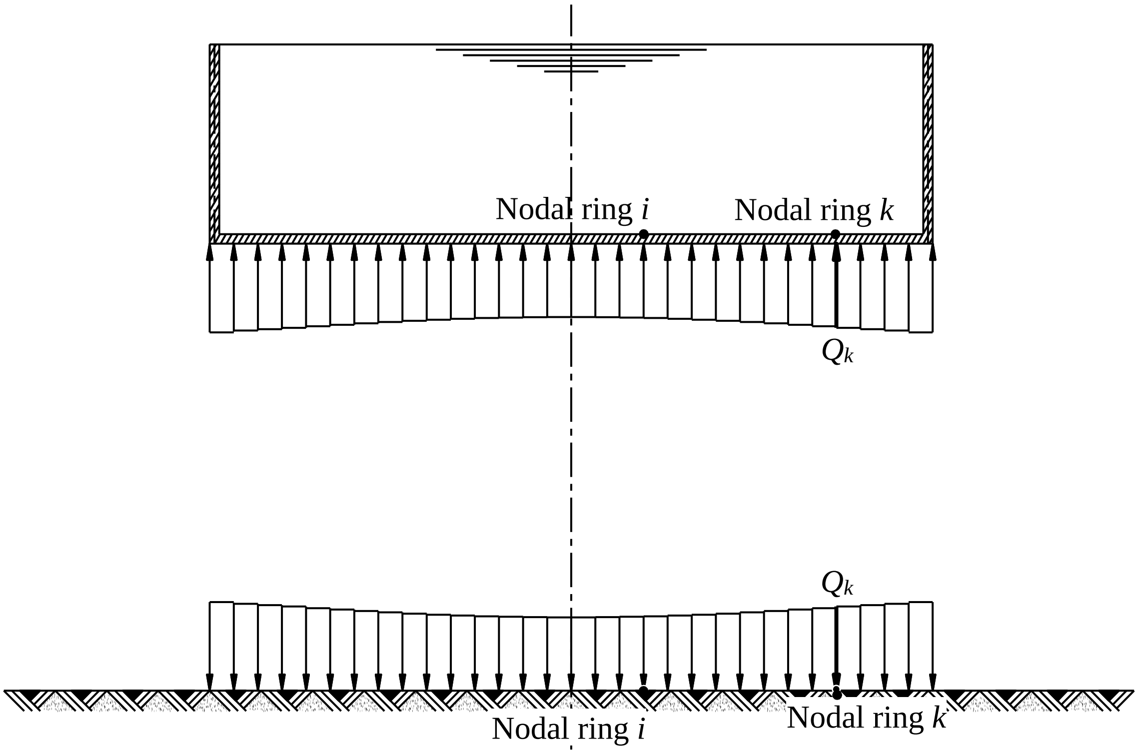
Analyzing Axisymmetric Structures and Tanks by the Program *ELPLA*

Part III: Tutorial Examples



O. El Gendy
Y. El Gendy
M. 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

Contents Page

1. Introduction

This Tutorial Manual contains an overview of dealing with analyzing circular cylindrical shells structures. It is also considered, circular cylindrical tank resting on layered soil as one unit taking into account the soil-structure interaction effect. It describes the processes of modeling the problems, carrying out the analyses, viewing and printing the results. It provides the user skills, which he needs to use *ELPLA*. It also takes the user gradually through some simple examples. Carrying out these examples will help the user to become familiar with the most important functions of *ELPLA* for analyzing shell structures. Before attempting a real project with *ELPLA*, it is recommended that the user tries to carry out the given problems.

This Tutorial Manual will not present the theoretical background of modeling the problems. For more information concerning the methods of analysis, a complete reference for calculation methods and numerical models is well documented in the books:

1- Analysis of Axisymmetric Structures and Tanks by the Program *ELPLA*,
Part I: Numerical models

2- Analysis of Axisymmetric Structures and Tanks by the Program *ELPLA*,
Part II: Verification Examples

In addition, a complete reference for all menus and dialog boxes of the program is to be found in the User's Guide or in the online help system.

ELPLA General Features

ELPLA general features are common features that may be repeated under different menus or command groups. They are listed in *ELPLA* user's guide. Common features are similar those of Windows CAD and Rendering programs. If you are familiar with Windows CAD and Rendering programs, you can certainly get a head start from Tutorial Examples.

2. Starting *ELPLA*

After installing *ELPLA*, a new program group and program items will be created automatically for *ELPLA* in the Windows-Start-Menu. In addition, a program icon will be created on the Windows desktop. The usage of the program is typically such that first data files are created describing a certain problem through the "Data" Tab. Then the project problem is analyzed by calculation commands through "Solver" Tab. Finally, results can be presented as graphical drawings, graphs and tables through "Result" Tab. Names and short descriptions of the function of the main program Tabs are given in Table 1.

Table 1 Names and descriptions of *ELPLA* Tabs

Tab name	Description of the tab function
"Data" Tab	Editing project data
"Solver" Tab	Analyzing the project problem
"Result" Tab	Displaying data and results graphically, listing project data and calculated results, displaying results in diagrams at specified sections and displaying boring logs graphically
"Setting" Tab	Setting options, formats, main data of *ELPLA*
"View" Tab	Appearance or disappearance "Tree View" and "Status bar"

Start the *ELPLA* by double clicking on its icon on the Windows desktop. After starting *ELPLA* for the first time, the window in Figure 1 appears.

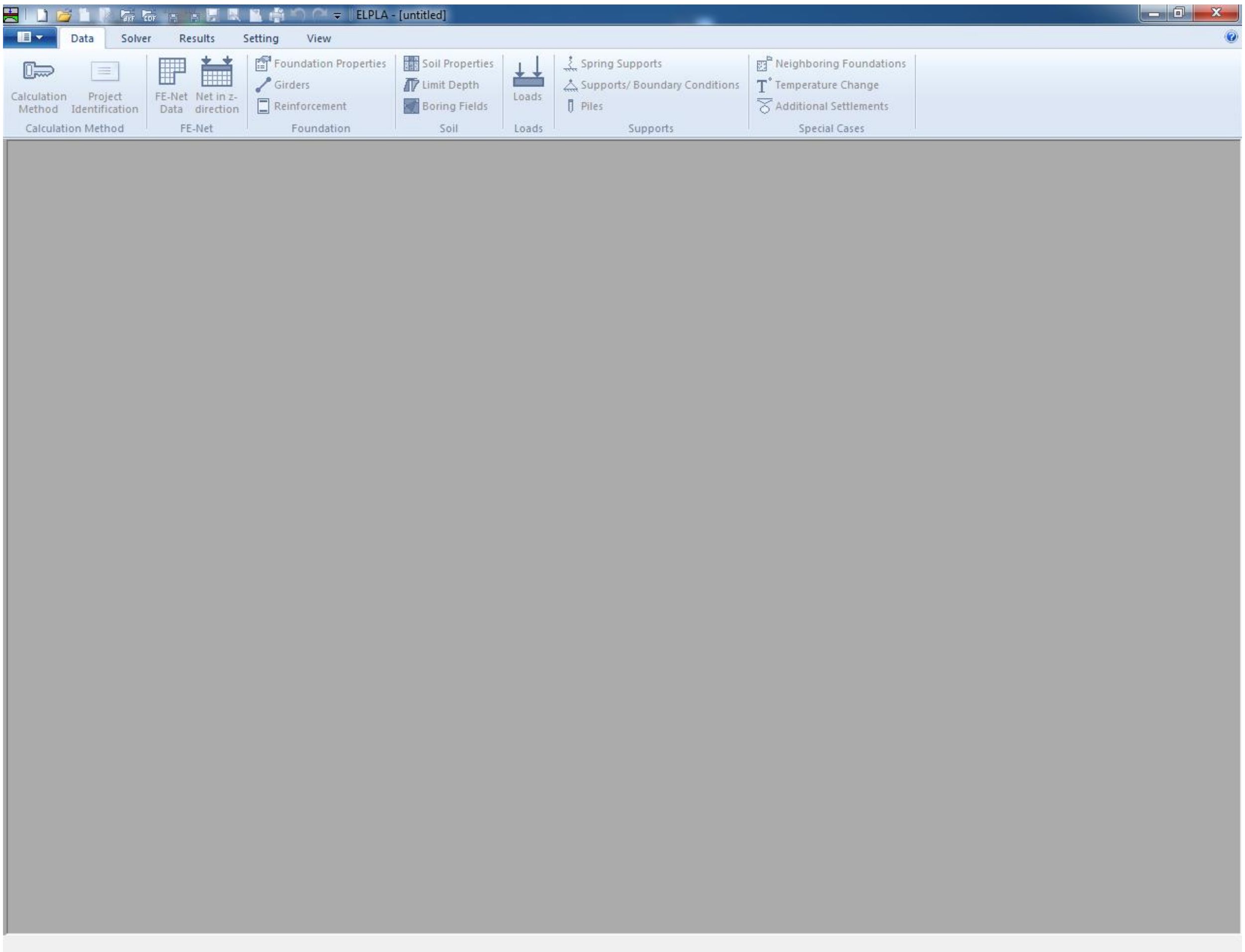


Figure 1 Main window of *ELPLA*

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

3. Toolbar Buttons and Keyboard Shortcuts

Many of *ELPLA* commands can be accessed by clicking buttons on the toolbars, or by pressing shortcut keys on the keyboard. When you select a menu, the available toolbar buttons and keyboard shortcuts for that menu are shown next to their corresponding commands as shown in Figure 2. You can also directly see what command is associated with a toolbar button by holding the mouse cursor over the button. After a brief pause a legend (Screen Tip) will appear next to the cursor showing the menu command associated with the button.

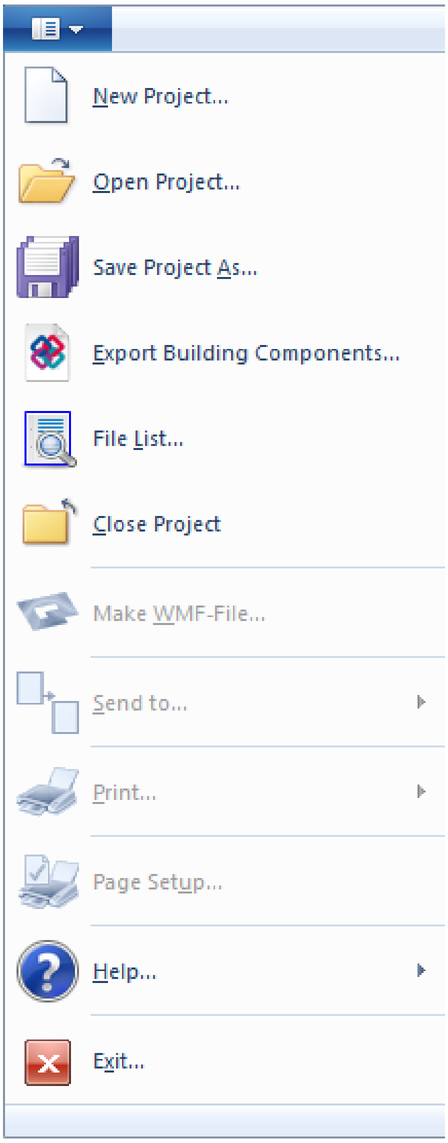


Figure 2 Available toolbar buttons and keyboard shortcuts for the file menu

Switching between individual tabs

The program's tabs appear at the top of the program window. The user can go to a specific tab by clicking the name of that tab. The "Data" tab, which is used for entering the project data, appears in the upper left corner of this window.

