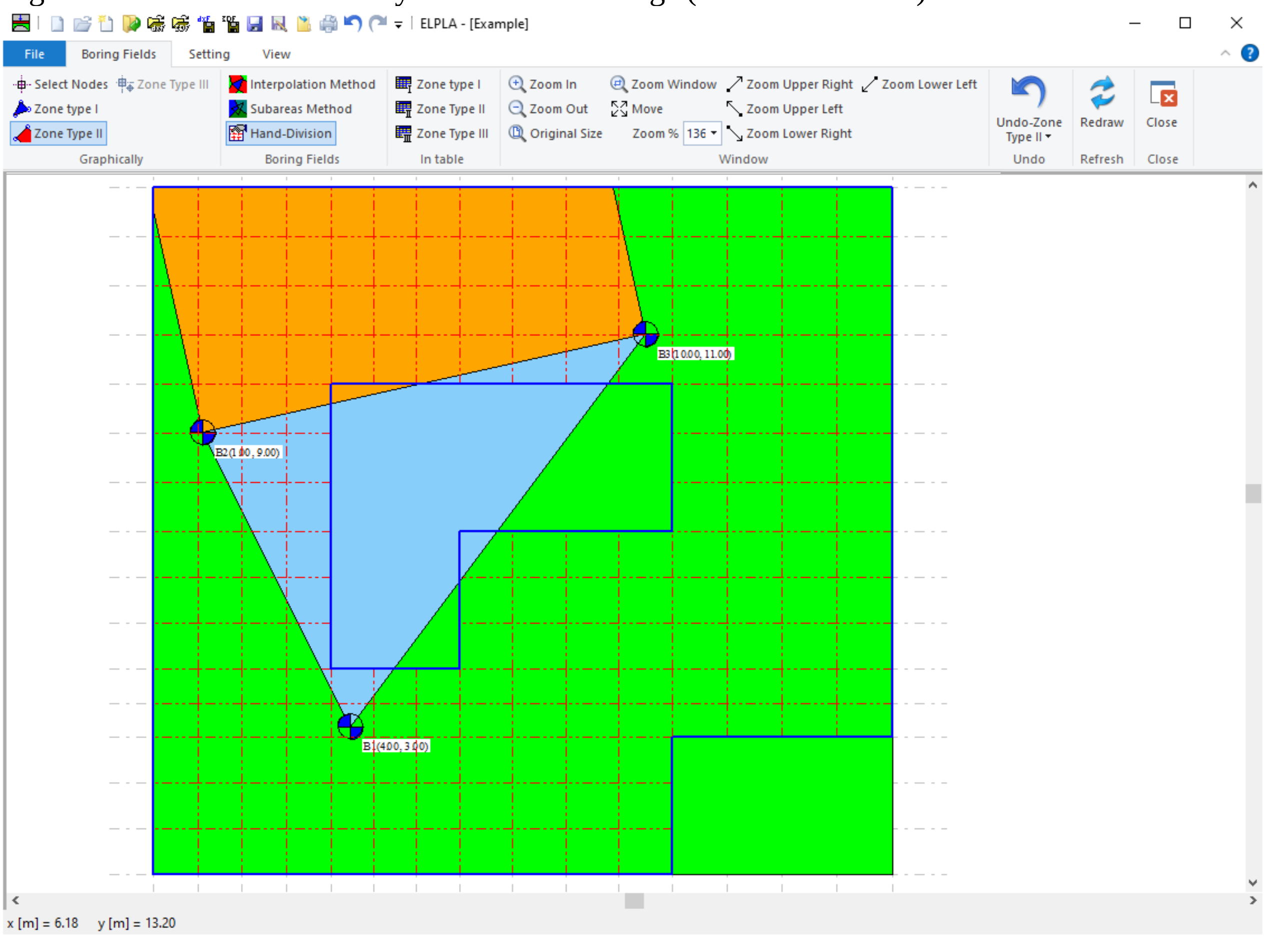


Figure 59 The soil is defined by a number of borings (Hand-Division of boring logs to nodes)

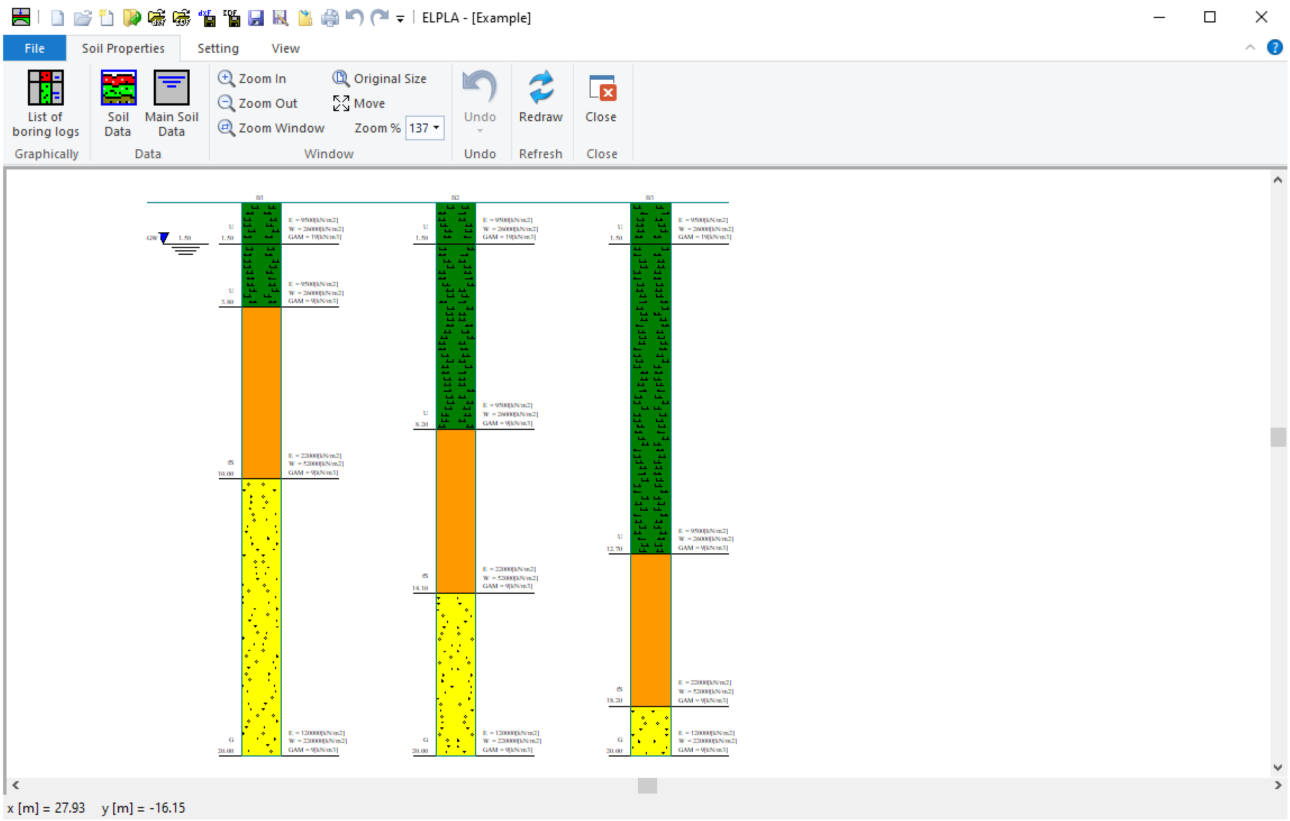


Figure 60 Variable thickness soil strata

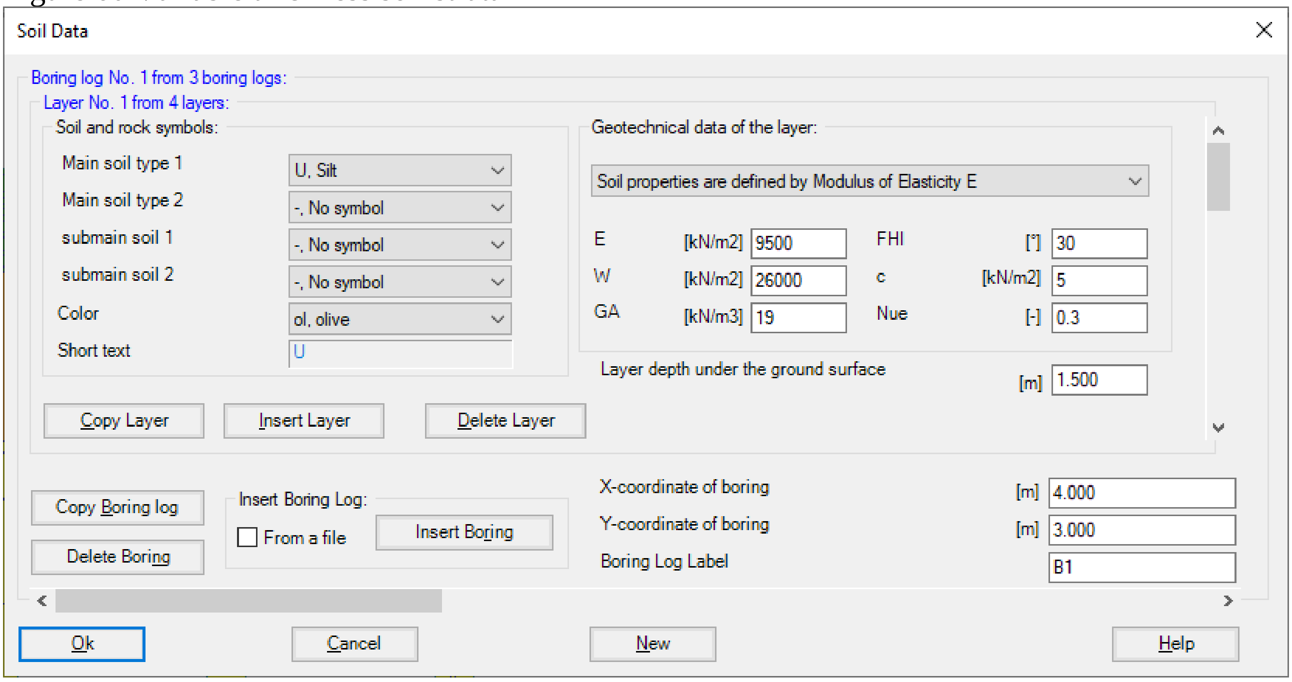


Figure 61 Loading and reloading soil modulus are considered

10 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

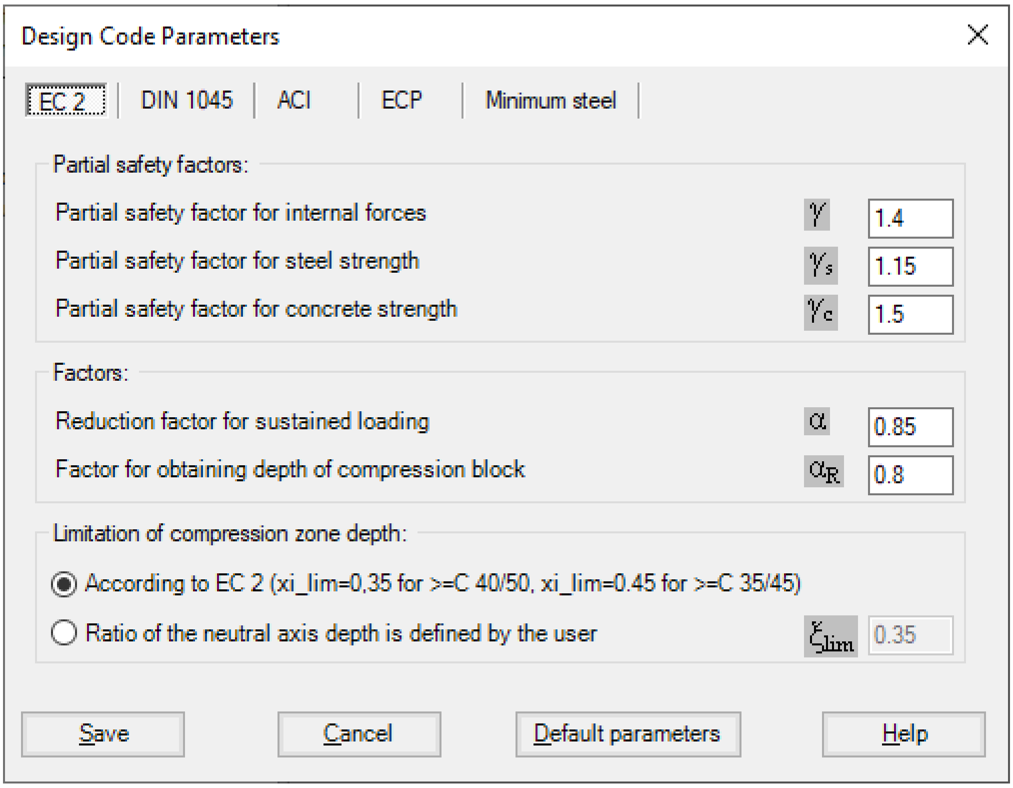


Figure 62 "Design code parameters" Dialog box

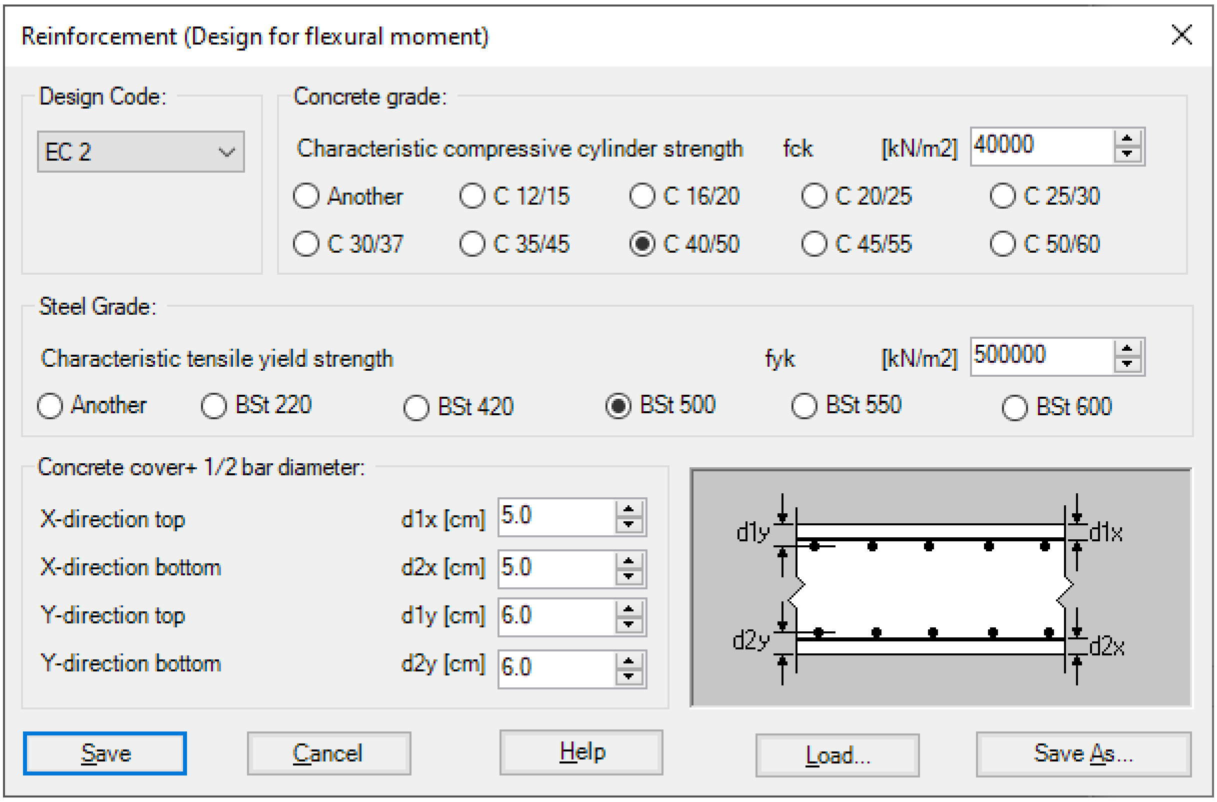


Figure 63 "Reinforcement" dialog box

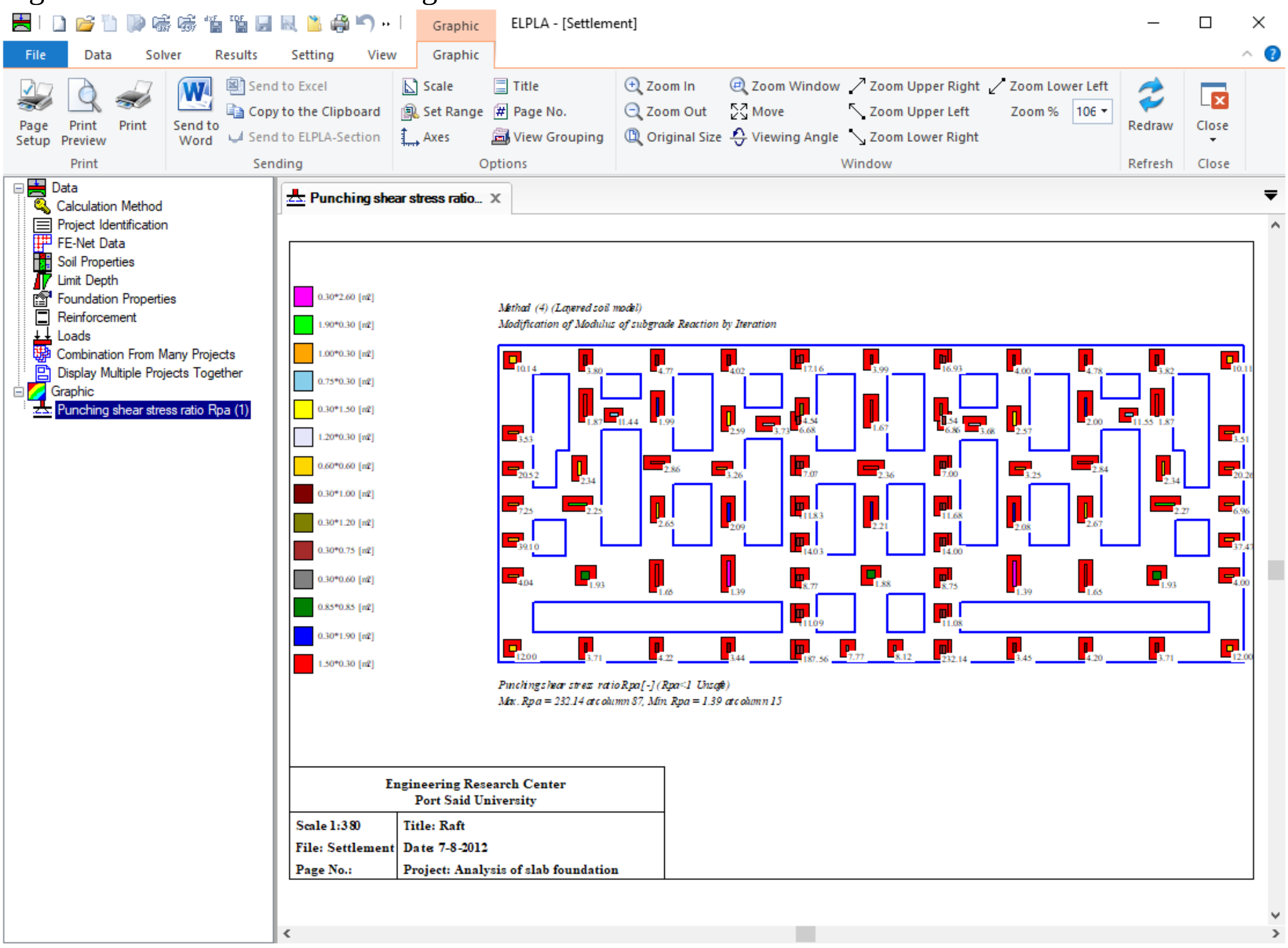


Figure 64 Area of a critical section of punching shear

11 Graphical drawing of data and results

You can display, plot, and print data and results graphically using the "Graphic" tab. 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 65 to Figure 76).