Mouse Cursor Modes

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

Node Selection

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

Undo and Redo Commands

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

Snap Tools

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

Defining Data

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

Project identification

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

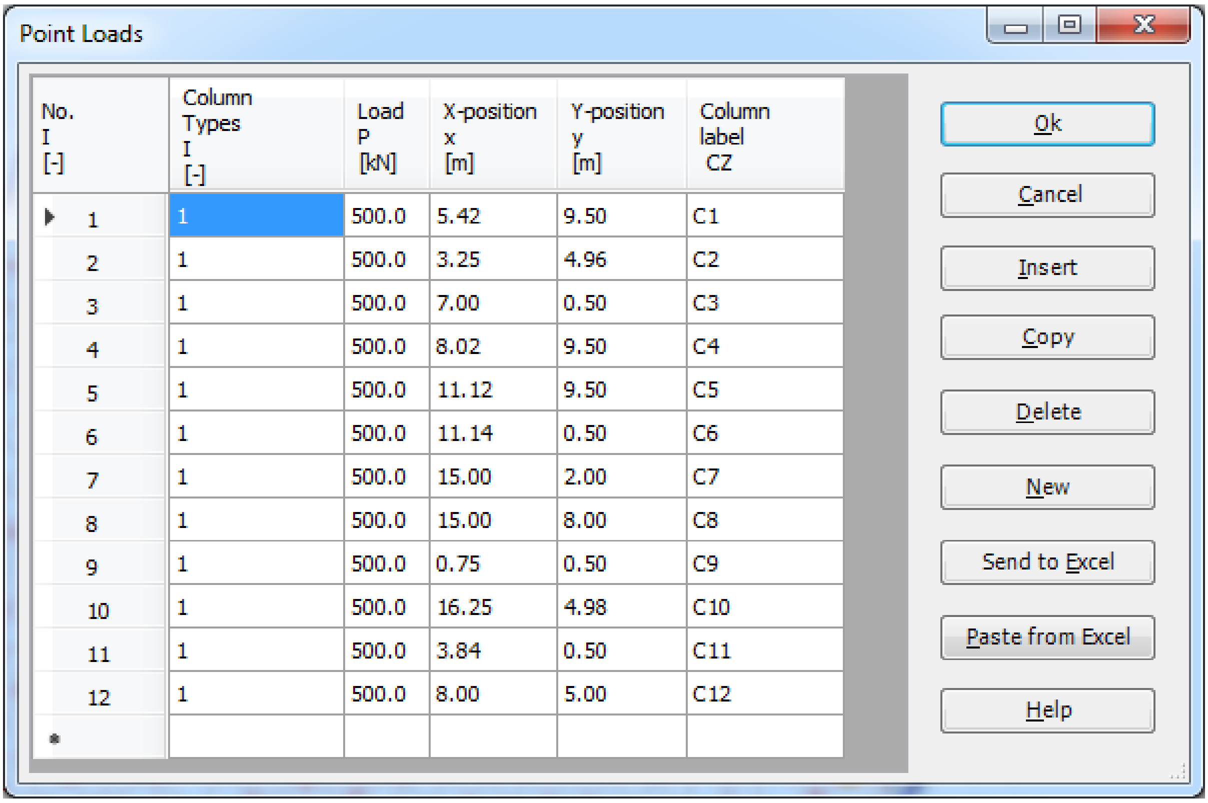
Reinforcement data

The design code parameters such as partial safety factors for concrete strength, steel strength and internal stresses are defined by choosing the "Design code parameters" command from the "Main data" menu in "Setting" Tab.

Reinforcement data such as design code, concrete grade, steel grade and concrete covers are defined by choosing the "Reinforcement" command from “Slab” menu in "Data" Tab. Design code parameters are standard data for all projects, while reinforcement data may be varied from project to another.

Edit List Box

Some *ELPLA* data are defined by list box dialog as shown in Figure 3. In this list box the "Insert", "Copy" and "Delete" commands are applied for the selected row. To define or modify a value in this table, type that value in the corresponding cell, then press "Enter" key.



No. I [-]	Column Types I [-]	Load P [kN]	X-position x [m]	Y-position y [m]	Column label CZ
1	1	500.0	5.42	9.50	C1
2	1	500.0	3.25	4.96	C2
3	1	500.0	7.00	0.50	C3
4	1	500.0	8.02	9.50	C4
5	1	500.0	11.12	9.50	C5
6	1	500.0	11.14	0.50	C6
7	1	500.0	15.00	2.00	C7
8	1	500.0	15.00	8.00	C8
9	1	500.0	0.75	0.50	C9
10	1	500.0	16.25	4.98	C10
11	1	500.0	3.84	0.50	C11
12	1	500.0	8.00	5.00	C12

Figure 3 List box used by *ELPLA*

Analysis progress

During the analysis of problems, all computations and analysis progression according to the defined method are progressively reported. As shown in Figure 4, the analysis progress menu reports the various phases of calculations. In addition, the status bar of "Solver" Tab displays information about the progress of calculation as the program analyzes problems.

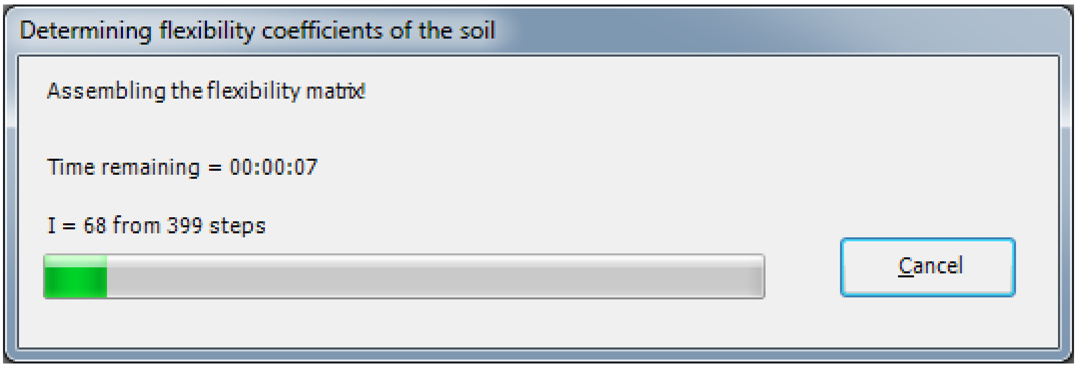
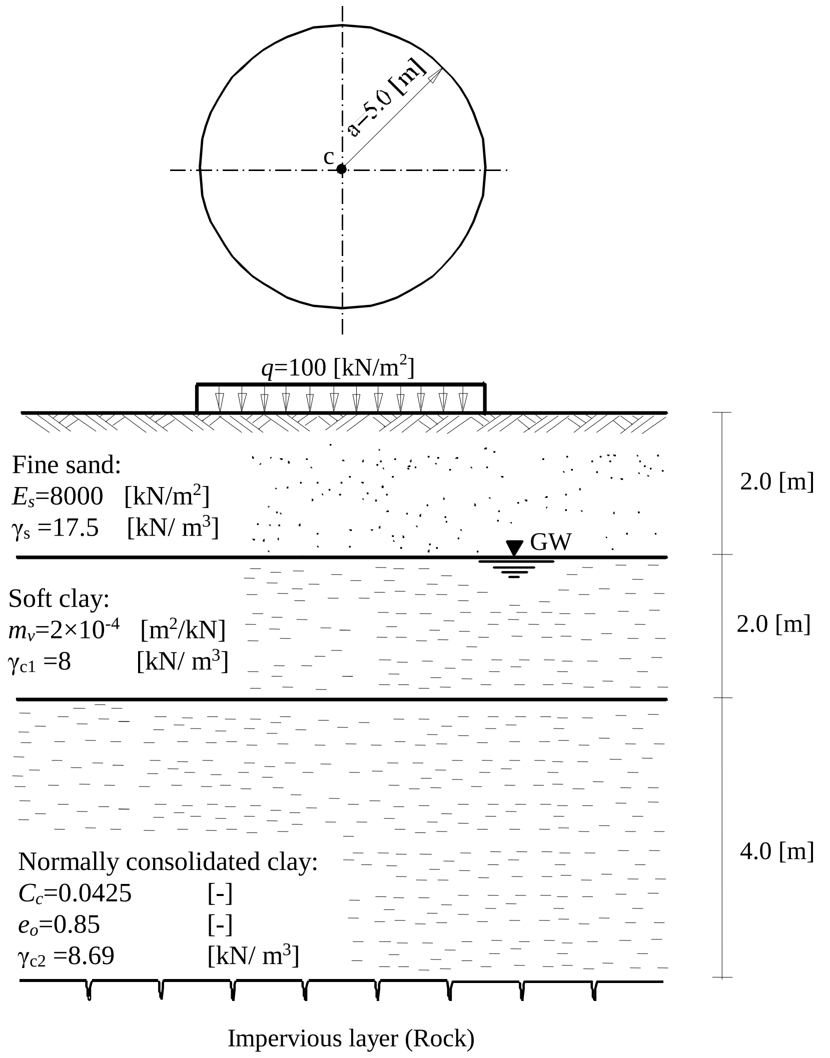


Figure 4 Analysis progress menu

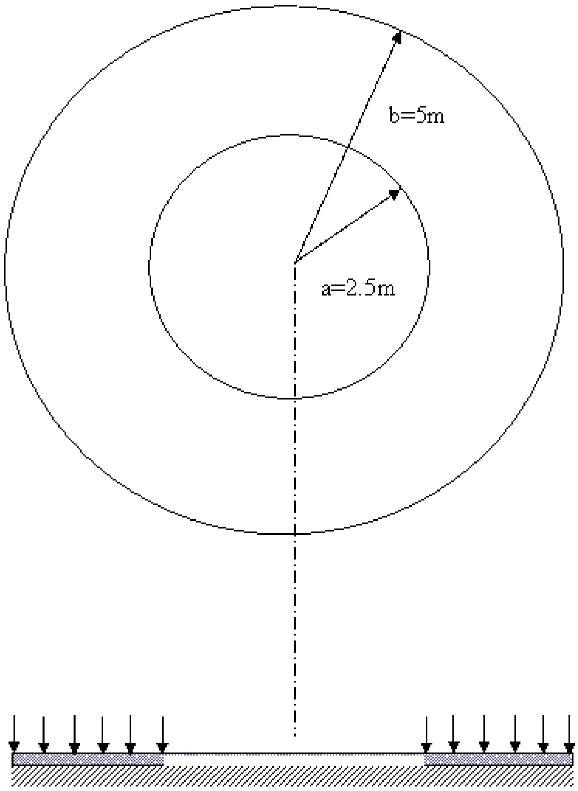
4. Description of the examples

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

Example 1 Analysis of a circular loaded area resting on different soil layers

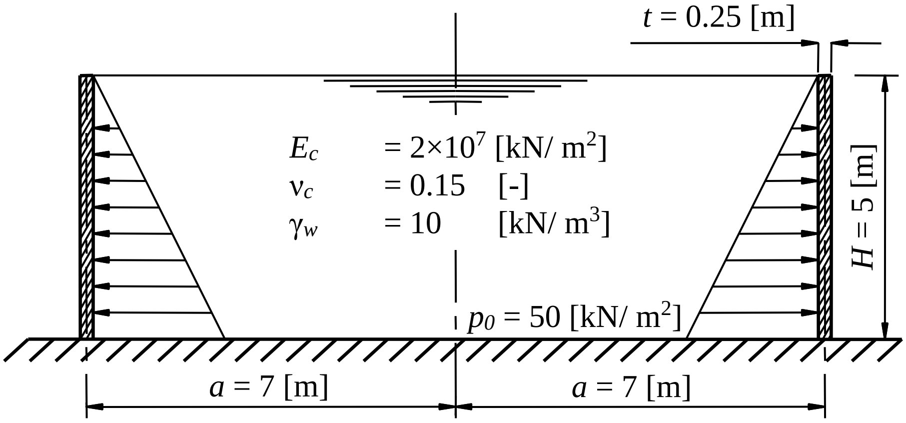
<table>
<tr><td>File</td><td>Soil profile under the circular loaded area</td></tr>
<tr><td>Circular loaded area</td><td>
</td></tr>
<tr><td colspan="2">An example of a loaded area resting on different soil layers is selected to illustrate some features of ELPLA for analyzing shell elements using circular and annular elements</td></tr>
</table>