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.

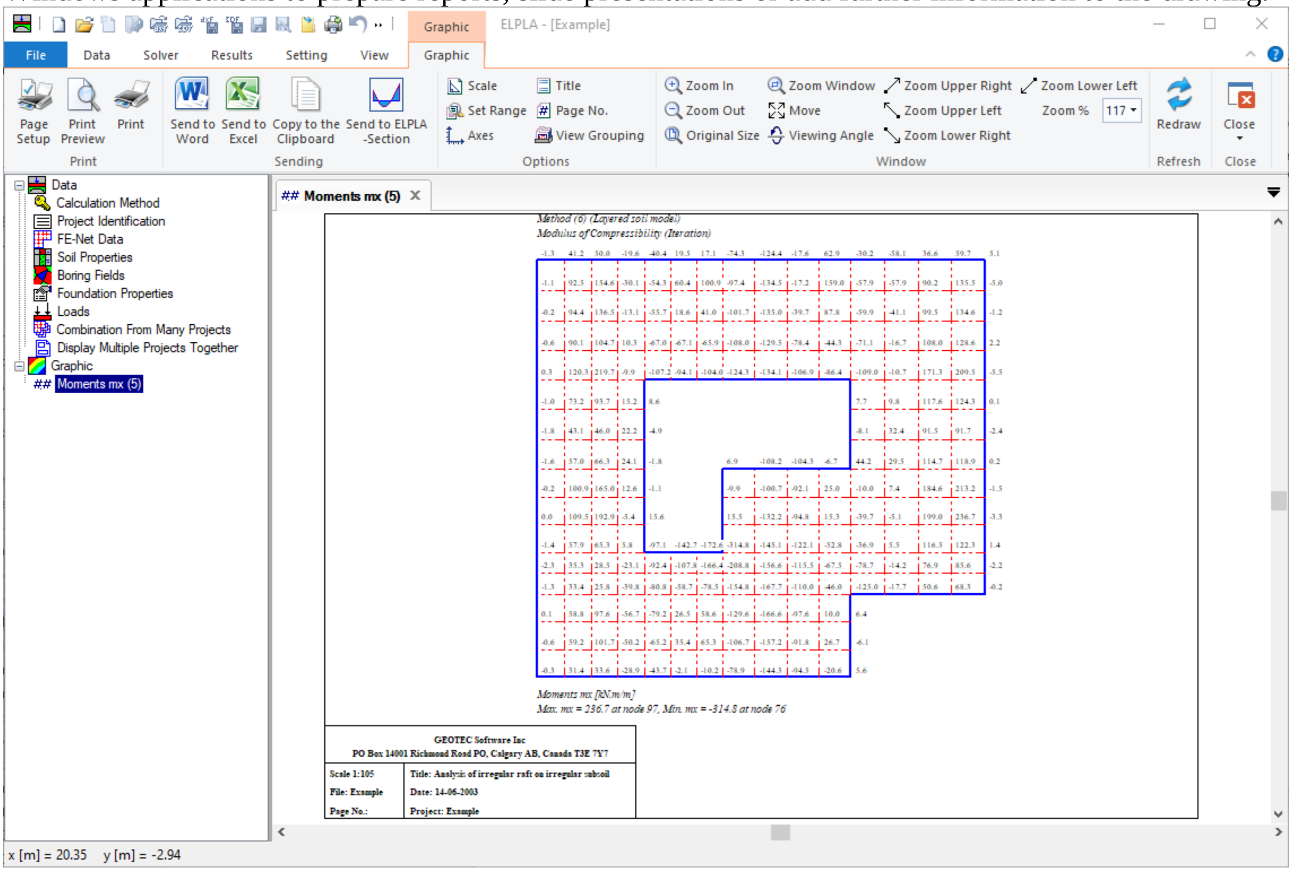


Figure 65 Results can be tabulated on the mesh

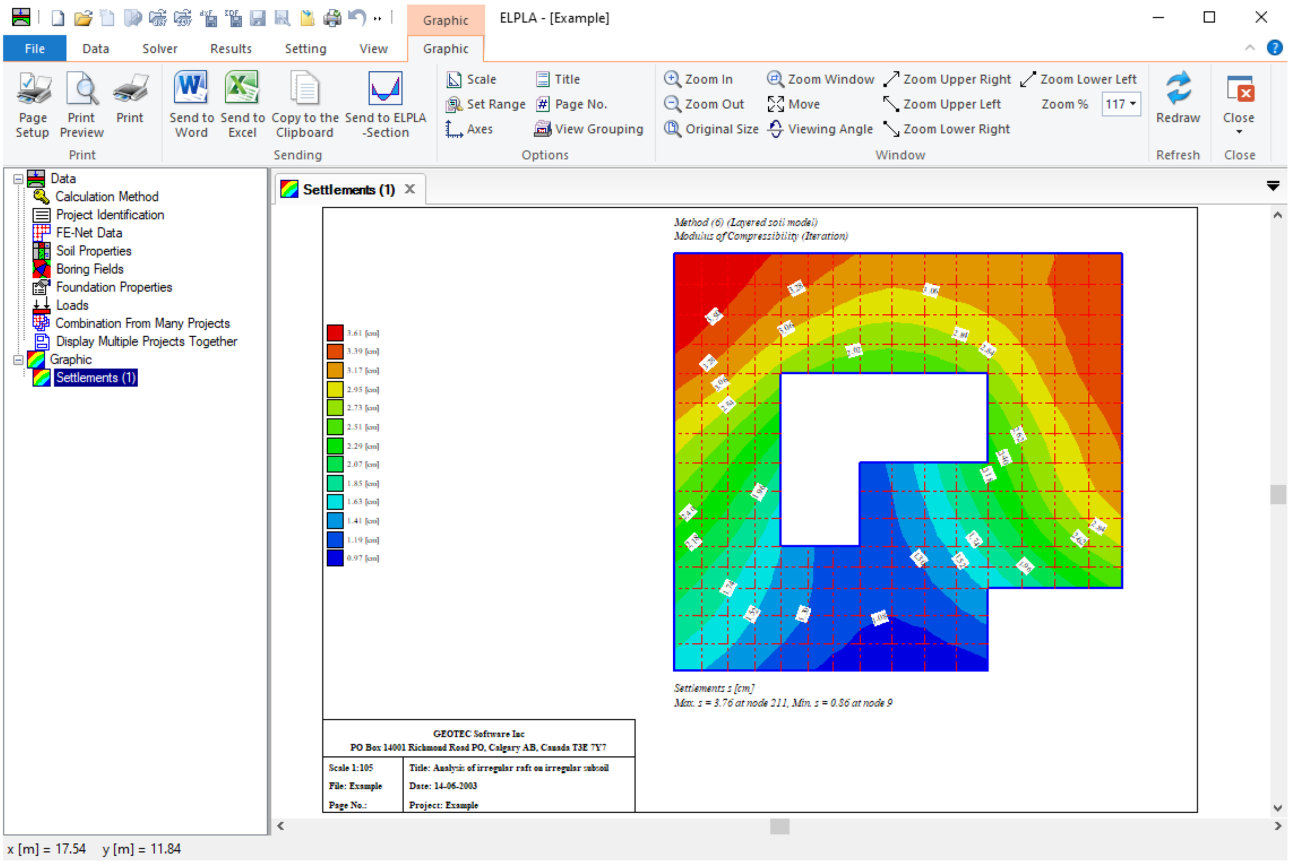


Figure 66 Results can be contoured

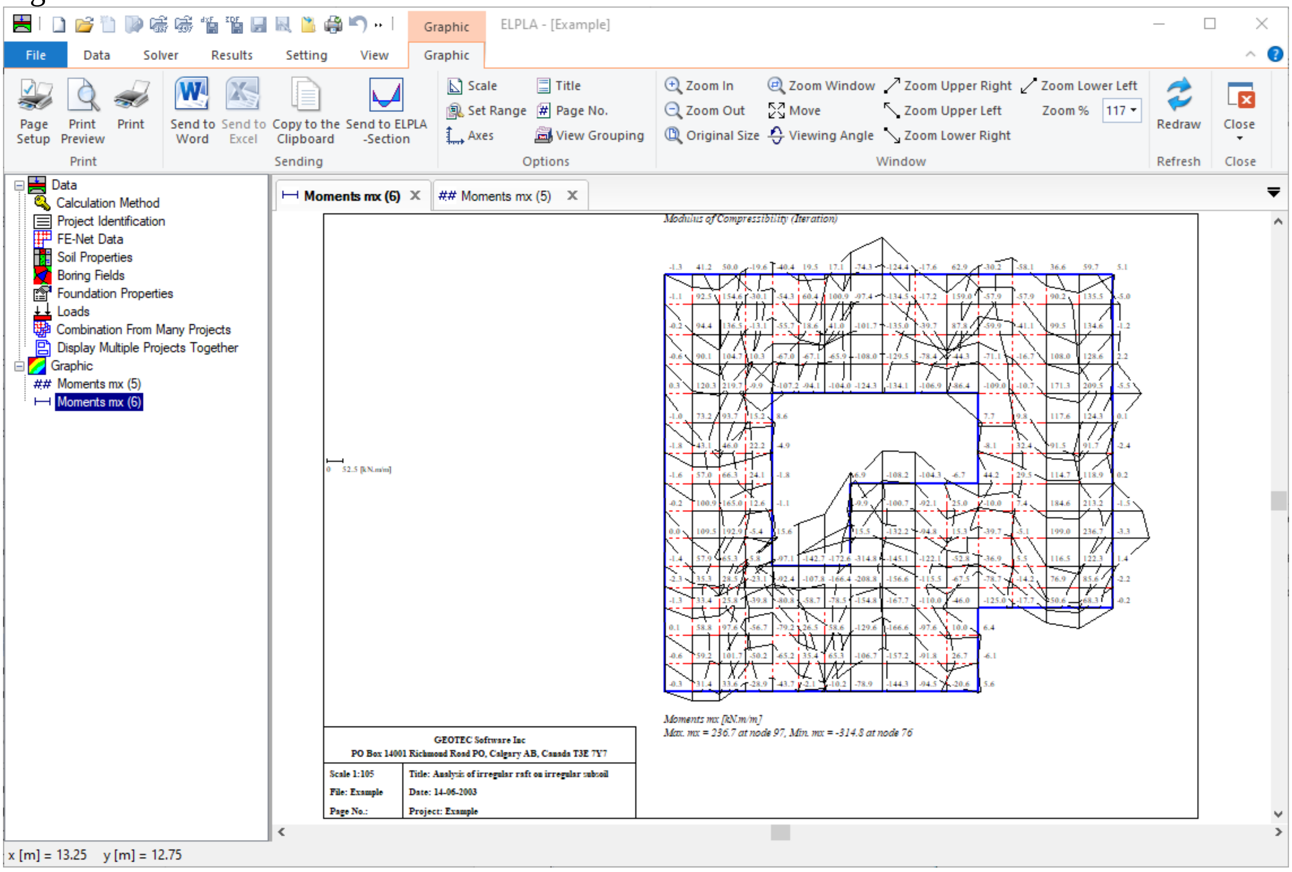


Figure 67 Moment distribution on the raft can be plotted

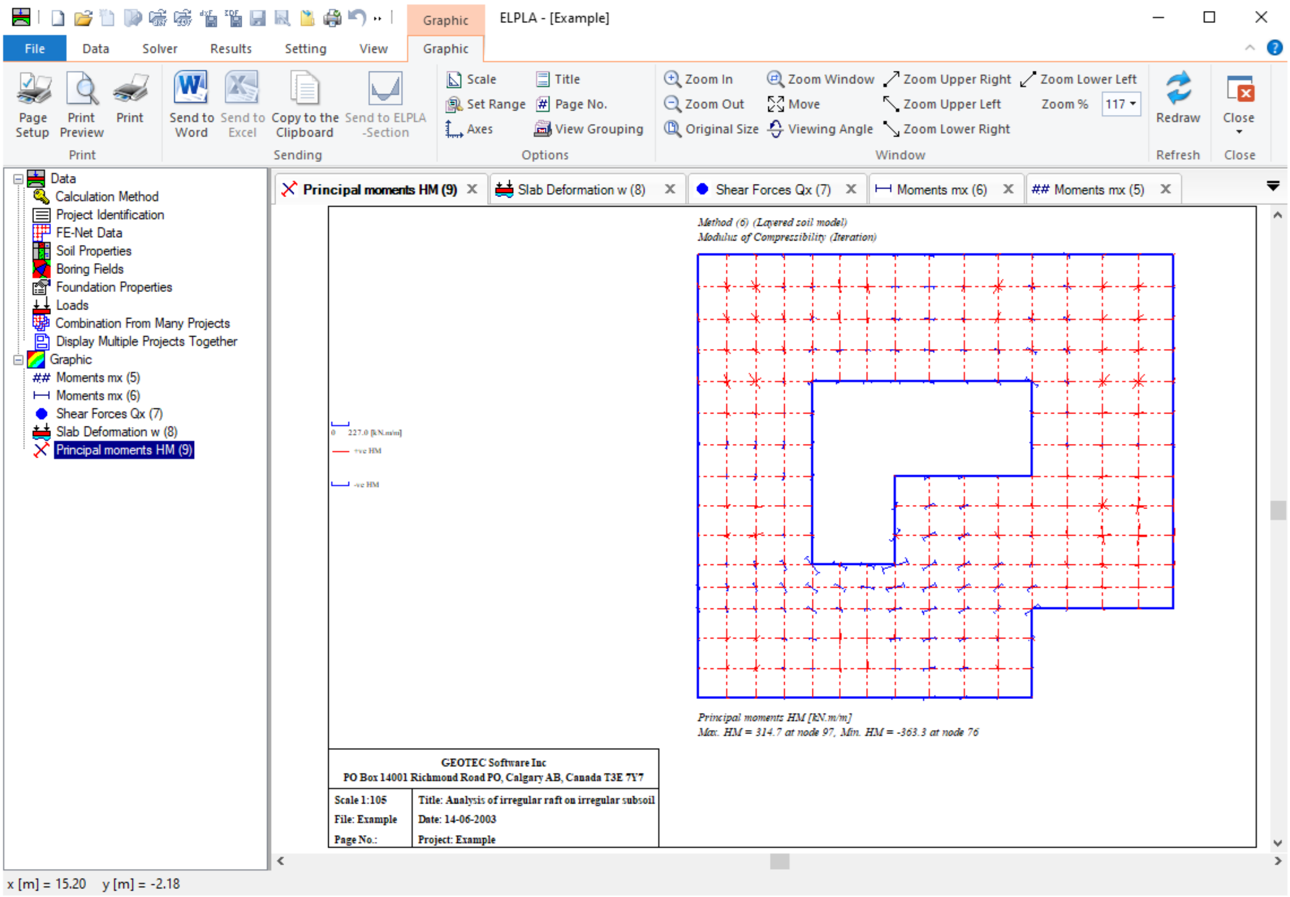


Figure 68 Principal moments as streaks

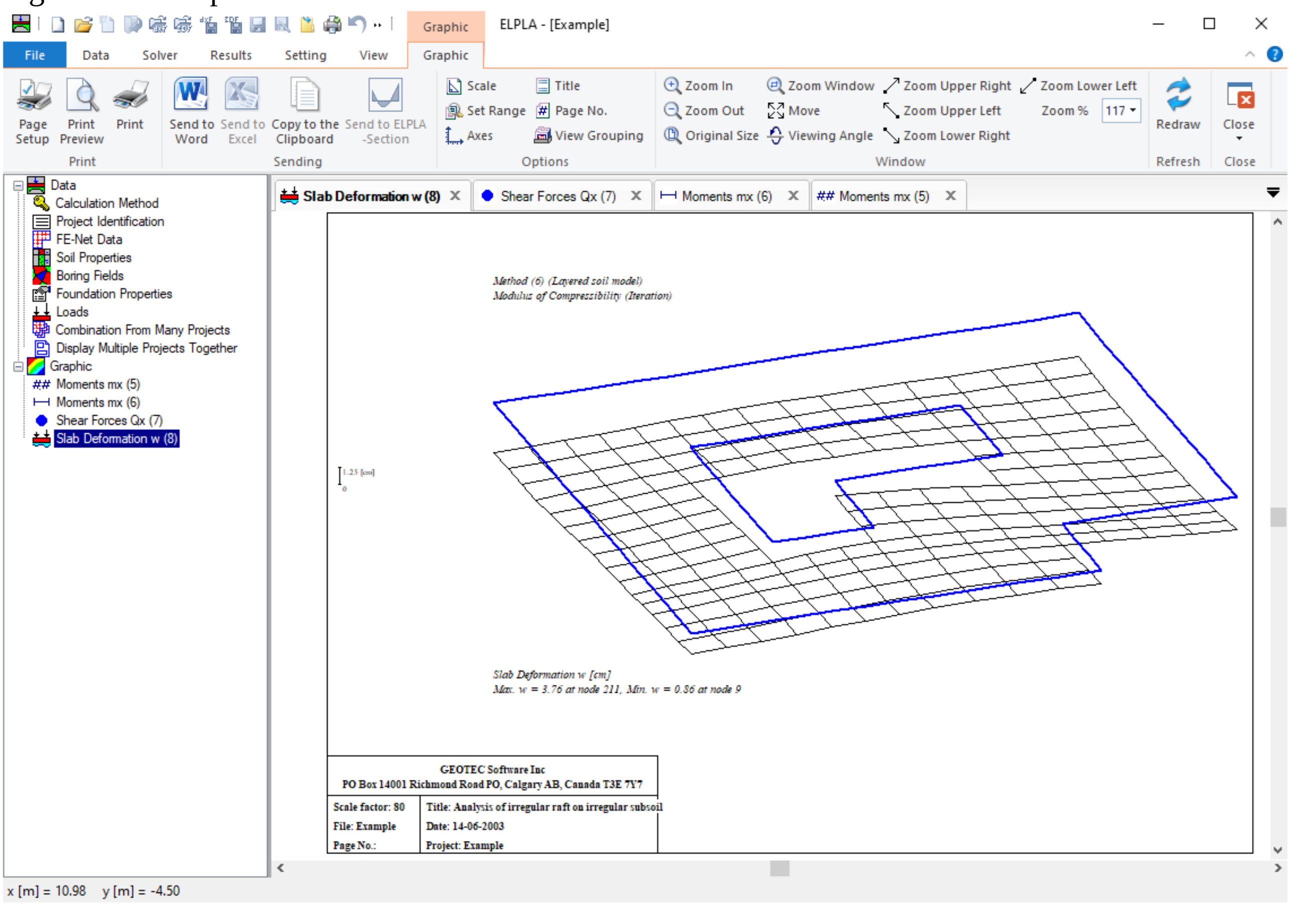


Figure 69 Raft deformation can be plotted as a deformed mesh

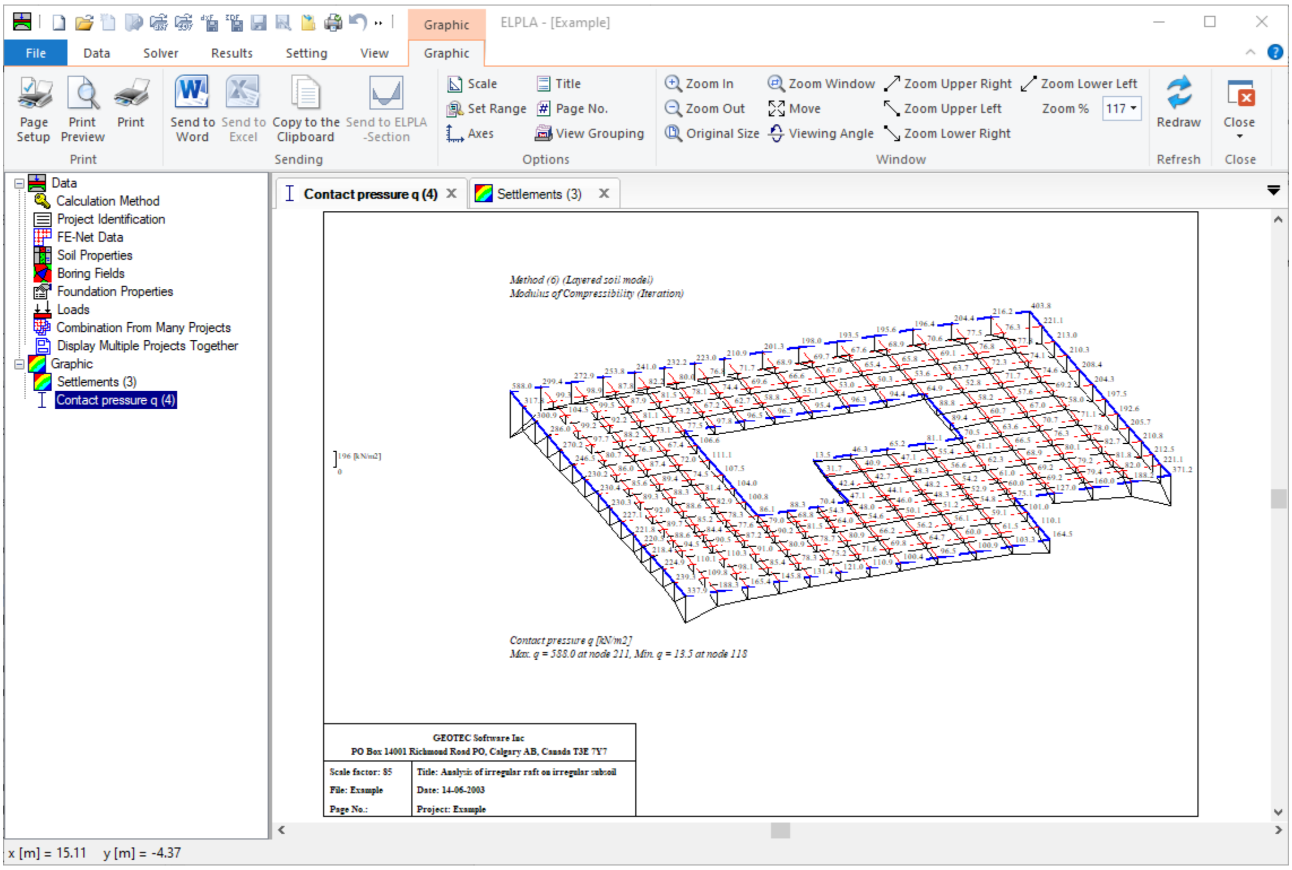


Figure 70 Results can be plotted in isometric shape

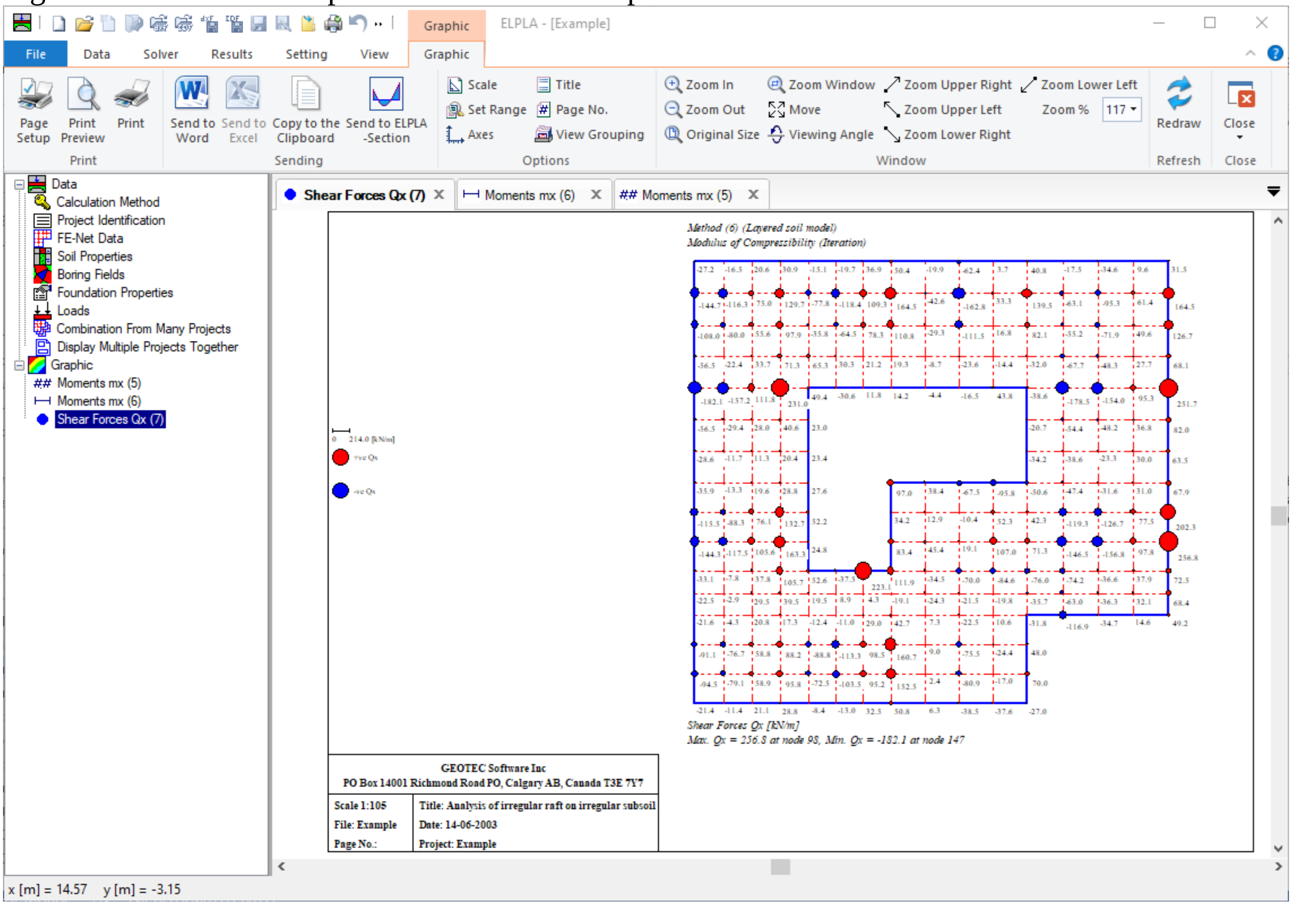


Figure 71 Results can be plotted as a circular diagram

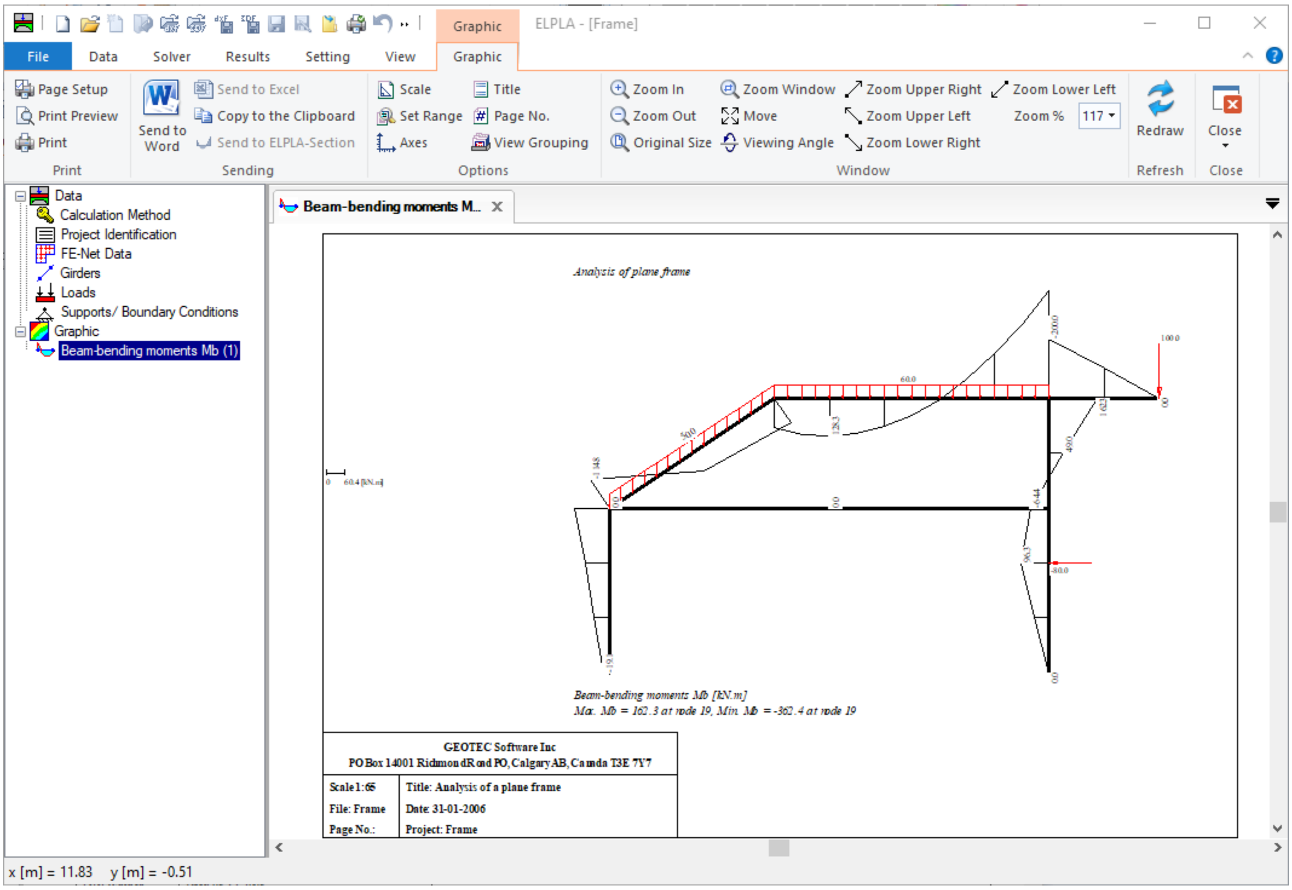


Figure 72 Distribution of bending moments

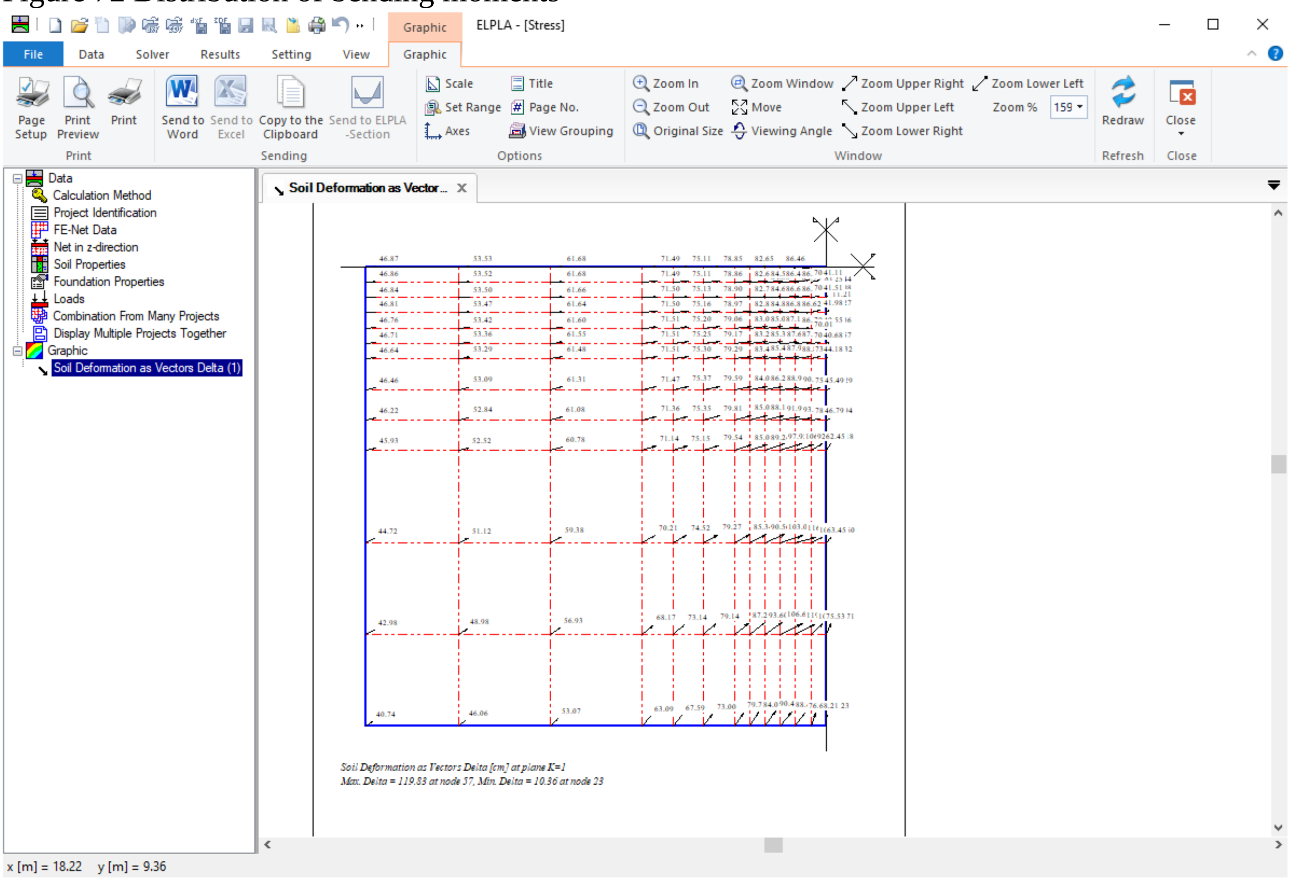


Figure 73 Soil deformation as vectors

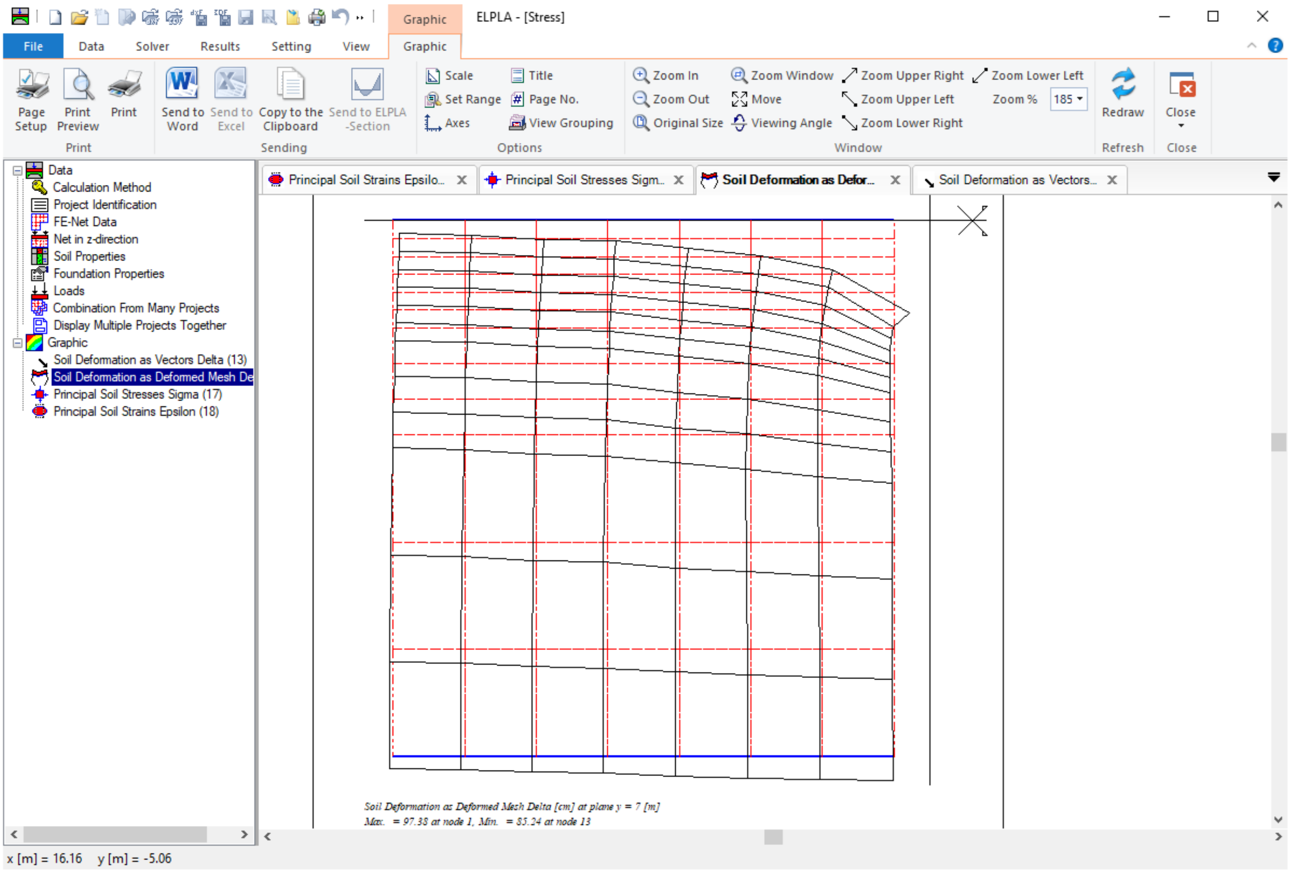


Figure 74 Soil deformation as deformed mesh

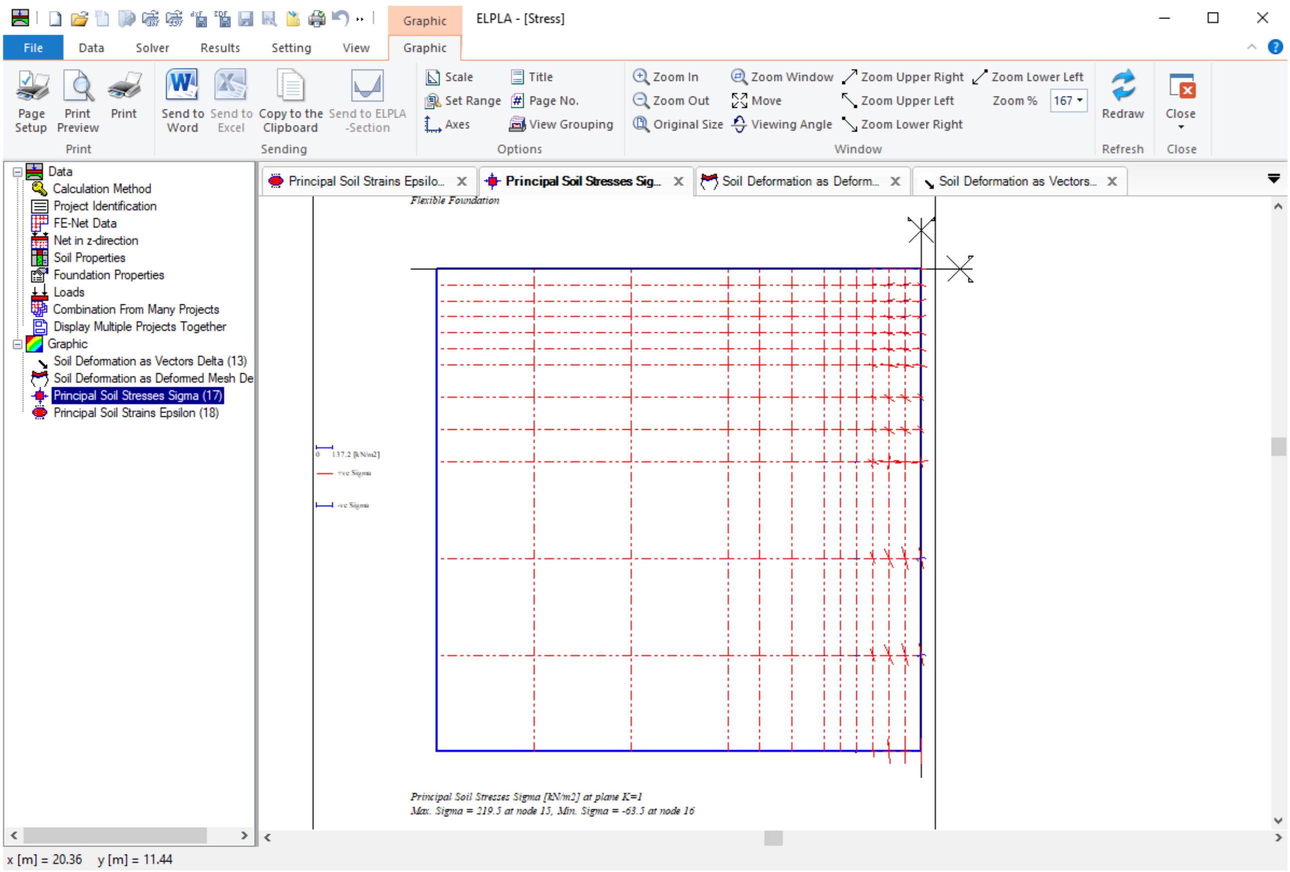


Figure 75 Principal stresses as streaks

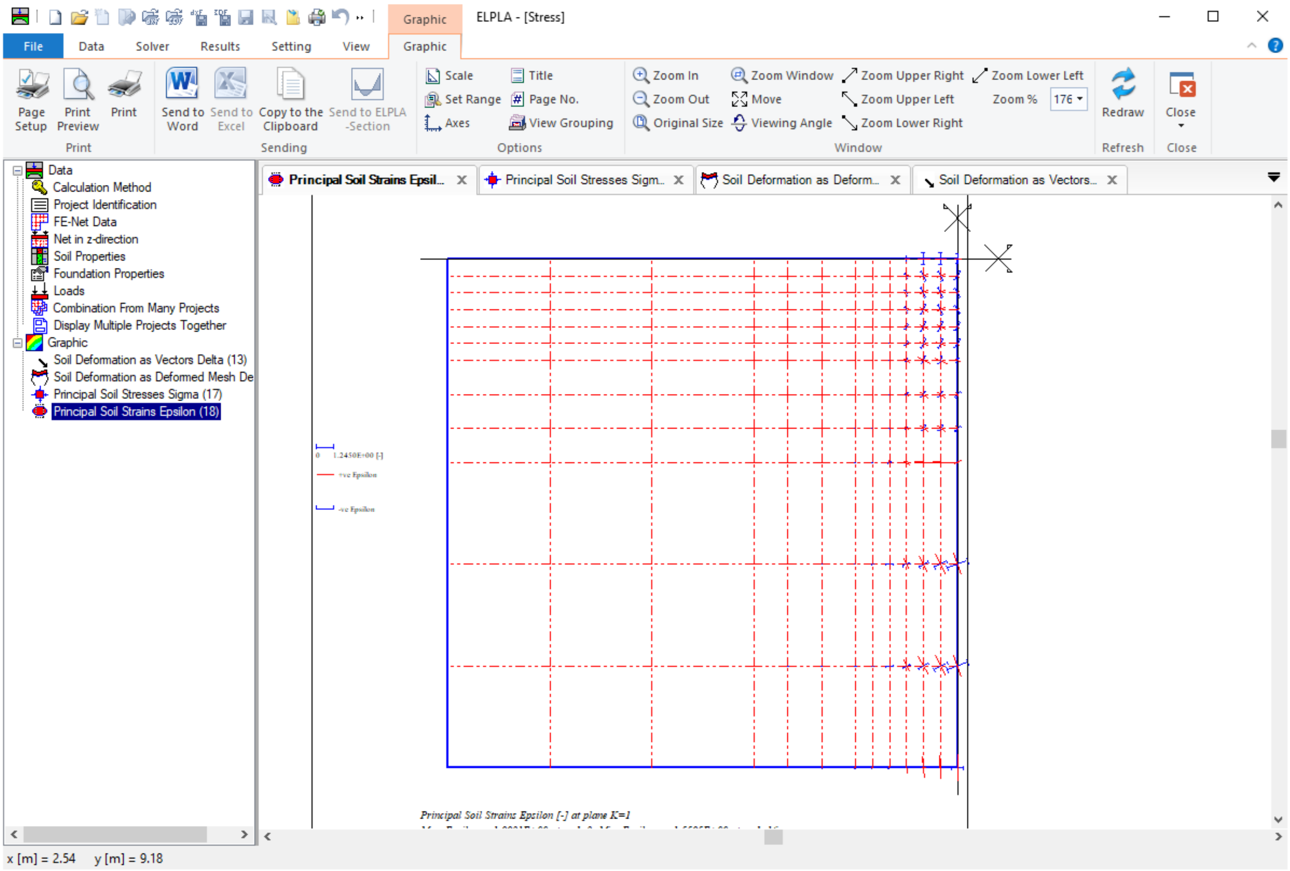


Figure 76 Principal strains as streaks

12 Drawing sections

Furthermore, you can display, plot, and print results at specified sections graphically using the "Section" tab. It is possible to draw settlements, contact pressures, deformation, internal forces, modulus of subgrade reaction, and reinforcement (Figure 77 to Figure 81). 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 the x-direction
- Overlapping in x-direction
- Section in y-direction
- Max./ Min. values in the 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.