Example 2 Analysis of an annular plate resting on *Winkler's* medium

File	Annular plate subjected to a uniform load
Annular plate  An example of an annular plate resting on *Winkler's* medium is selected, to illustrate some features of *ELPLA* for analyzing shell elements using circular and annular elements	

Example 3 Analysis of a tank with a fixed base

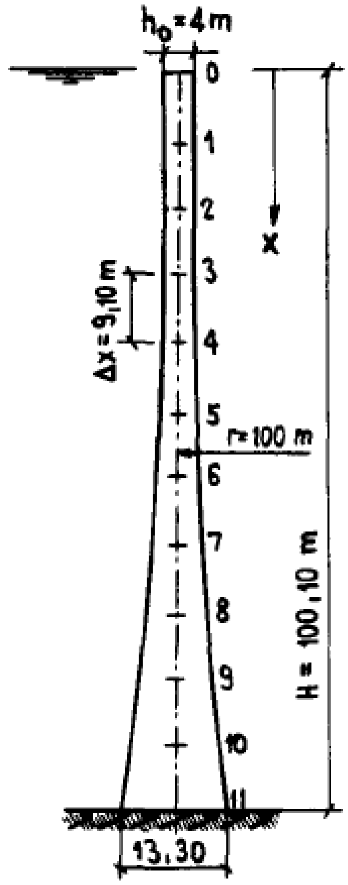
File	Cylindrical circular tank
Tank with fixed base	



An example of an axi-symmetrically circular cylindrical tank with a fixed base is selected to illustrate some features of *ELPLA* for analyzing shell elements

Example 4 Analysis of a reservoir wall with a variable wall thickness

File	Reservoir wall
Reservoir wall	

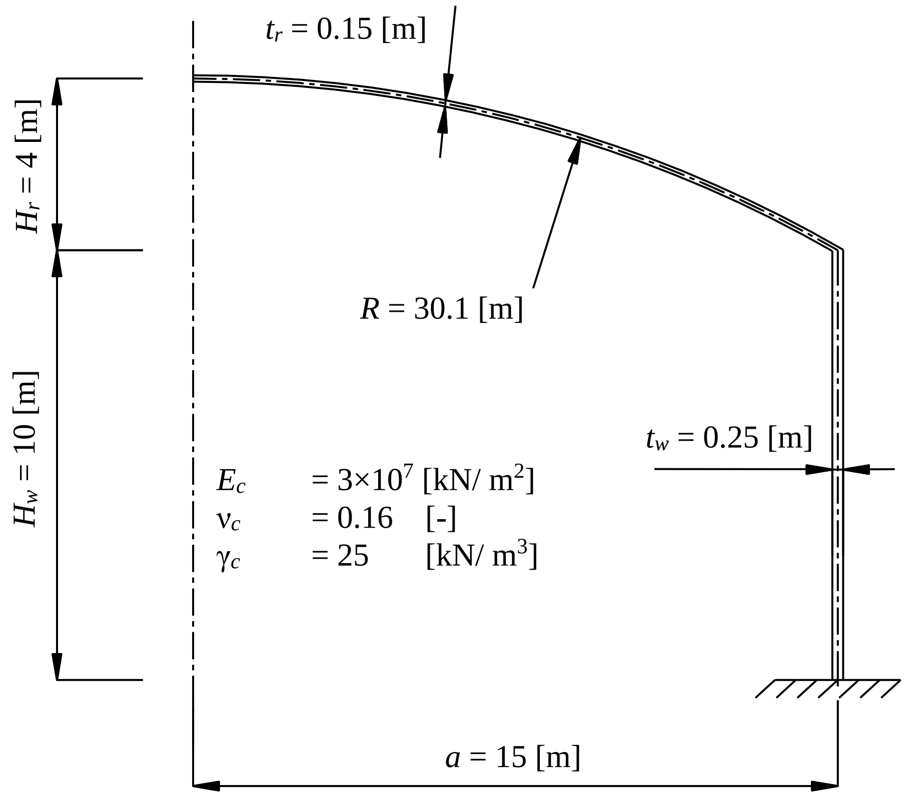


An example of a cylindrical shell with a variable wall thickness using the finite difference method is selected to illustrate some features of *ELPLA* for analyzing shell elements

Example 5 Analysis of a tank covered with a spherical dome

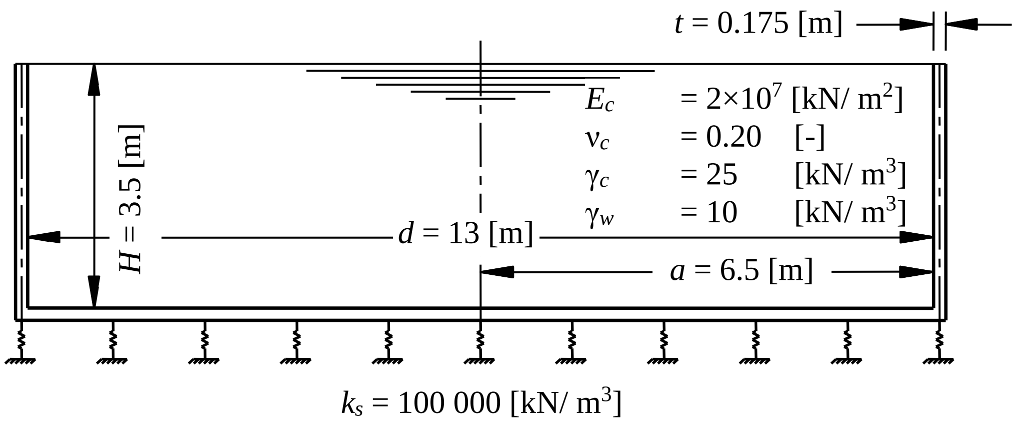
File	Tank covered with a spherical dome roof

Tank with covered roof



An example of an axi-symmetrically circular cylindrical tank covered with a spherical dome roof is selected illustrate some features of *ELPLA* for analyzing shell elements

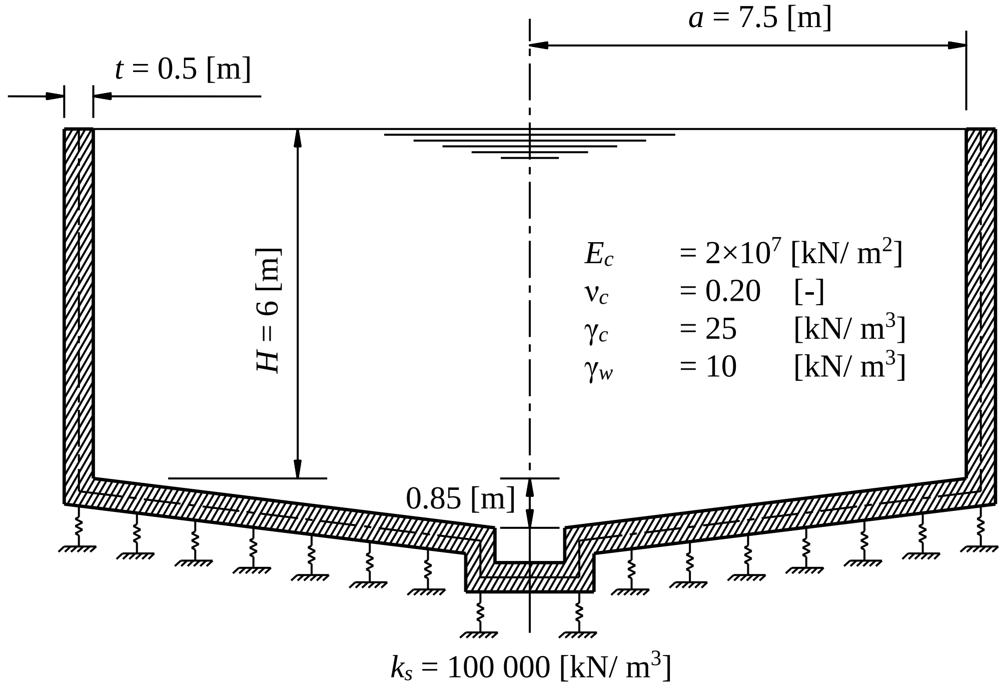
Example 6 Analysis of a tank resting on *Winkler's* medium

<table>
<tr><td>File</td><td>Circular cylindrical tank resting on isolated springs</td></tr>
<tr><td colspan="2">Tank on Winkler's medium


An example of an axi-symmetrically circular cylindrical tank resting on elastic foundation using Winkler's model is selected to illustrate some features of ELPLA for analyzing shell elements</td></tr>
</table>

Example 7 Analysis of a tank with a conical base resting on *Winkler's* medium

File	Circular cylindrical tank on isolated springs
Conical Tank	

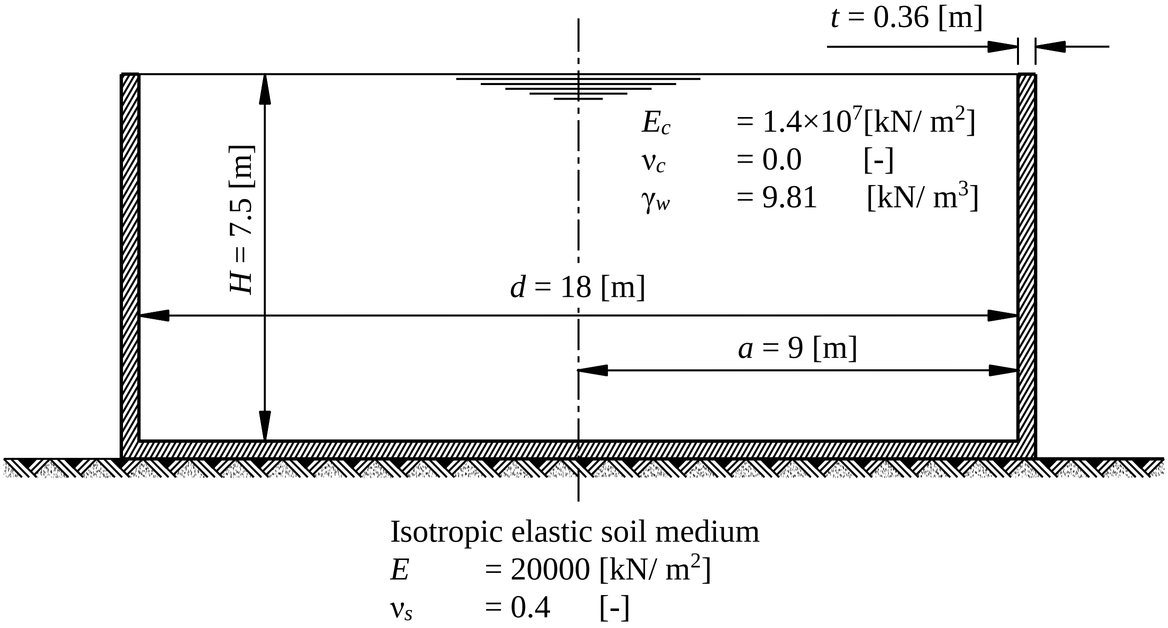


An example of a circular cylindrical tank with a conical base resting on elastic foundation using *Winkler's* model is selected to illustrate some features of *ELPLA* for analyzing circular cylindrical shell elements