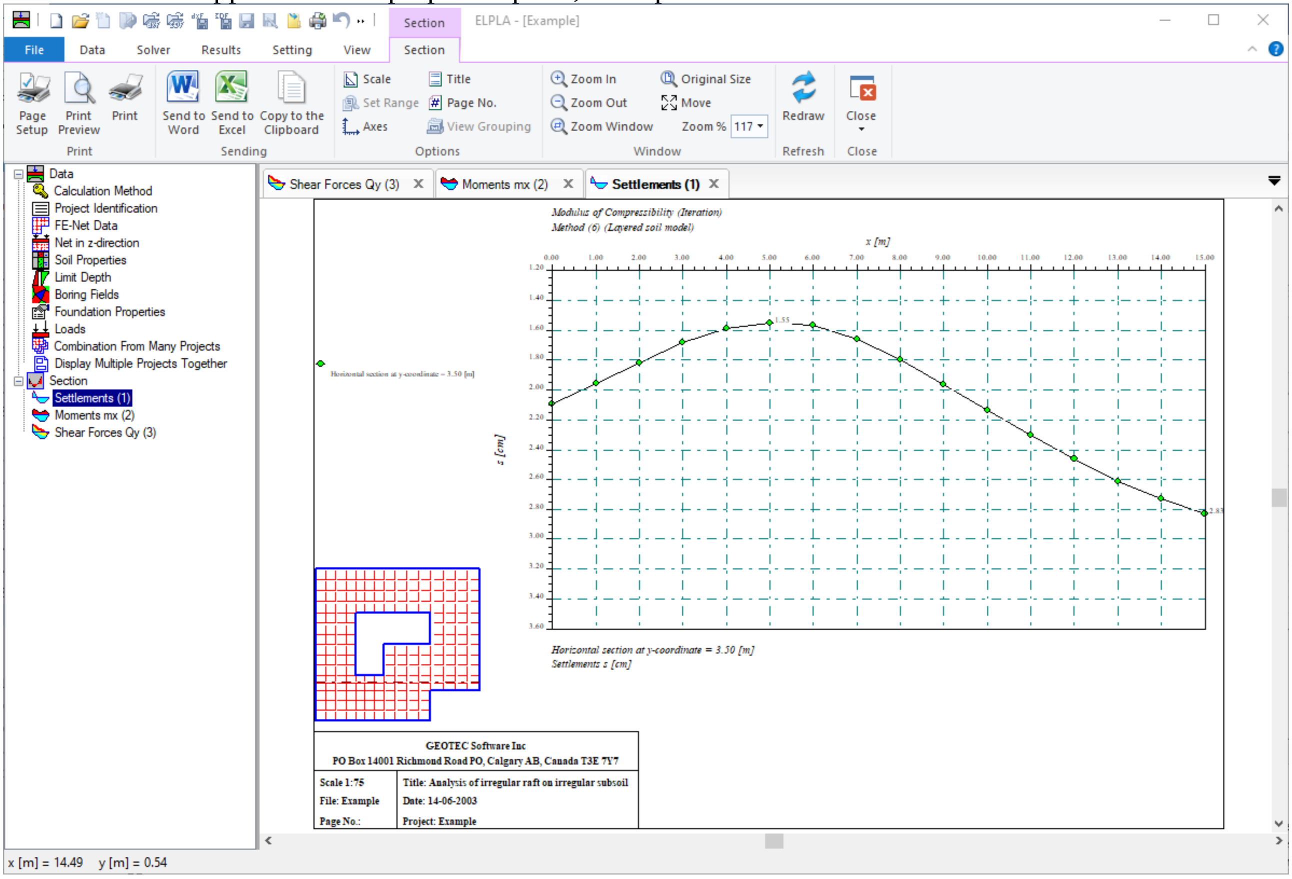


Figure 77 Results can be plotted at the specified section

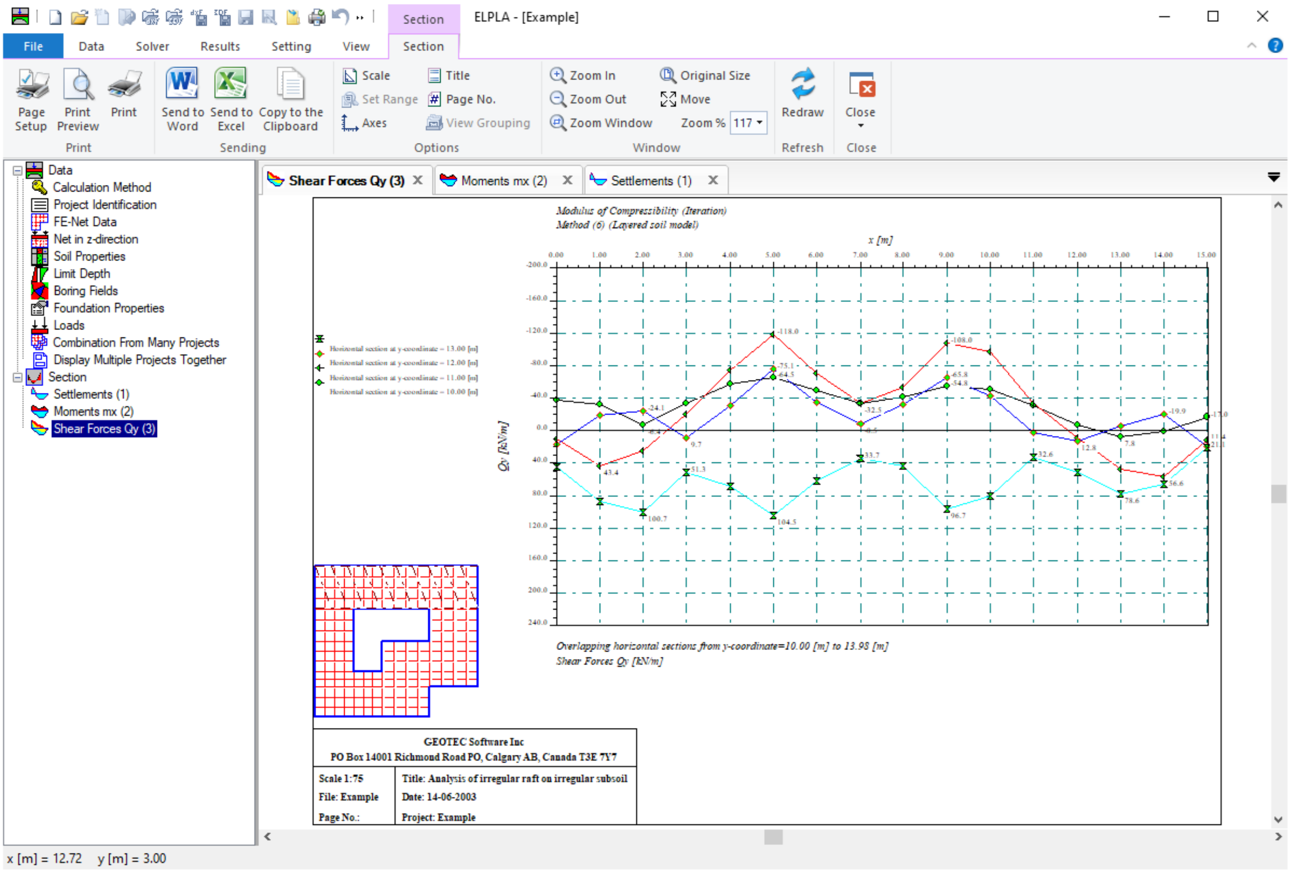


Figure 78 Results from a specified zone can be plotted together

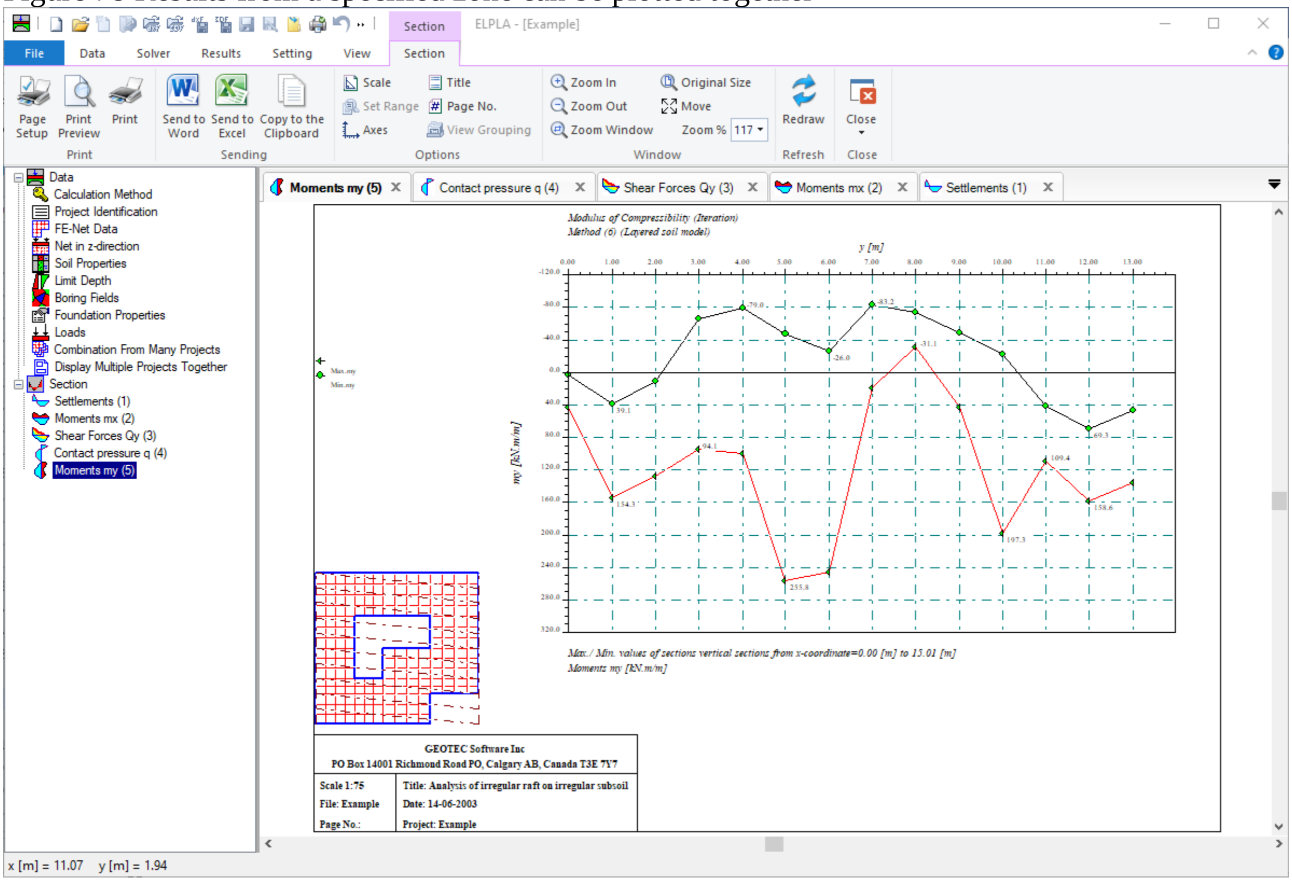


Figure 79 Max. and Min. values can be calculated and plotted together

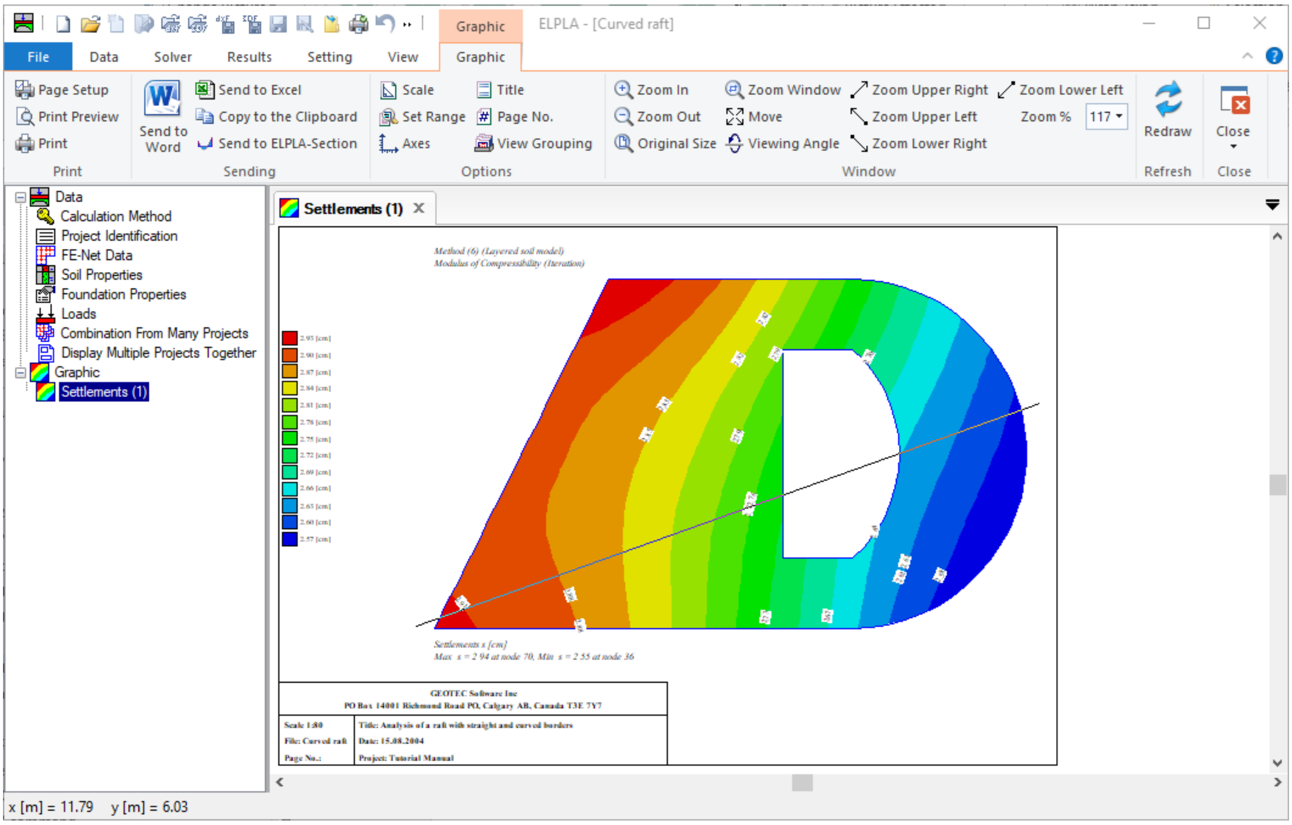


Figure 80 specify the arbitrary section

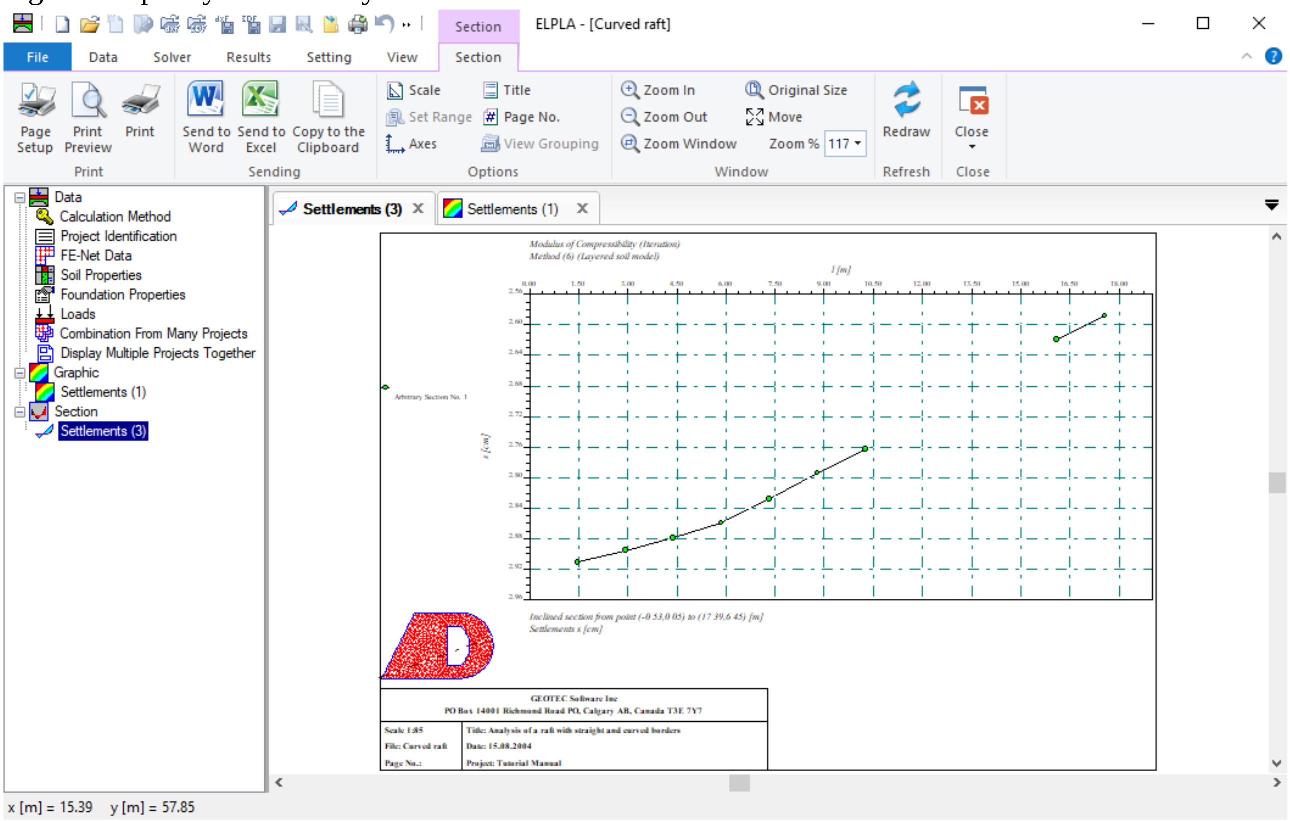


Figure 81 Specified arbitrary section

13 Tabulation of data and results

You can list data and results using the "List" tab. Listing the data and results can be displayed first on the screen and then can be sent to the printer (Figure 82 to Figure 83). 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.

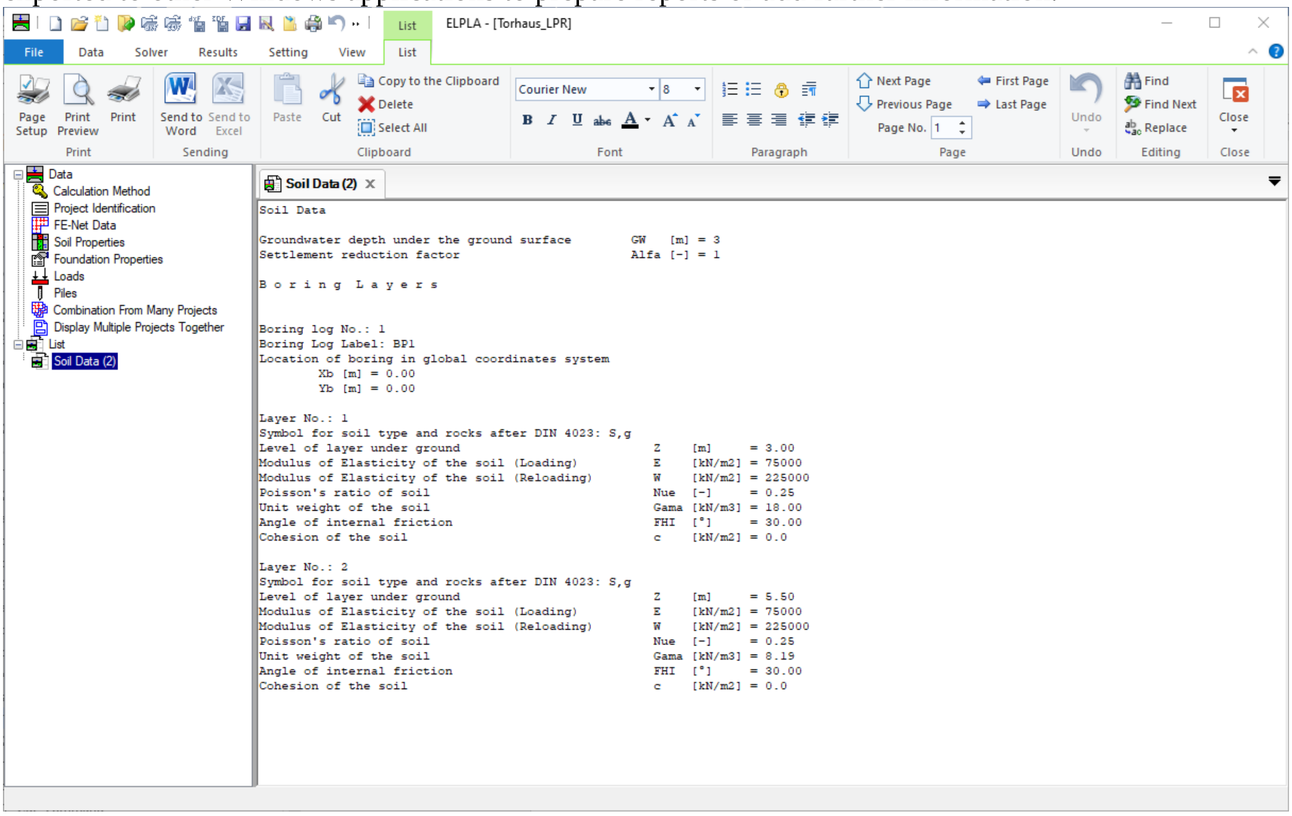


Figure 82 Data can be tabulated

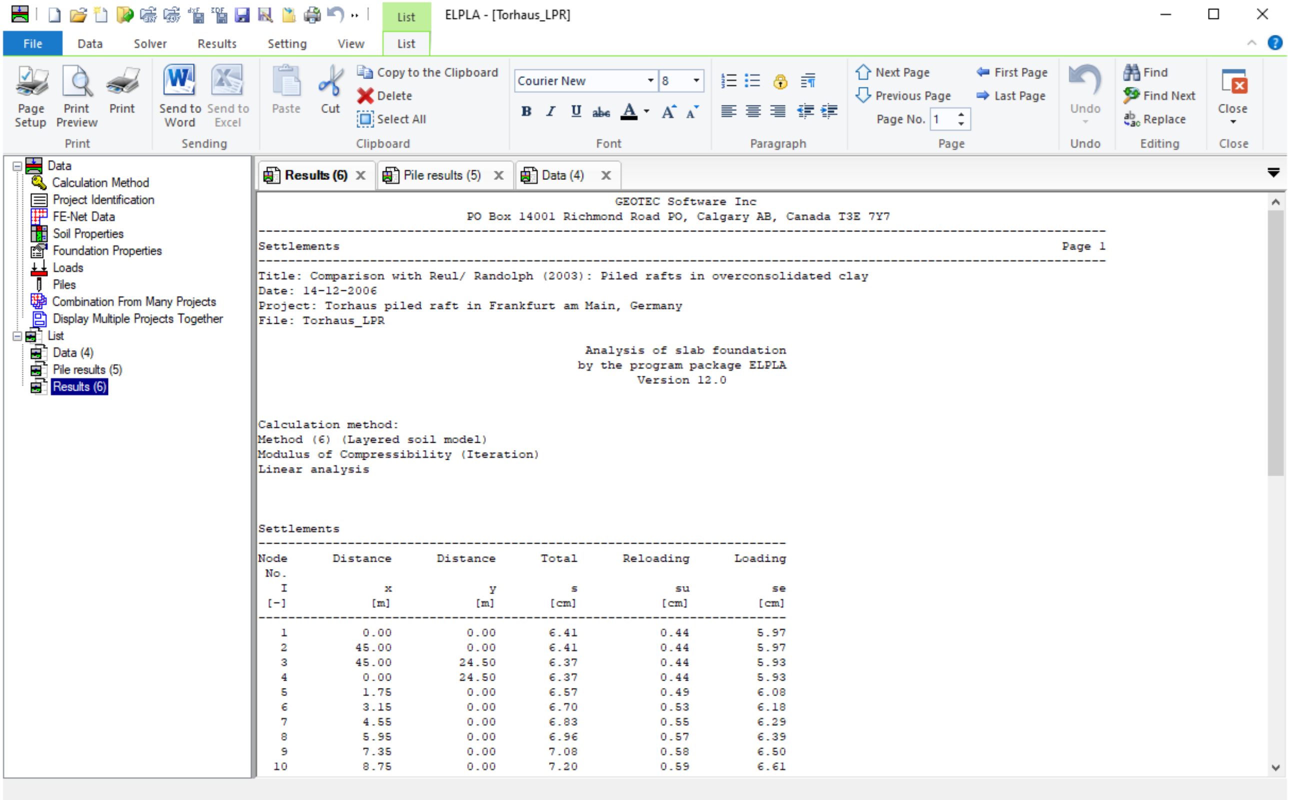


Figure 83 Results can be tabulated

14 Display Multiple Projects Together in a Single View

ELPLA allows the user to display data or results of multiple projects together in a single view. For example, the following figure presents four different projects solved by different methods in the same view.

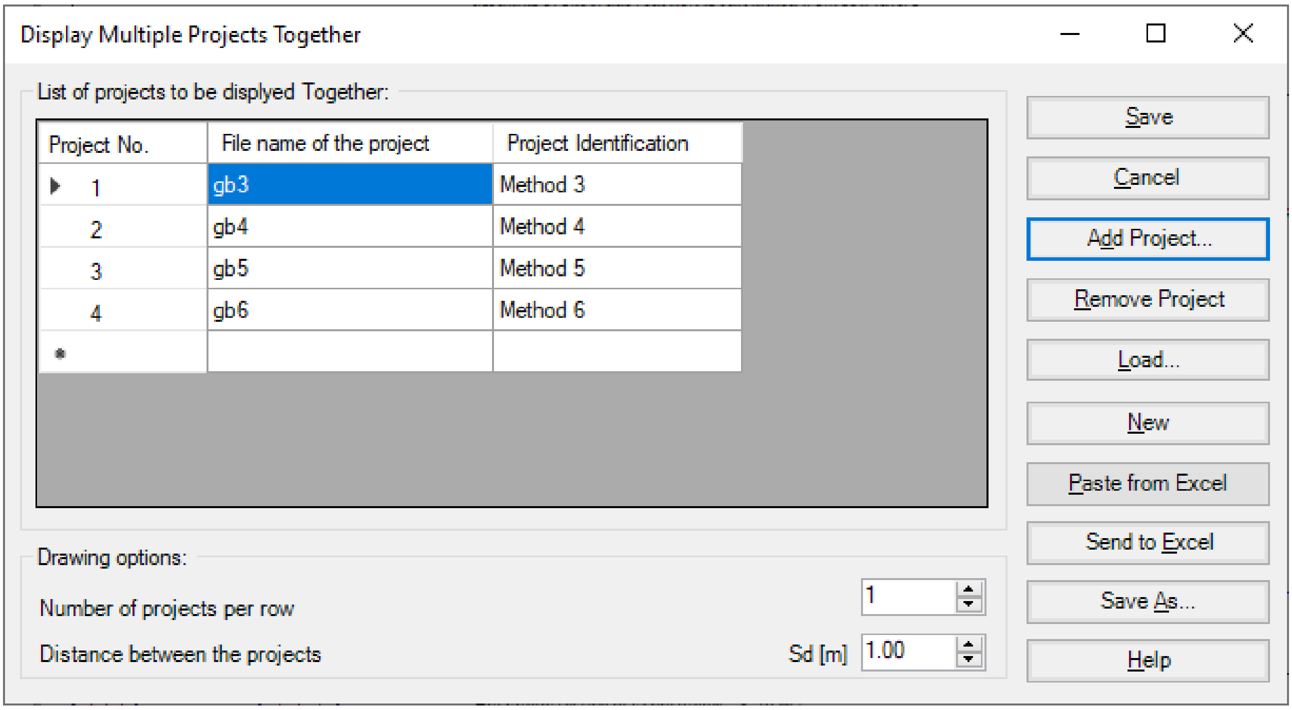


Figure 84 "Display Multiple Projects Together" input Dialog box

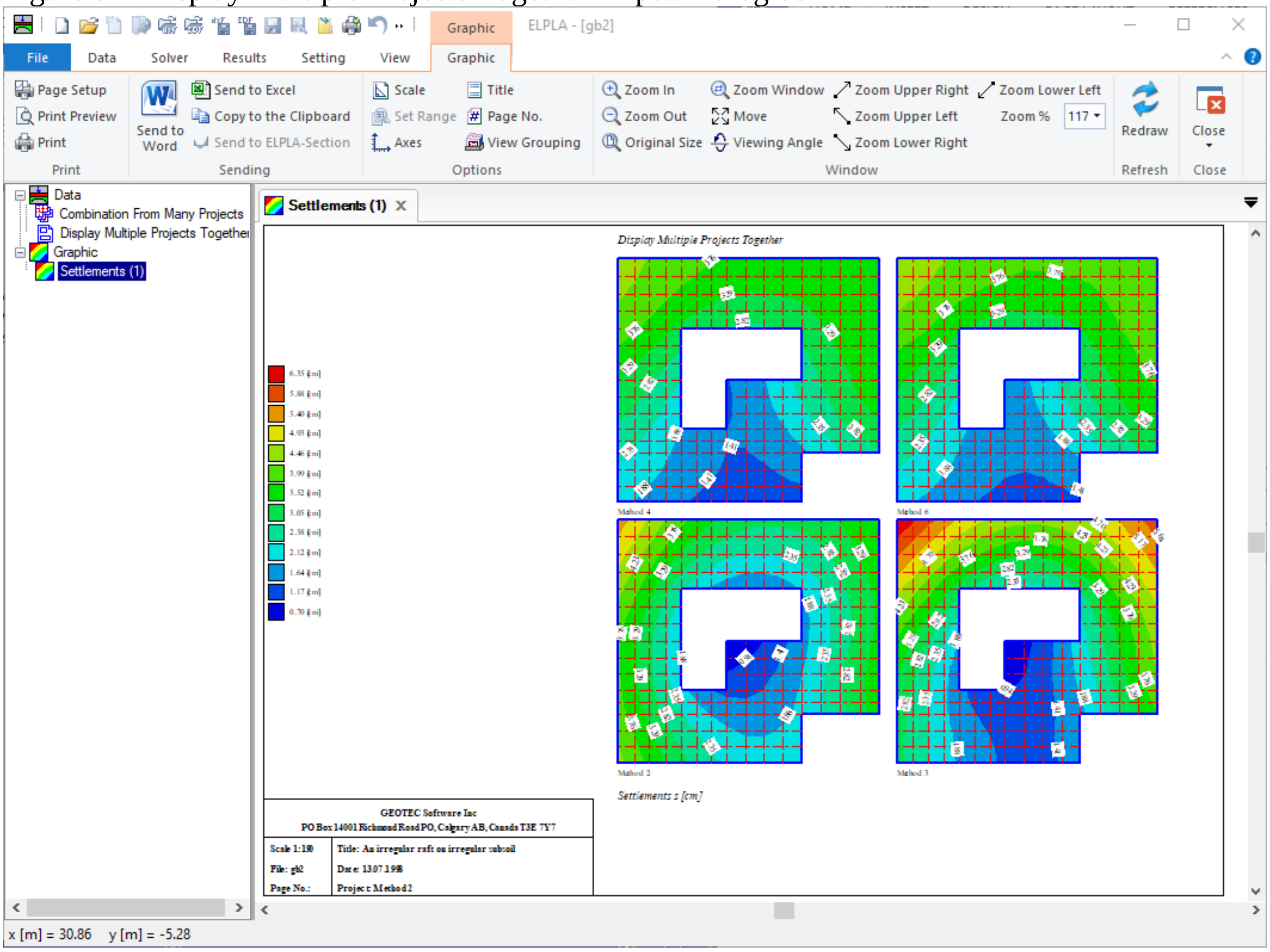


Figure 85 Combination of many projects

15 Combination of Results of Many Projects in Diagrams

The option combination from many projects allows the user to display diagrams from different loading cases, soil models, calculation methods, slab properties, boundary conditions, and special cases at a specified section or area in one chart.

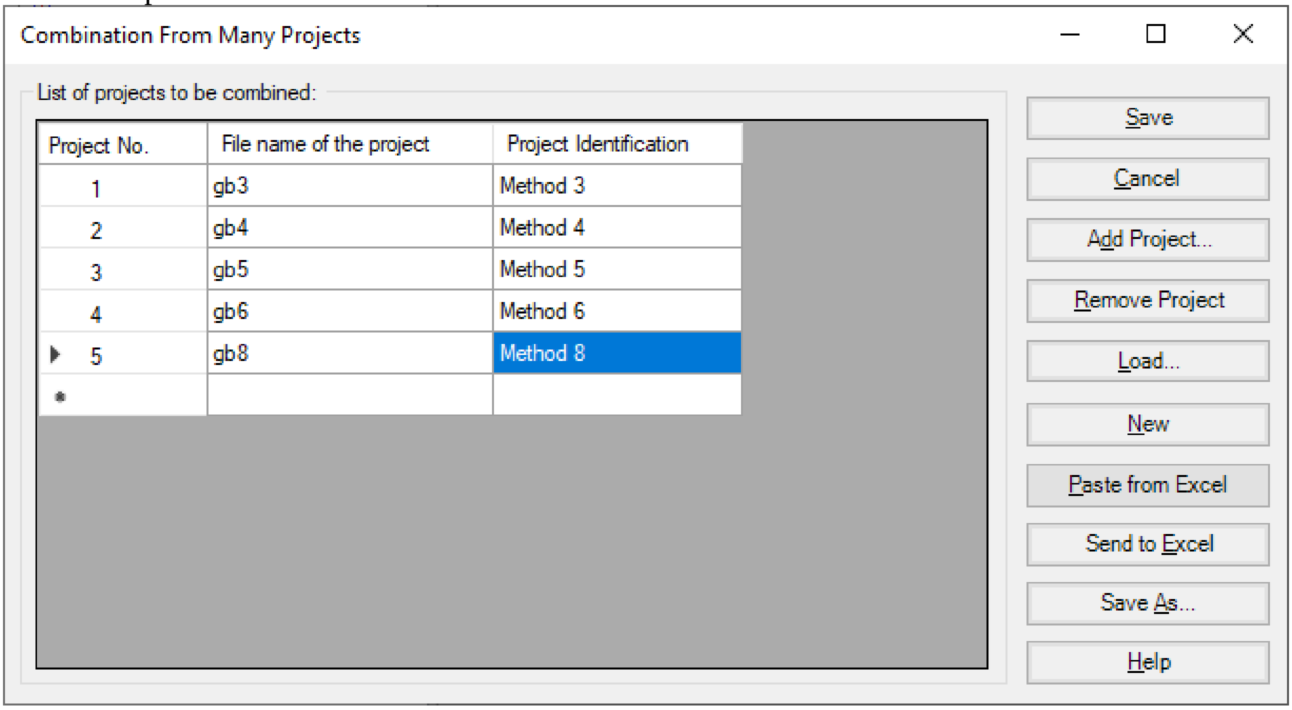


Figure 86 "Combination of many projects" input Dialog box

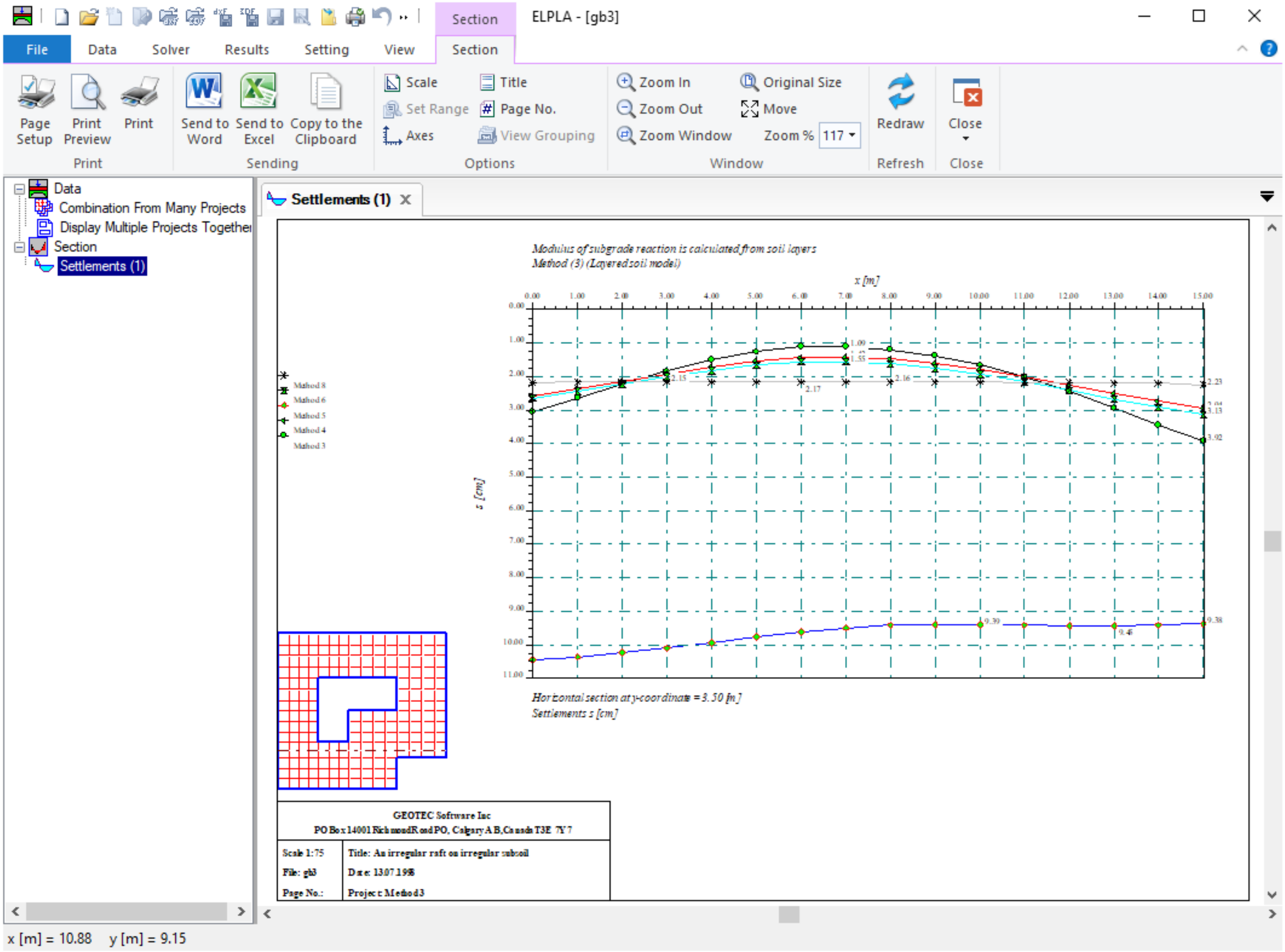


Figure 87 Combination of many projects

16 Export Project Elements to BIM

Object Viewer option allows the user to read and view Building Information (BIM) Models of the project in the IFC format.

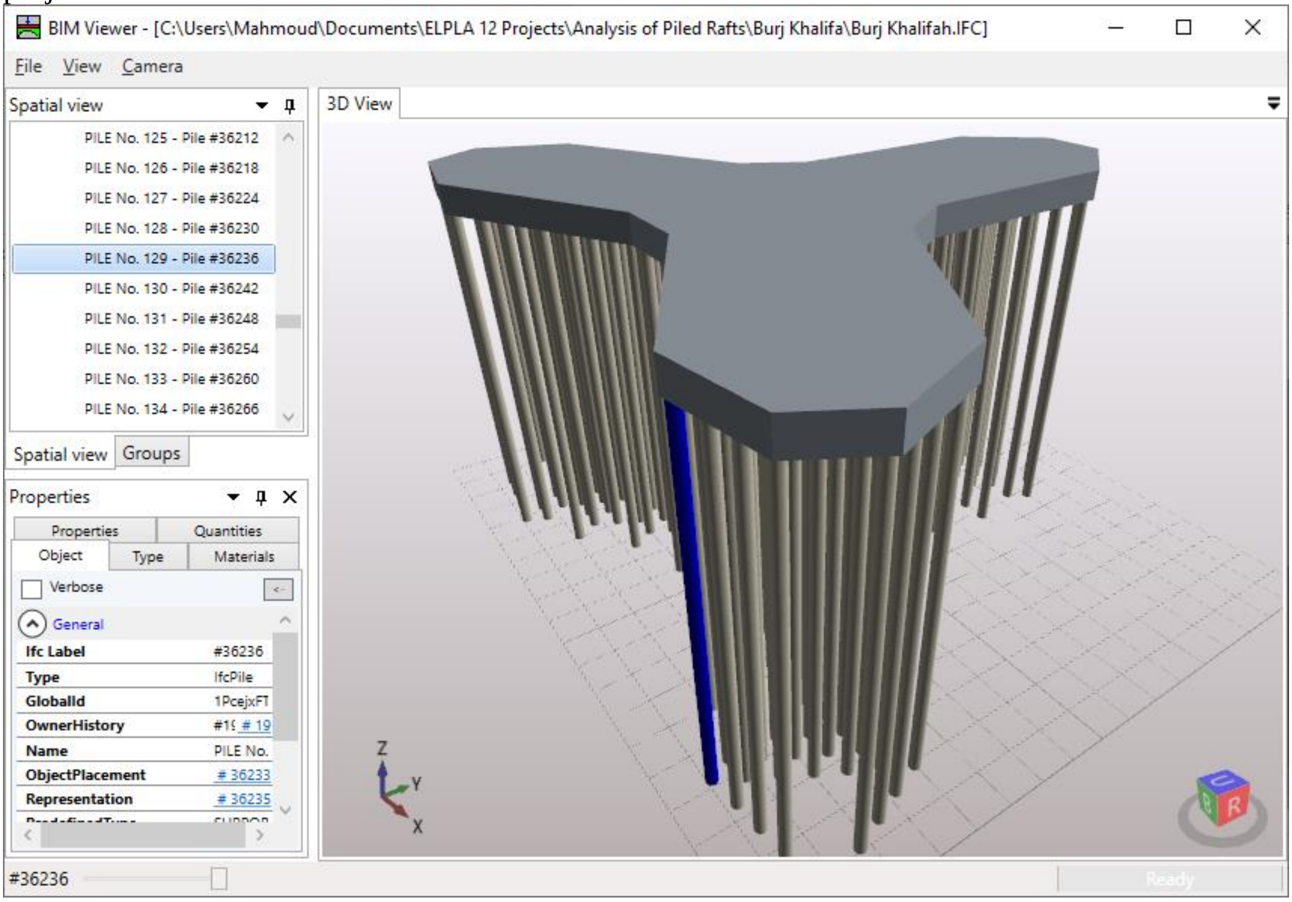


Figure 88 Export Project Elements to BIM system

17 Exporting data and results

- It is easy to export data and results to MS Word as text, tables or drawings, Figure 89.
- It is easy to export data and results to MS Excel as tables or diagrams, Figure 90.
- By "Copy to the clipboard" 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 91.
- The user can create a DXF-File from the finite element mesh or any graphic in the FE-Net mode (Figure 92). FE-Net in DXF format (AutoCAD Drawing Exchange File), can be created by the "Make DXF-File" command from the "File" menu of the FE-Net, Figure 93.
- The User can create a 3DFACE-File from the finite element mesh of rectangular elements, Figure 94. This option enables the user to further work in the finite element mesh for enhancing it, modifying it, or adding elements. FE-Net in 3DFACE format (AutoCAD Drawing Exchange File), can be created by the "Make 3DFACE-File" command from the "File" menu of the FE-Net, Figure 95.