Example 8 Analysis of a tank resting on half space soil medium

File	Circular cylindrical tank resting on an isotropic elastic soil medium

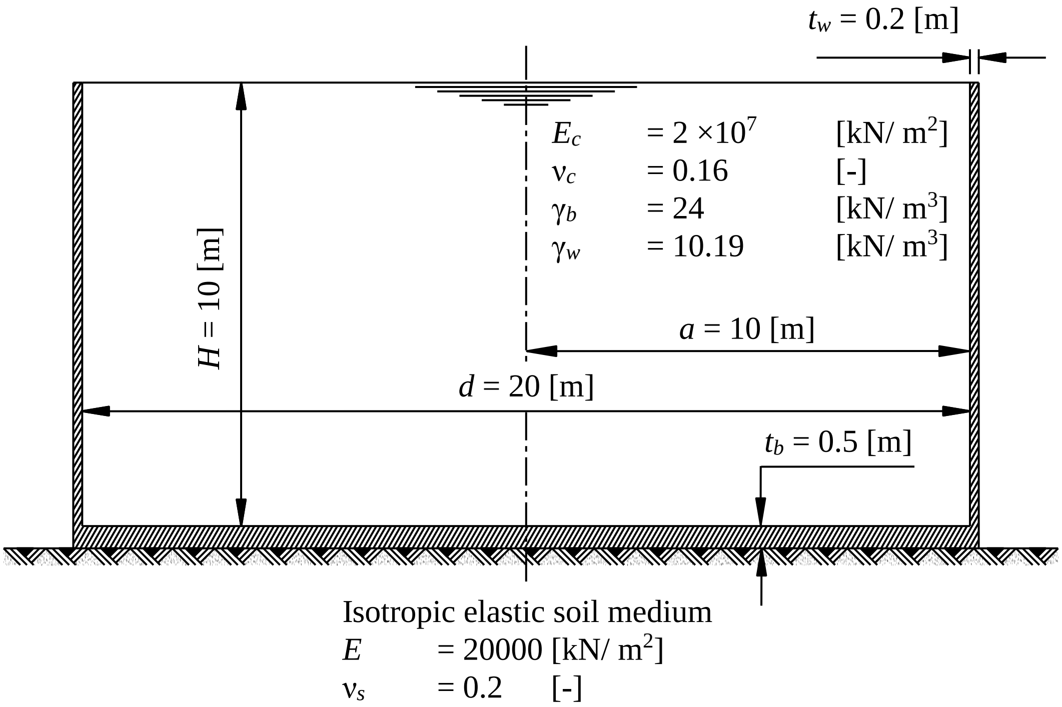
Tank resting on half space soil medium



An example of a circular cylindrical tank resting on an isotropic elastic half space soil medium is selected to illustrate some features of *ELPLA* for analyzing shell elements

Example 9 Analysis of a tank with a different base thickness resting on half space soil medium

File	Circular cylindrical tank resting on an isotropic elastic soil medium
Tank with different base thickness	

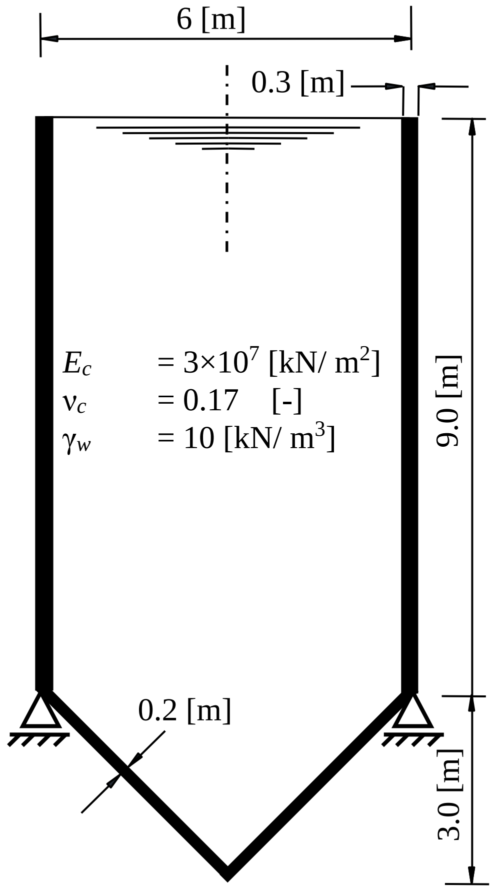


An example of a circular cylindrical storage tank resting on an isotropic elastic half space soil medium is selected to and illustrate some features of *ELPLA* for analyzing circular cylindrical shell elements

Example 10 Analysis of a cylindrical water container with a conical base

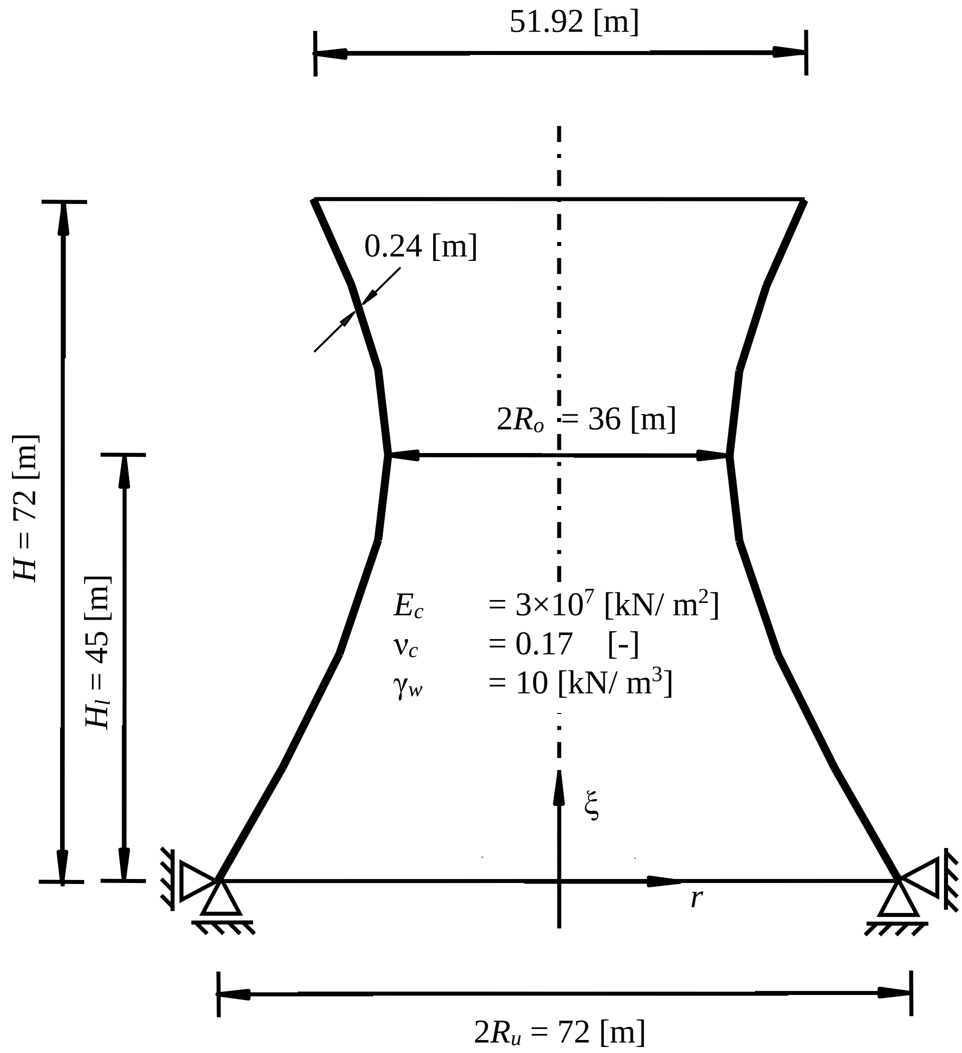
File	Cylindrical water container

Water container



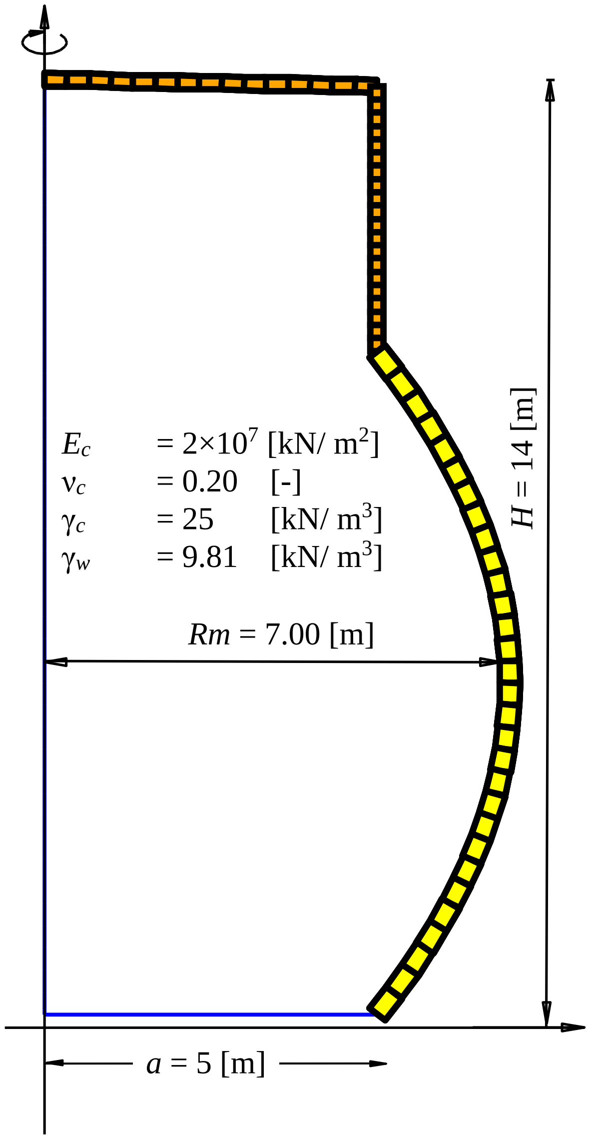
An example of a cylindrical water container with a conical base is selected to illustrate some features of *ELPLA* for analyzing circular cylindrical shell elements

Example 11 Analysis of a chimney with a hyperbolic shell wall

File	Geometry of the chimney
Hyperbolic chimney	
An example of an axi-symmetrically chimney with hyperbolic shell wall under a uniform external pressure is selected to illustrate some features of *ELPLA* for analyzing shell elements	

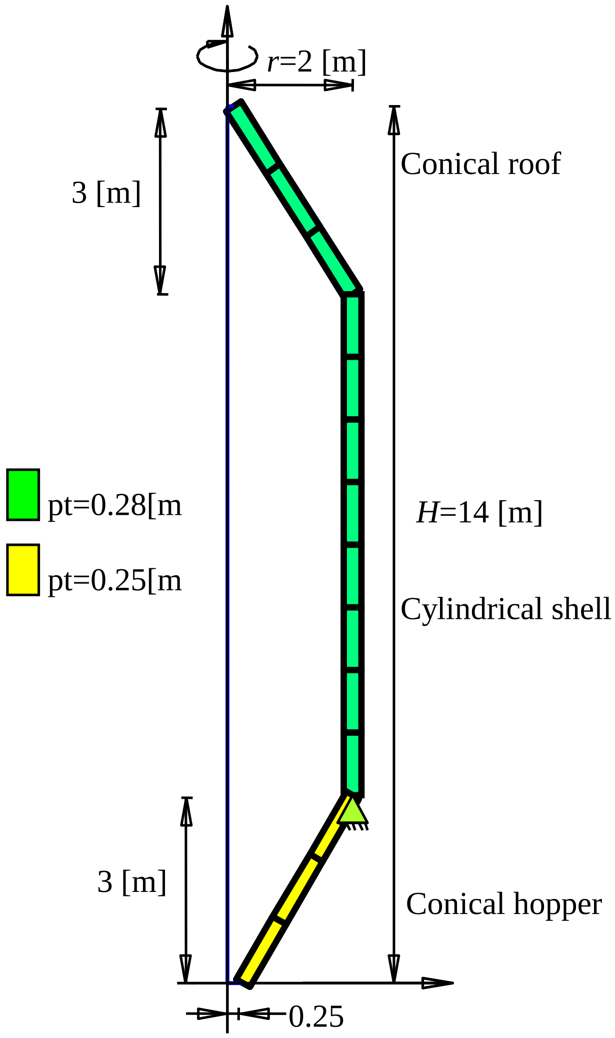
Example 12 Analysis of an irregular container resting on a layered soil

File	Irregular container
Container on layered soil	



An example of an axi-symmetrically container with irregular shape resting on a layered soil is selected to illustrate some features of *ELPLA* for analyzing shell elements

Example 13 Analysis of a circular silo for storing bulk materials

File	Geometry of the silo
Silo	
An example of a circular concrete silo is selected to illustrate some features of *ELPLA* for analyzing circular silos for storing bulk materials	

Example 1

Analysis of a circular loaded area resting on different soil layers

Contents Page

1 Description of the problem

An example of a loaded area resting on different soil layers is selected to illustrate some features of *ELPLA* for analyzing shell elements using circular and annular elements.

2 Geometry and properties

A circular loaded area of a radius a=5 [m] acting on three different soil layers as shown in Figure 1.1.

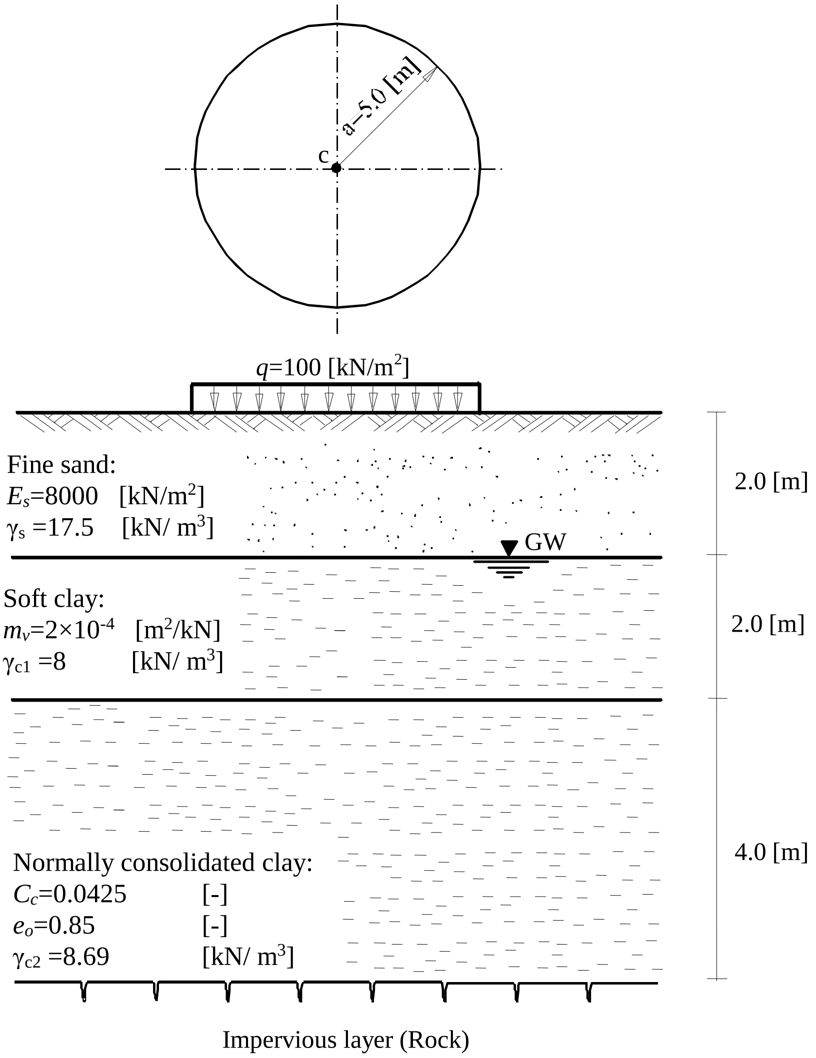


Figure 1.1 Soil profile under the circular loaded area

3 Creating the project

In this section, the user will learn how to create a project for analyzing a circular loaded area resting on different soil layers. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

3.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 1.2. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of Forms. The first Form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 1.2).

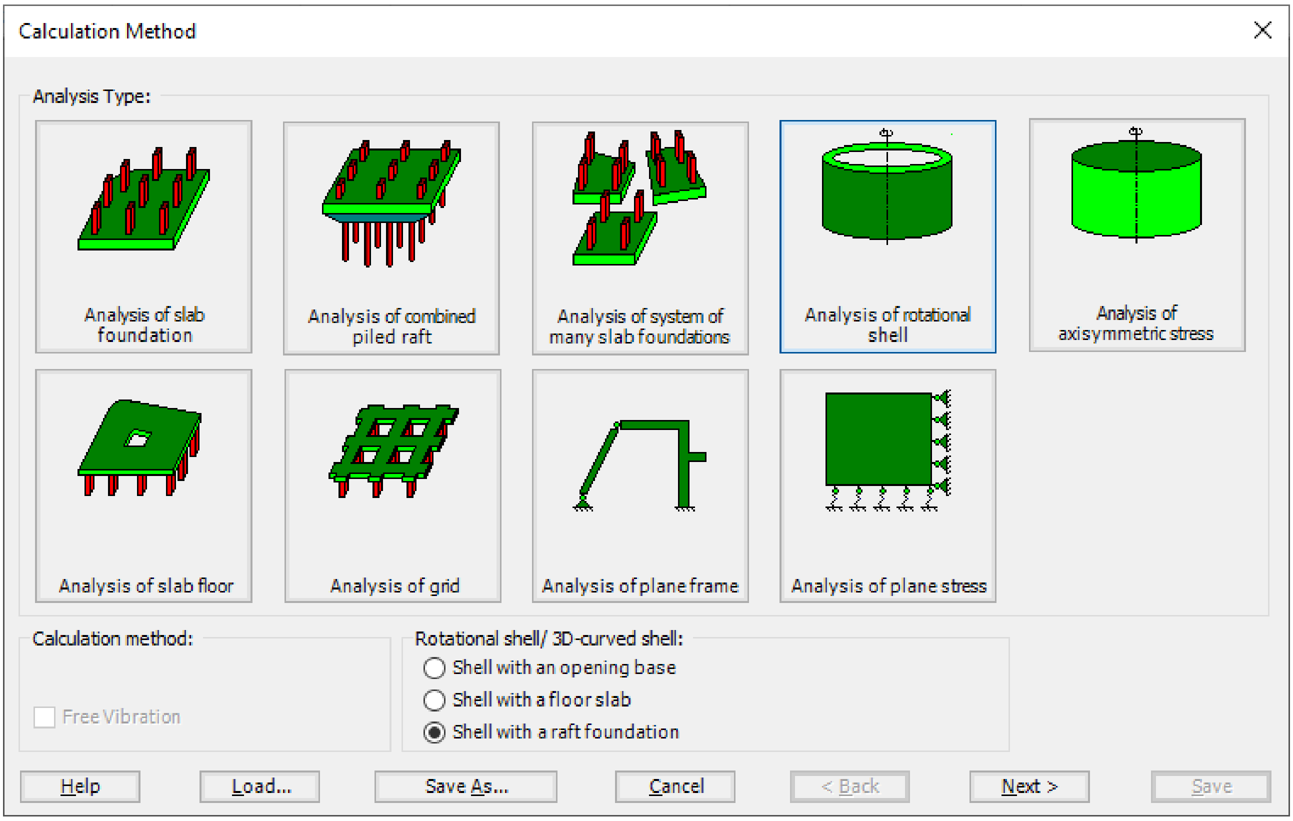


Figure 1.2 "Analysis Type" Form

As the analysis type is a circular loaded area problem, select "Analysis of rotational shell" button, and check "Shell with a raft foundation" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Calculation Methods" Form appears, Figure 1.3.

To define the calculation method:

- Select the calculation method "9-Flexible Foundation"
- Select the subsoil model as "Layered soil model"
- Click "Next" button to go to the next Form

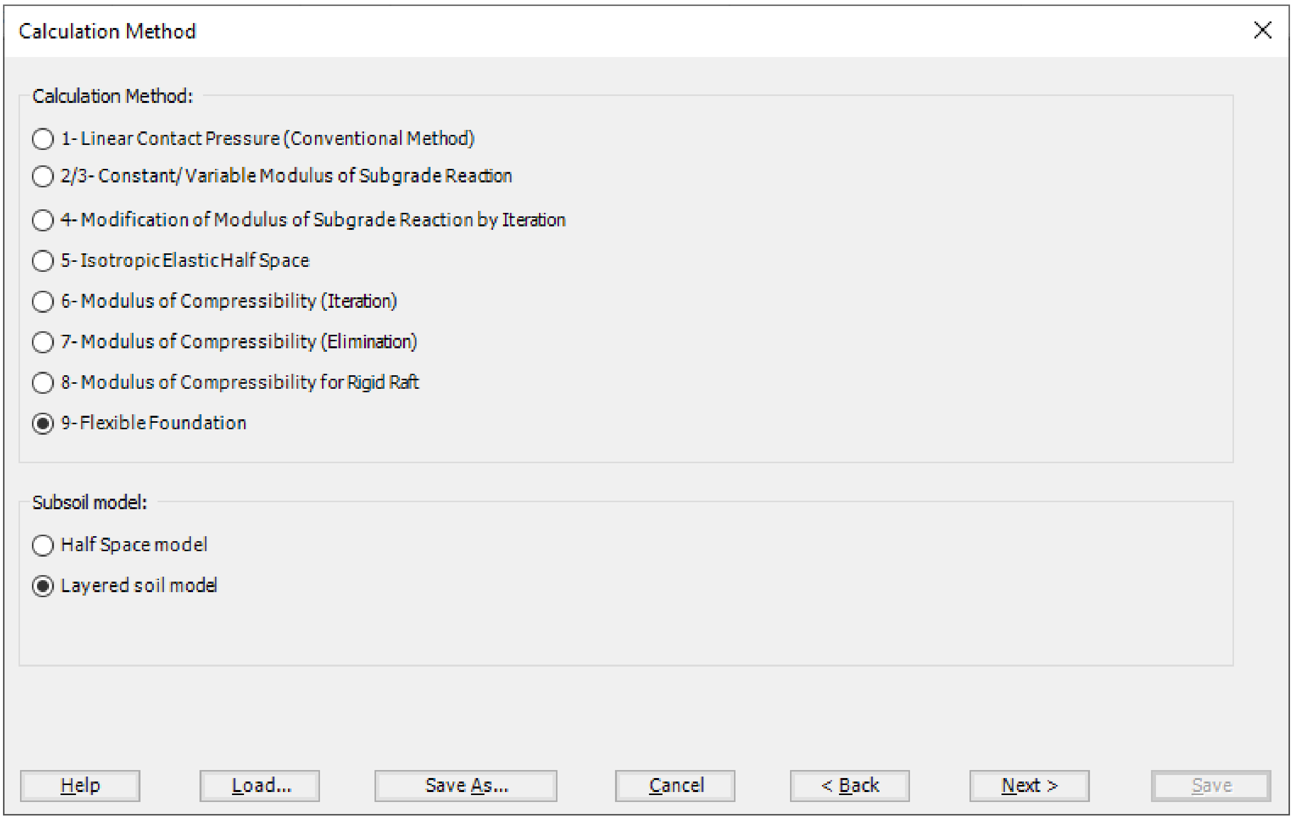


Figure 1.3 "Calculation Methods" Form

The last Form in the wizard is the "Options" Form, Figure 1.4. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Since no option will be considered in the analysis, click the "Save" button.