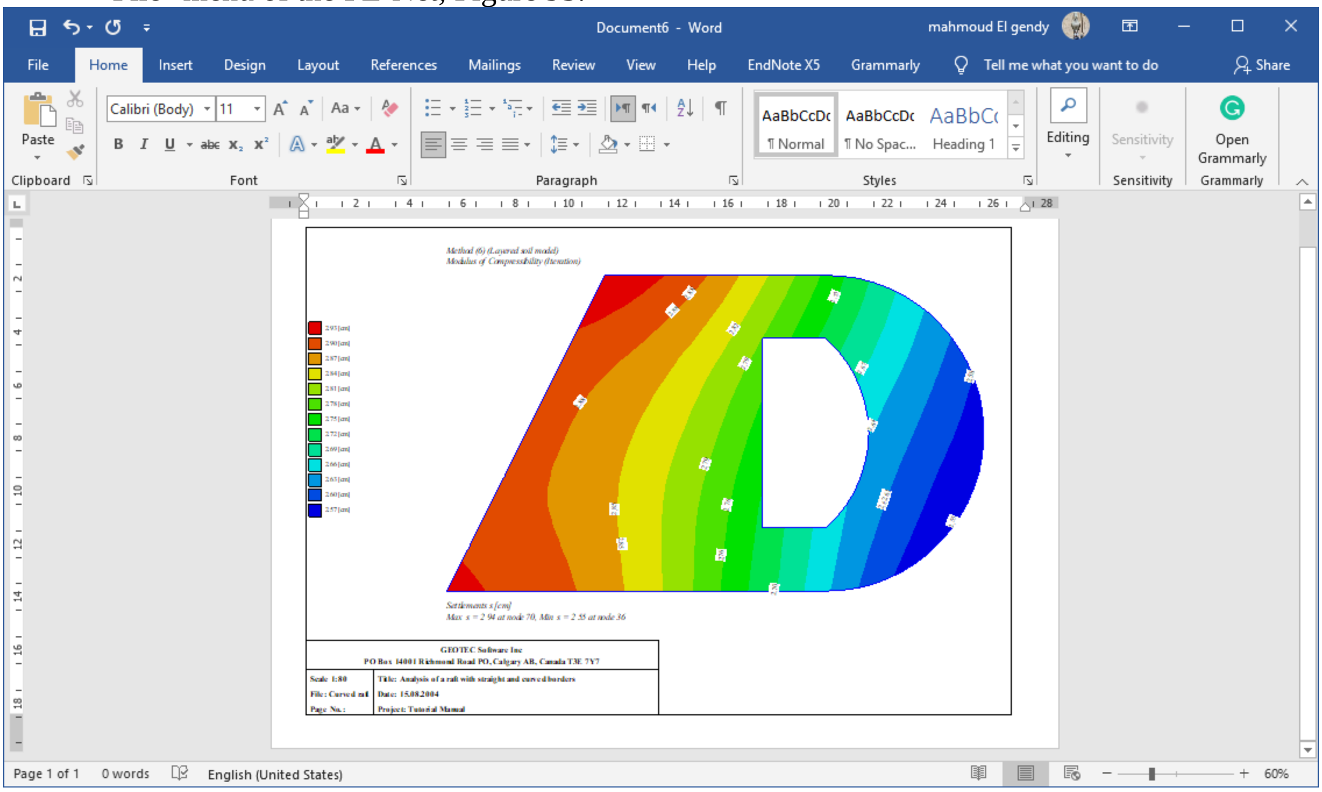


Figure 89 Exporting drawing to Word

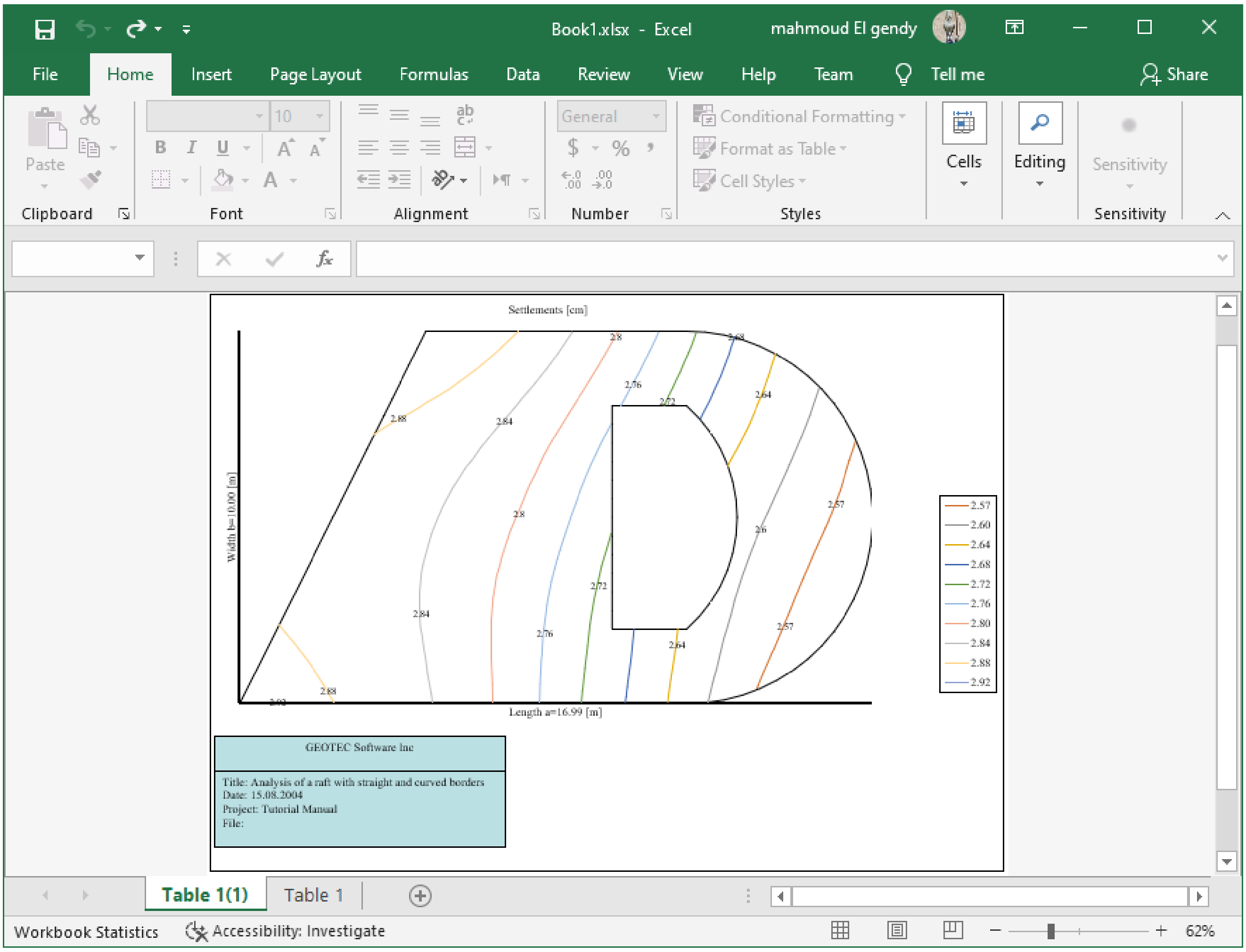


Figure 90 Diagram from ELPLA-Section in MS Excel

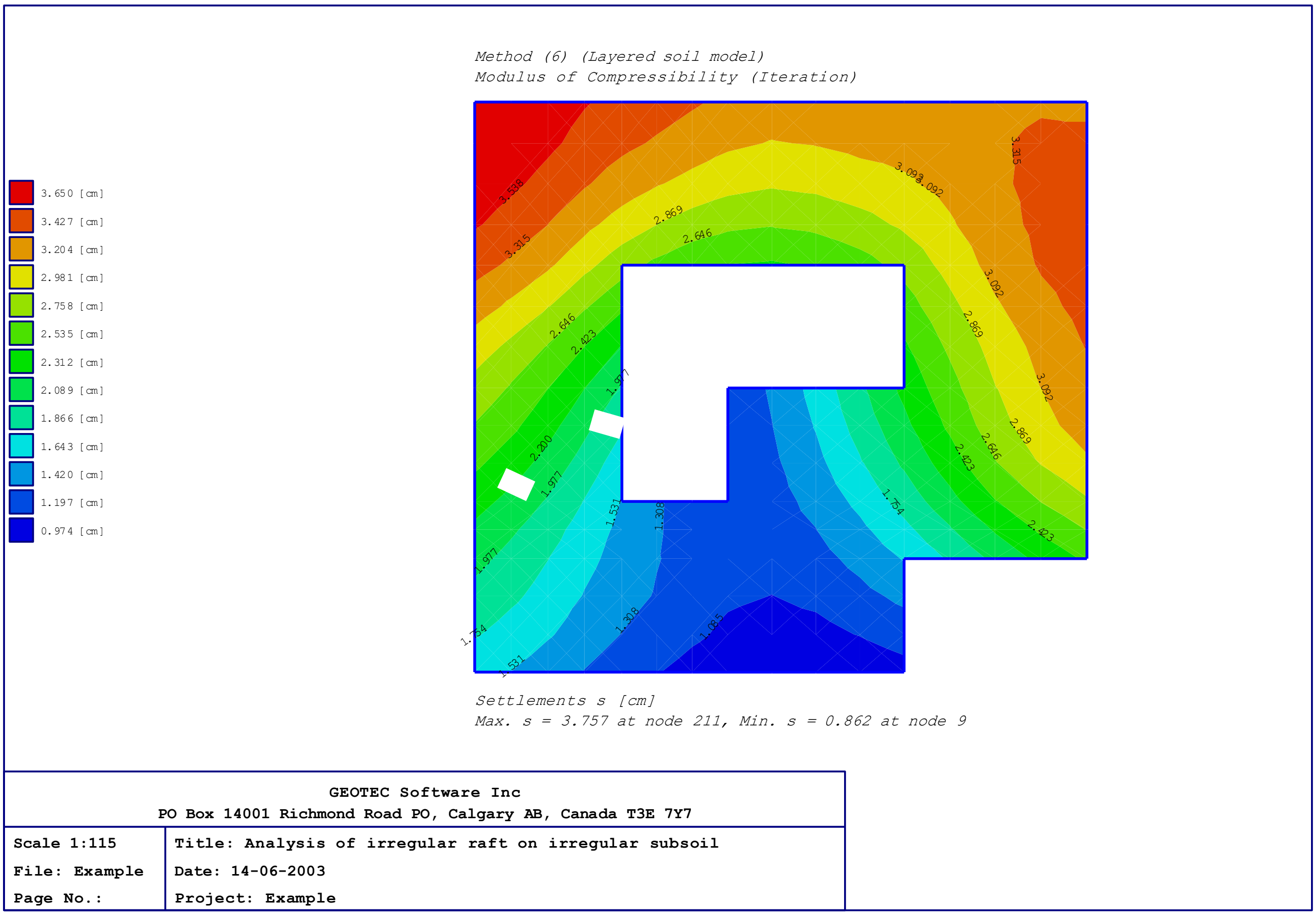


Figure 91 Drawing can be copied directly from ELPLA

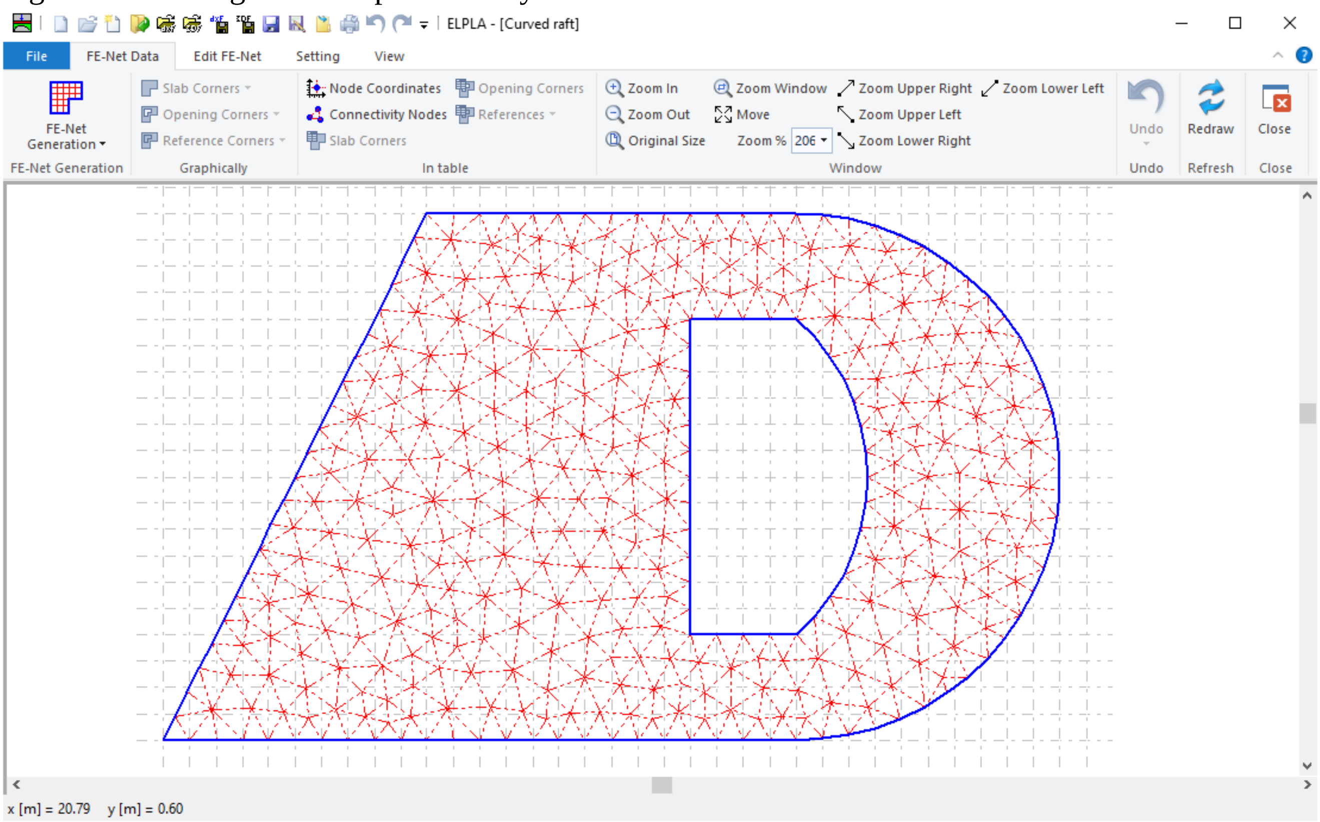


Figure 92 Finite element mesh to be imported into DXF-Format

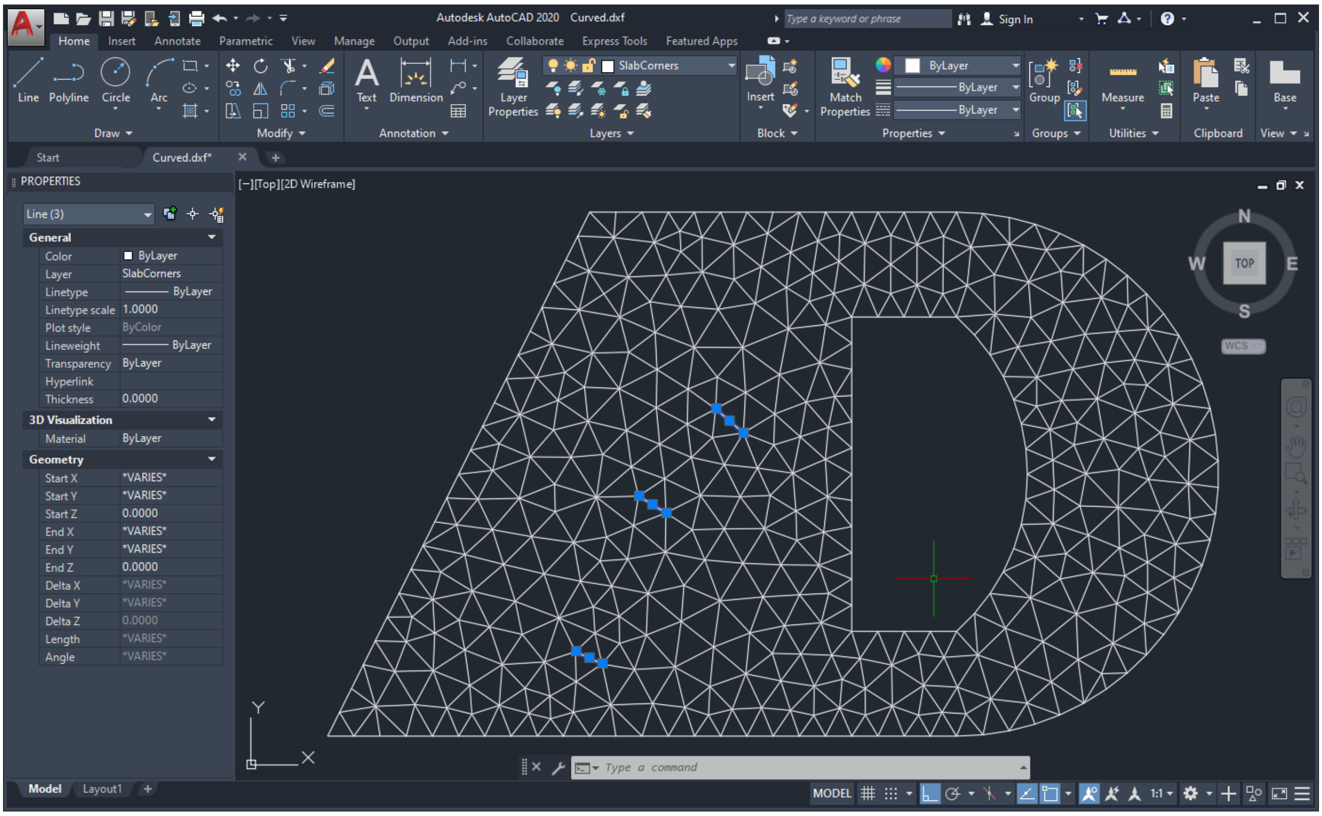


Figure 93 FE-Net is imported into AutoCAD from a DXF-file

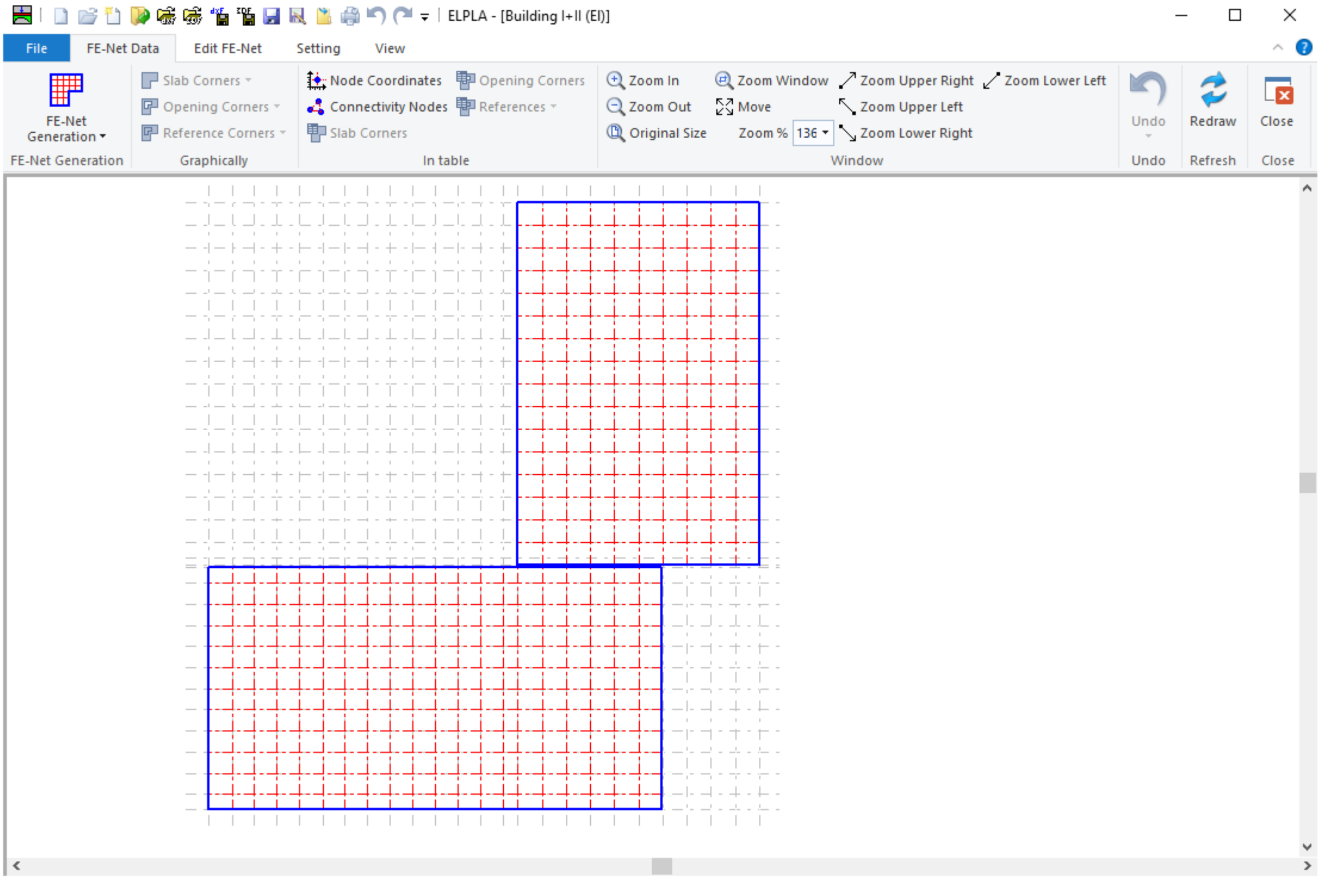


Figure 94 Finite element mesh required to be in 3DFACE-Format

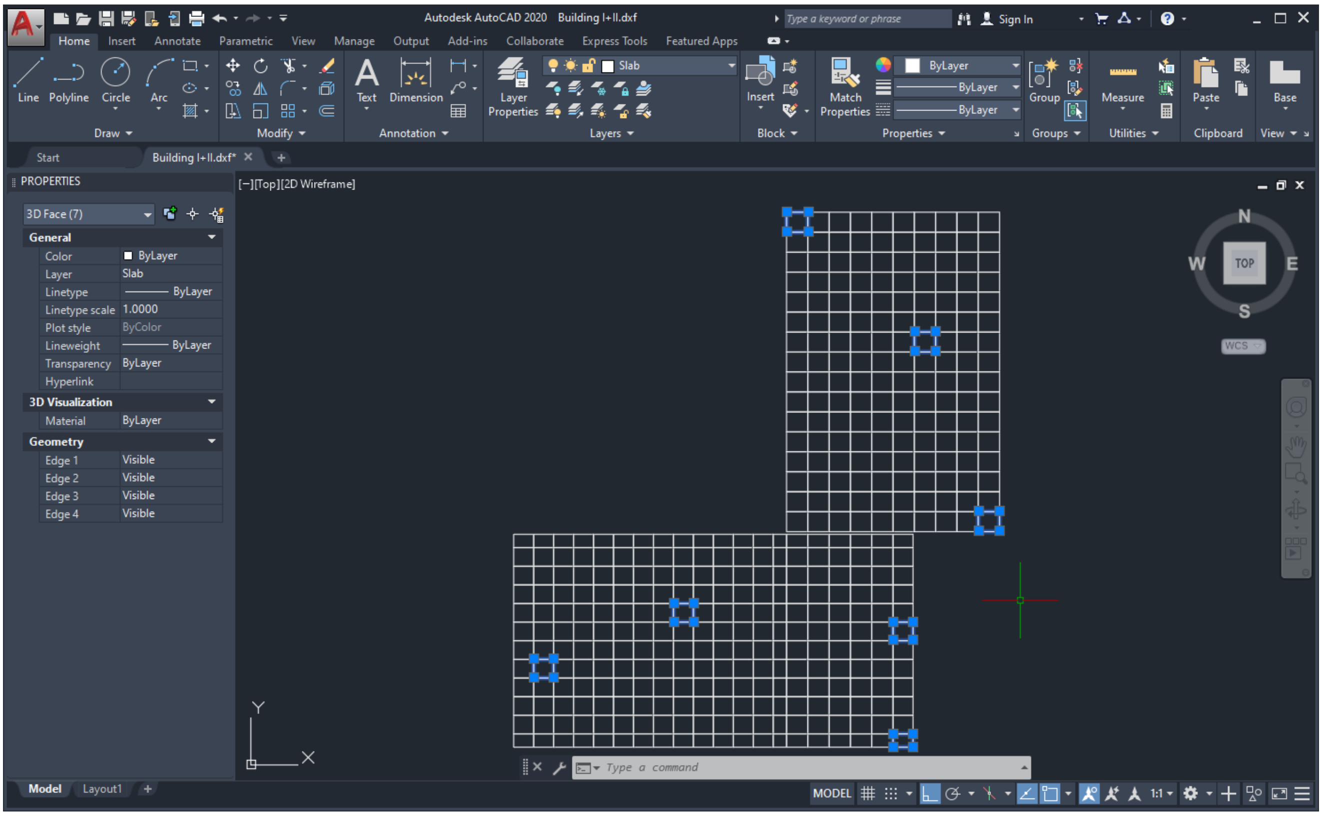


Figure 95 FE-Net is imported into AutoCAD from a 3DFACE file

18 Tips and Tricks

- By clicking the right mouse button on the screen for one of the data or result tabs, the user can also obtain the Popup-Options-Menu, Figure 96.
- In FE-Net Tab, you can click on a node and hold the mouse to move the node to a new location.
- In Graphic Tab, you can click and hold the mouse to move the drawing to a new location.
- By double-clicking on legend, firm header, title, or project identification, the corresponding dialog box appears.
- By double-clicking on the scale in the identification box, the "Scale"-Dialog box appears.