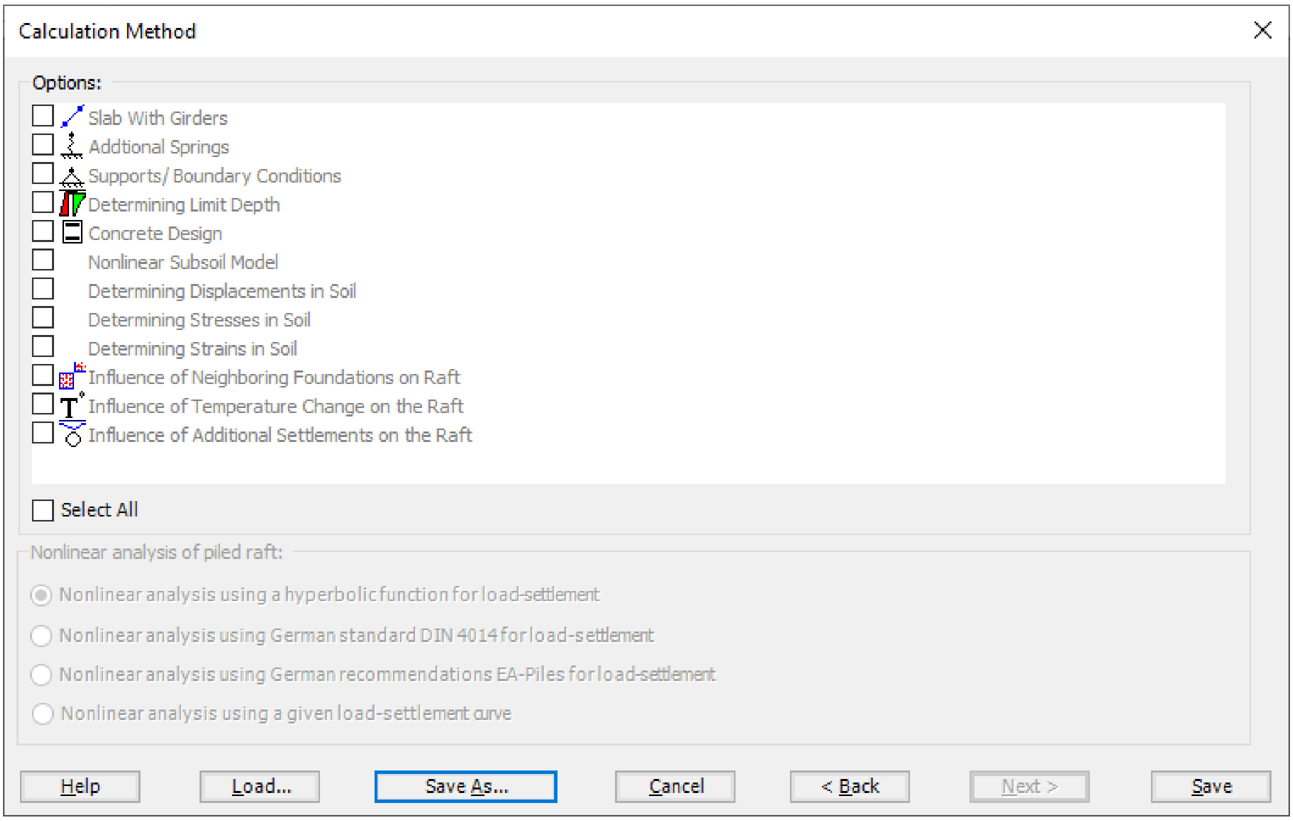


Figure 1.4 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 1.5. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Circular loaded area ". *ELPLA* will use automatically this file name in all reading and writing processes.

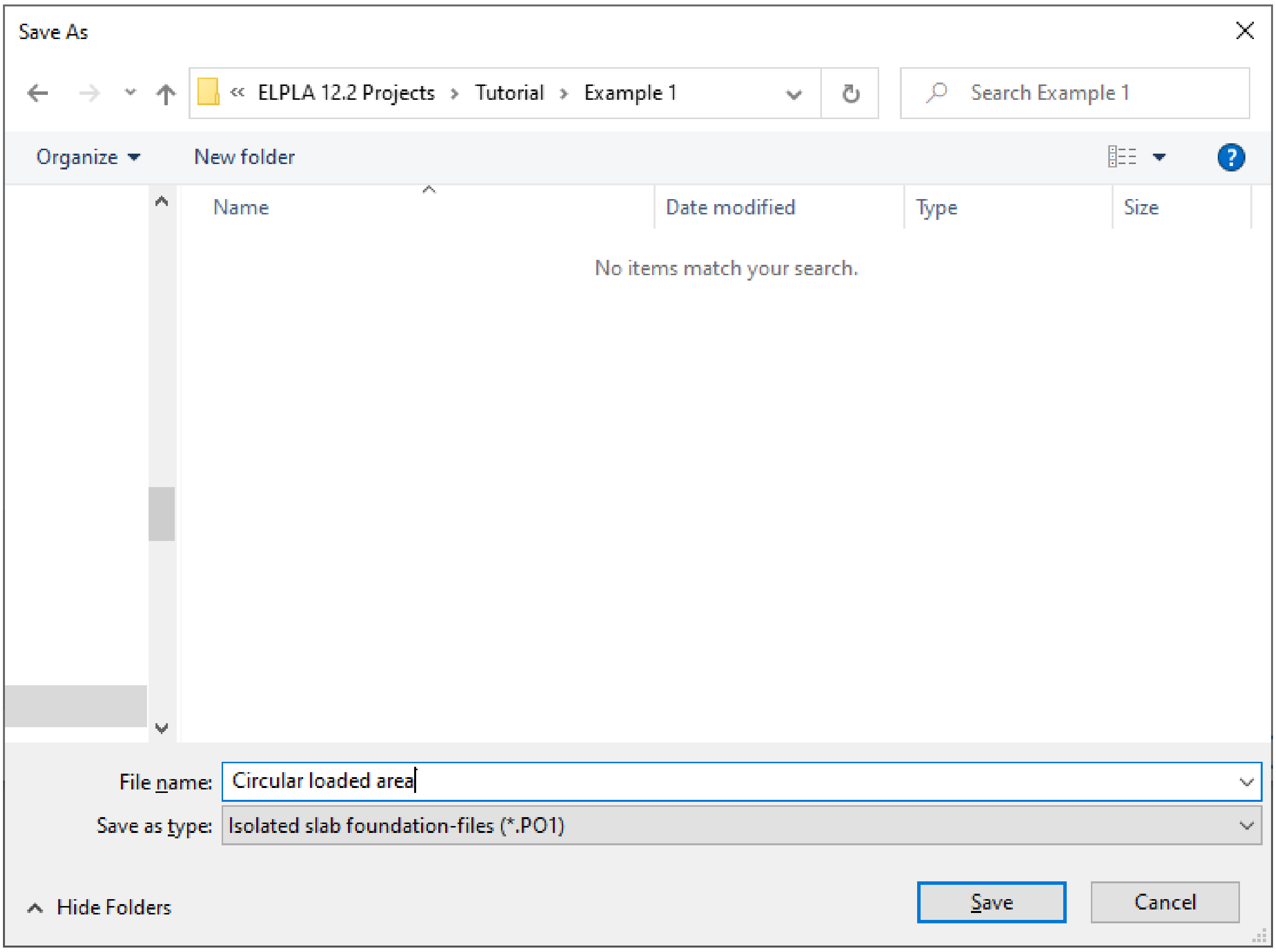


Figure 1.5 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Circular loaded area] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

3.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 1.6 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a circular loaded area resting on different soil layers"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

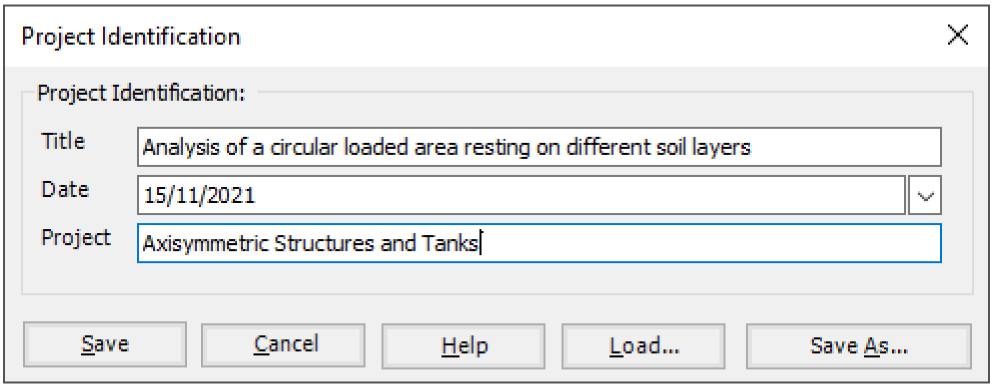


Figure 1.6 "Project Identification" dialog box

3.3 FE-Net data

For the given problem, the shell has a circular shape with a radius of a=5 [m] and is divided into 10 segments. *ELPLA* has different procedures for defining the FE-Net.

To define the FE-Net for this shell, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 1.7. This wizard will guide you through the steps required to generate a FE-Net. The first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of shells. These net templates are used to generate standard nets.