- By double-clicking on the file name in the identification box, the "Open"- Dialog box appears.
- By double-clicking on page No. in the identification box, the "Page No."- Dialog box appears.
- By double-clicking on a specified node on the FE-Net the corresponding node information appears in Figure 97.

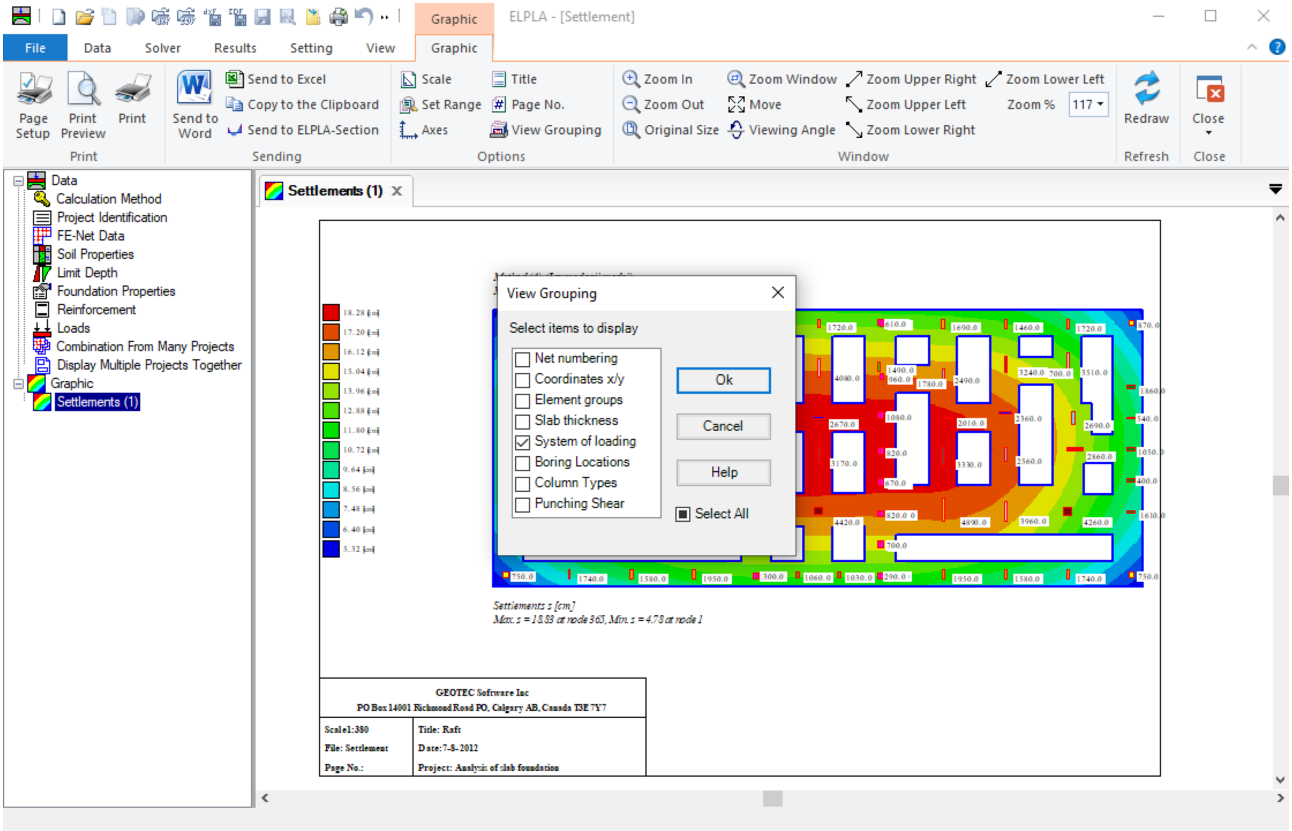


Figure 96 Menu "Popup-Options"

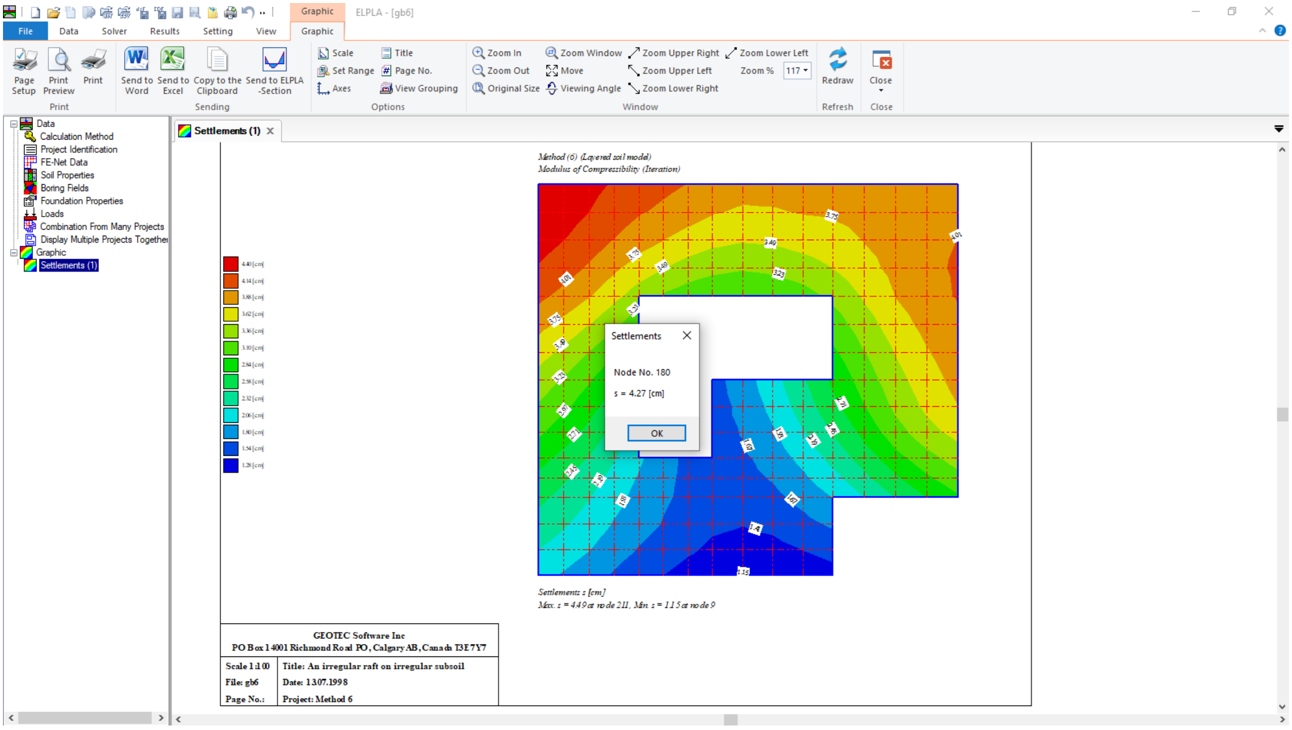


Figure 97 Node information

19 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, rafts, and piled raft according to codes ACI, EC 2, DIN 1045 and ECP
- Determining stresses, strains, and displacements in soil
- Analysis of plane frame
- Analysis of plane stress
- Analysis of grid
- Analysis of a system of many slab foundations
- Analysis of rotational shell
- Analysis of axisymmetric stress
- Analysis of axisymmetric structures
- Analysis of cylindrical tanks
- Analysis of beam on elastic foundation
- Analysis of grid on elastic foundation
- Dynamic analysis of beams
- Dynamic analysis of trusses
- Dynamic analysis of grids
- Dynamic analysis of space frames
- Dynamic analysis of shear walls
- Dynamic analysis of floor slabs
- Dynamic analysis of axisymmetric solids

20 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.