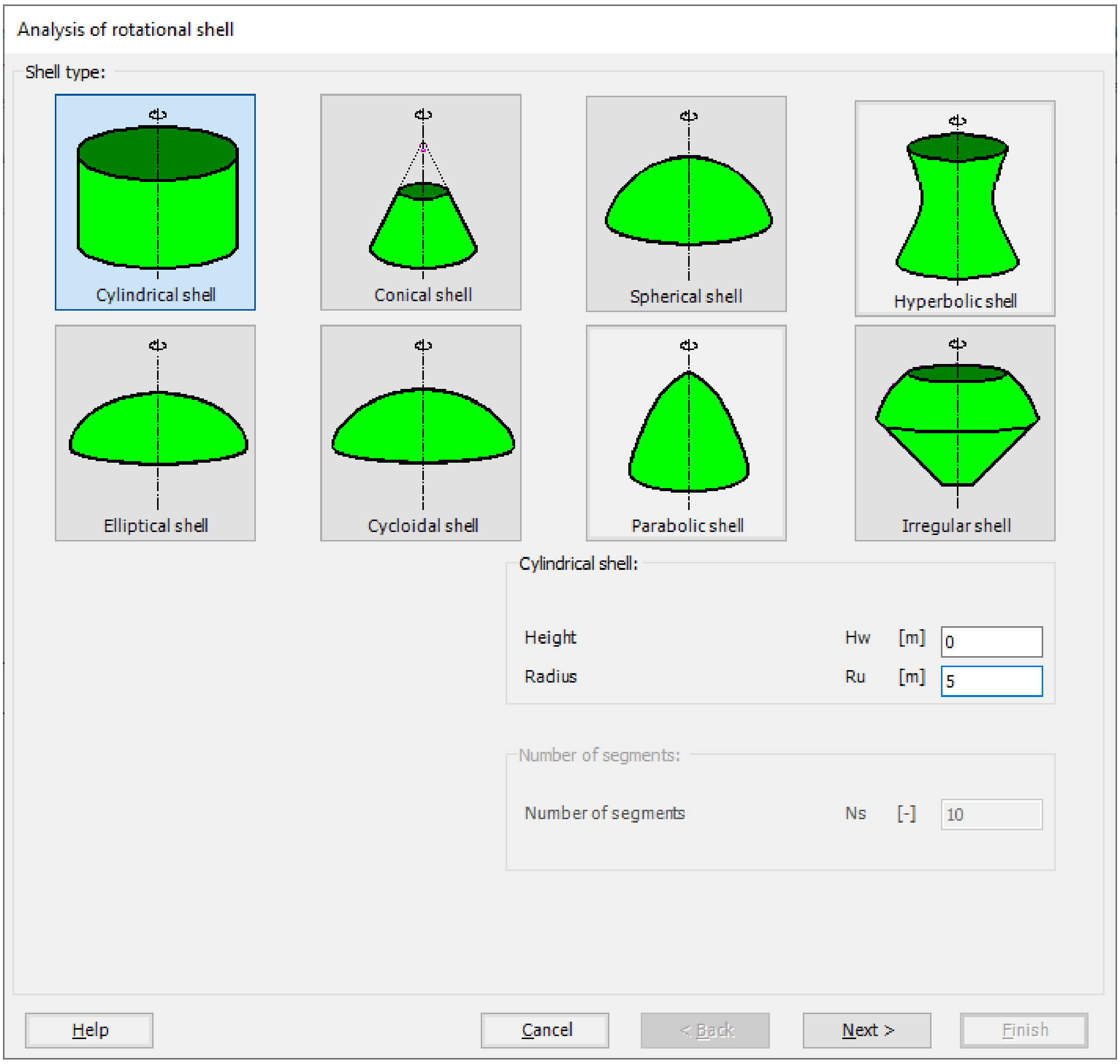


Figure 1.7 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Cylindrical shell" button
- Type 0 in the "Height" edit box, as the example is a circular plate
- Type 5 in the "Radius" edit box
- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Net of Base" Form appears Figure 1.8.

To edit the grid spacing in *x*-direction, do the following steps in "Grid in *x*-direction" frame:

- Choose "Constant grid interval" check box
- Type 10 in the "No. of grid intervals" edit box

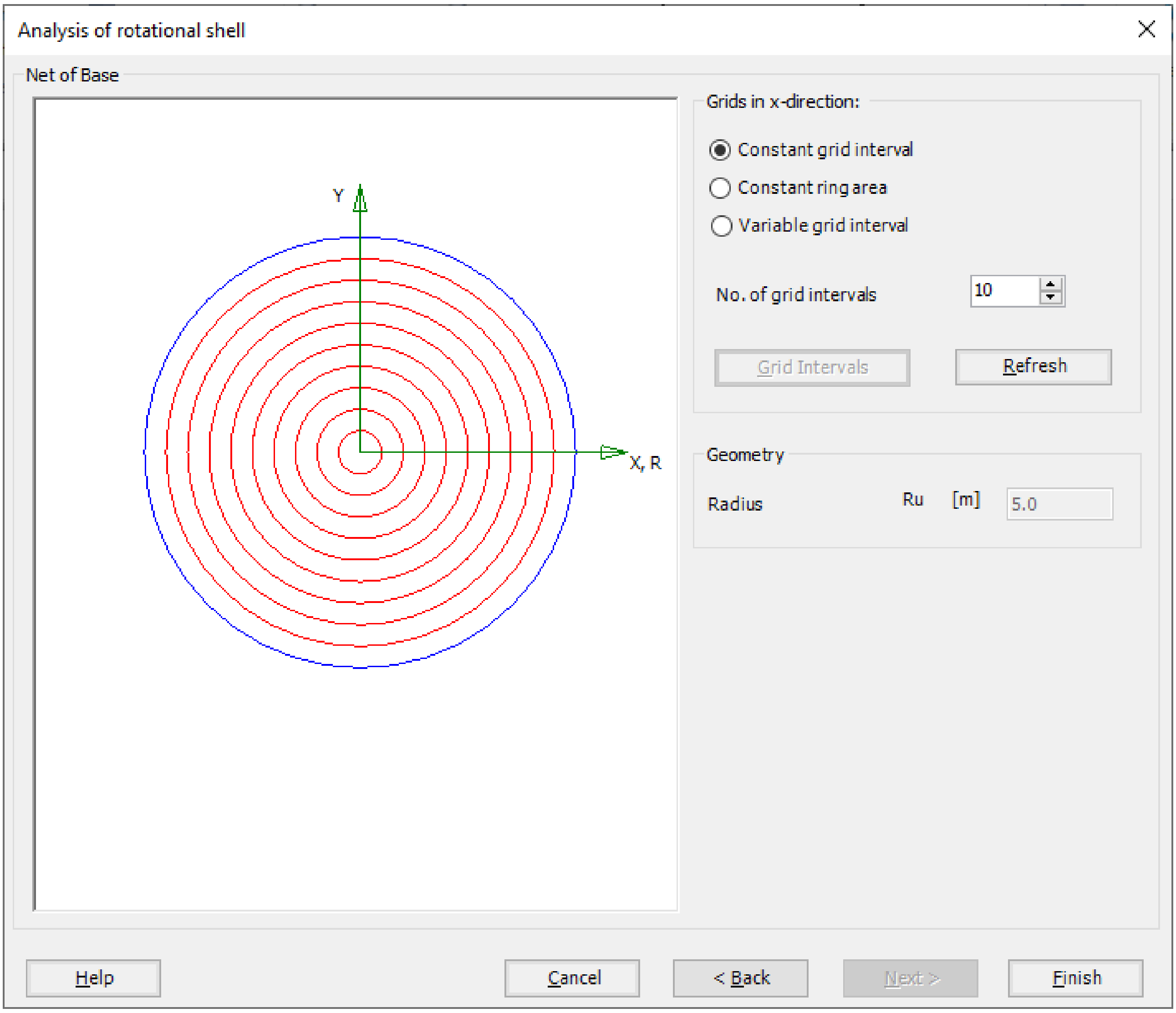


Figure 1.8 "Net of Base" Form

ELPLA will generate a sector from the circular area with 10 circular elements. The following window in Figure 1.9 appears with the generated net.

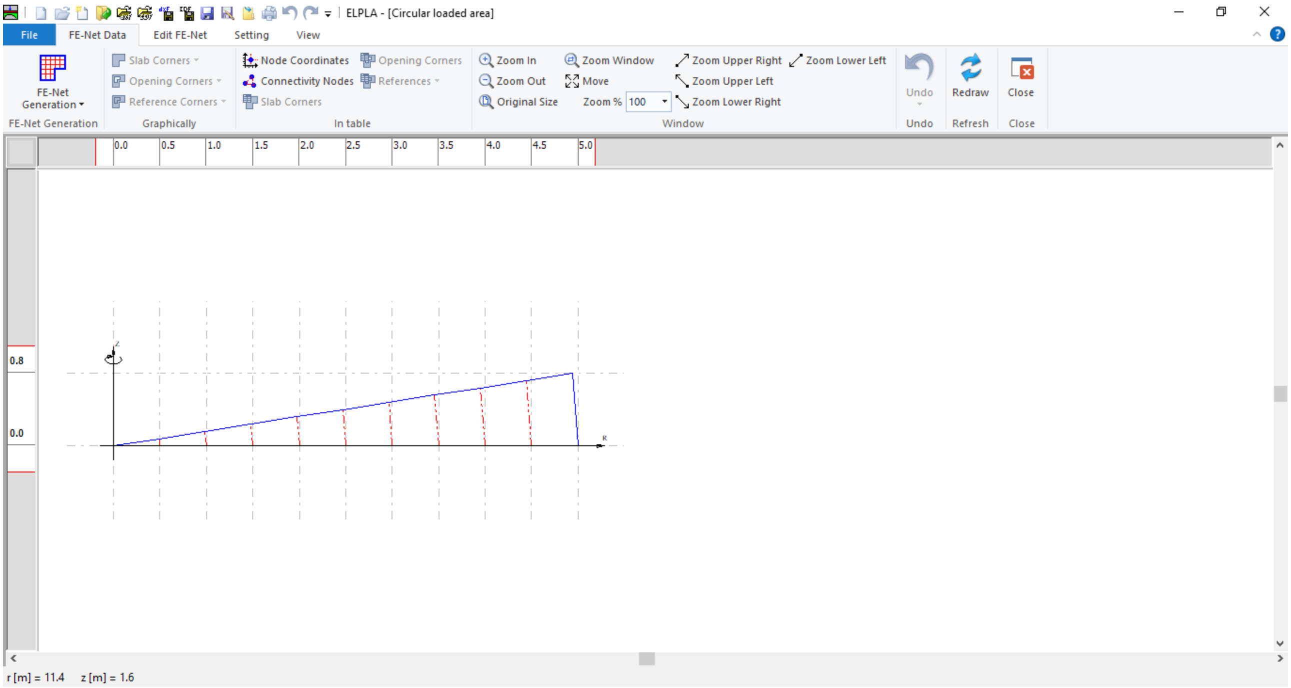


Figure 1.9 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 1.9 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 1.9 to close the "FE-Net" window and return to *ELPLA* main window

3.4 Soil Properties

In *ELPLA*, there are three different soil models with several calculation methods. Therefore, the soil properties for each method are required to be defined according to the used soil model. In the current example, the soil model, which is used in the analysis, is a "Layered Soil Model". This model requires that the subsoil have to be defined by boring logs. In the example, the boring log has multi-layers with different soil materials. The geotechnical data for each layer is unit weight of the soil γ_s and modulus of Elasticity for loading E_s and reloading W_s or Compression index *Cc* and recompression index *Cr*.

To define the soil properties, choose "Soil Properties" command from “Data” Tab. The following "Soil Properties" Window in Figure 1.10 appears with a default-boring log.

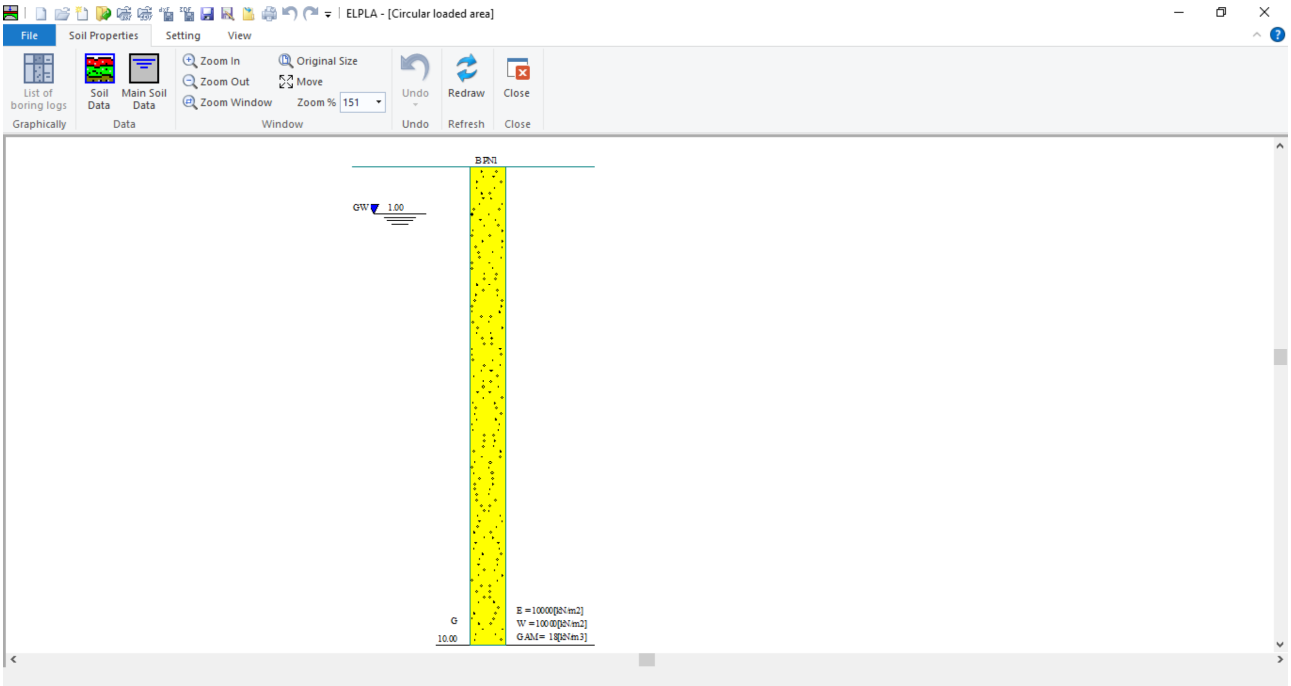


Figure 1.10 "Soil Properties" window with a default-boring log

In Figure 1.10, soil properties are defined through the "Data" menu, which contains the following two commands:

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

To enter the soil properties for the boring log of the current example

- Choose "Soil Data" command from "Data" menu in the window of Figure 1.10.
  The following dialog box in Figure 1.11 with default-boring log data appears

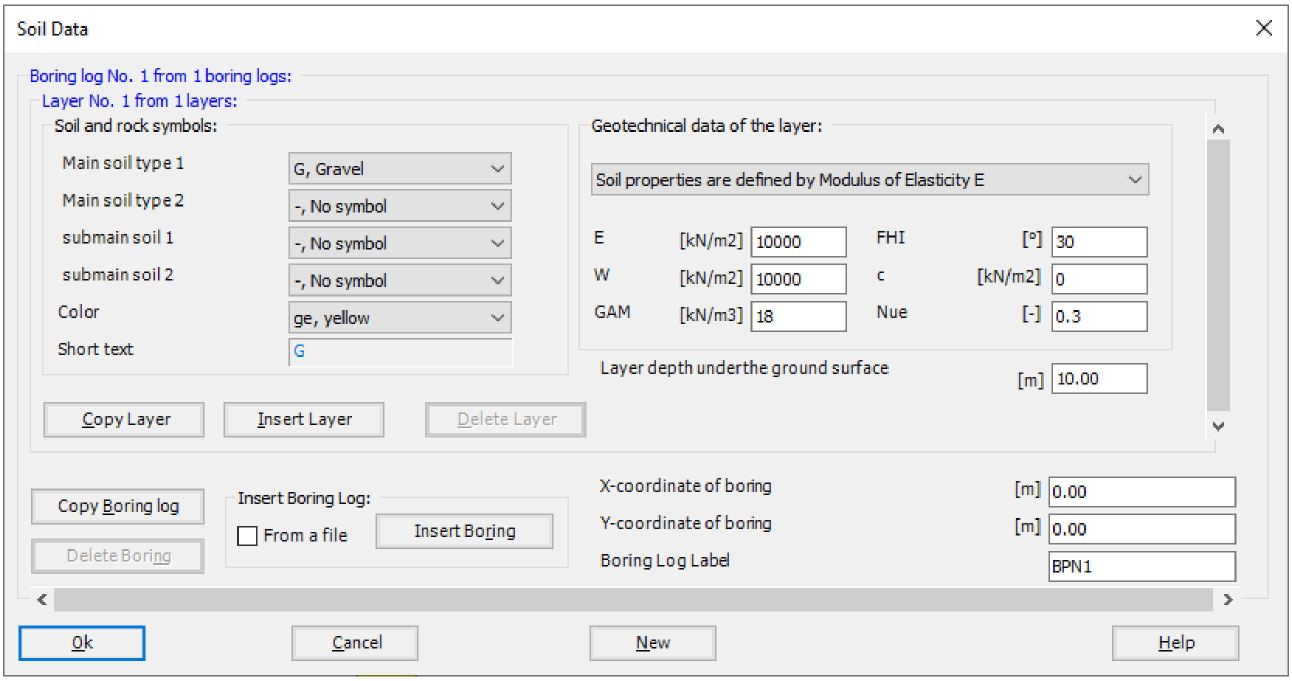


Figure 1.11 "Soil data" dialog box with default-boring log data

In the "Geotechnical data of the layer" dialog group box in Figure 1.11, Soil properties are defined by Modulus of compressibility, define the geotechnical data of the first soil layer of the boring log as follows:

E_s	= 8 000	[kN/m^2]
W_s	= 8 000	[kN/m^2]
Gam	= 18	[kN/m^3]
C	= 10	[kN/m^2]
φ	= 0	[°]

Due to the presence of the ground water, the soil above the ground water level has a differential unit weight from the soil under that level. Therefore, the layer depth of the first layer is taken to be 2 [m], which is equal to the ground water level. Now, type this value in "Layer depth under the ground surface" edit box.

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

To define the soil type and color for the first layer, select "fS, Fine sand" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box in Figure 1.11. The color of the fine sand layer according to the German Standard DIN 4023 will be automatically created. The user can change this color. In addition, a short text "fS" will be automatically created for the fine sand.

To enter the second layer of the boring log

- Click "Copy Layer" button in Figure 1.11. A layer that has the same properties of the first layer will be copied
- Use the vertical scrollbar to move to the second soil layer. Layer No. will be typed automatically at the upper-left corner of the main dialog box of soil layers as a head title
- Change the value of the unit weight of the soil for the second soil layer from 18 [kN/m^3] to 8 [kN/m^3]
- Change the value "*ES*" and "*Ws*" from 8000 [kN/m^2] to 5000 [kN/m^2]
- Change the value of the layer depth under the ground surface from 2 [m] to 4 [m]
- From "Soil and rock symbols" dialog group box, Change "fS, Fine sand" into "T, Clay" as the soil type of the second layer is clay

To enter the third layer

- Click "Insert Layer" button in Figure 1.11, then a layer will be inserted
- Use the vertical scrollbar to move to the third soil layer
- In "Geotechnical data of the layer" dialog group box in Figure 1.11, Soil properties are defined by Compression Index *Cc*, define the geotechnical data of the clay layer as follows:

Cc	= 0.04		Phi	= 30	[°]
Cr	= 0.04		*c*	= 10	[kN/m^2]
Gam	= 8	[kN/m^3]	*eo*	= 0.85	

- Select "T, Clay" as the soil type in "Main soil type 1" combo box in "Soil and rock symbols" dialog group box
- Type 8 in the "Layer depth under the ground surface" edit box

Note that the unit weight of the soil is used to determine the uniform load q [kN/m^2] on the circular area, which is equal to $\gamma_s \times d_f$.

Now all data and parameters for the boring log (Figure 1.12) have been entered.

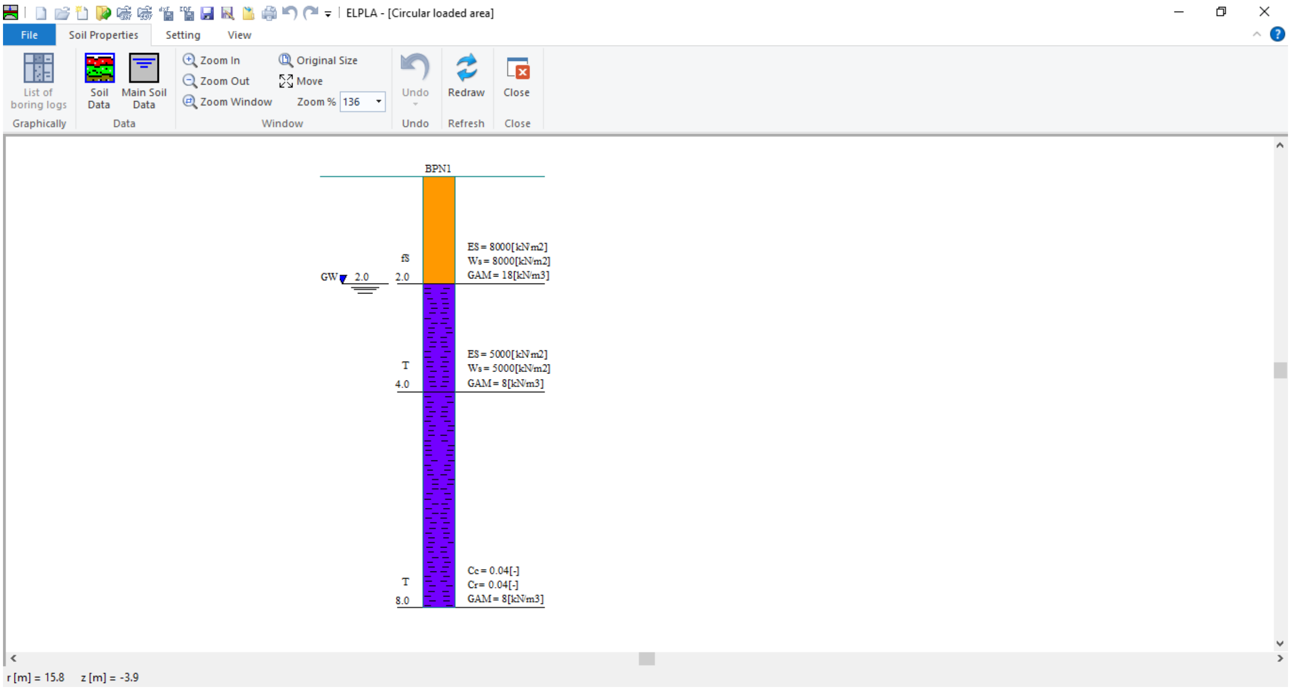


Figure 1.12 boring log

To enter the main soil data for all layers, choose "Main Soil Data" command from "Data" menu in Figure 1.10. The following dialog box in Figure 1.13 appears with default main soil data. The main soil data for the current example, which are required to be defined, are the settlement reduction factor α [-] and the groundwater depth under the ground surface G_w [m]. Any other data corresponding to main soil data are not required for this example. Therefore, the user can use the default values.

In the dialog box of Figure 1.13, enter the settlement reduction factor α [-] and the groundwater depth under the ground surface G_w [m]. Then click "OK" button.

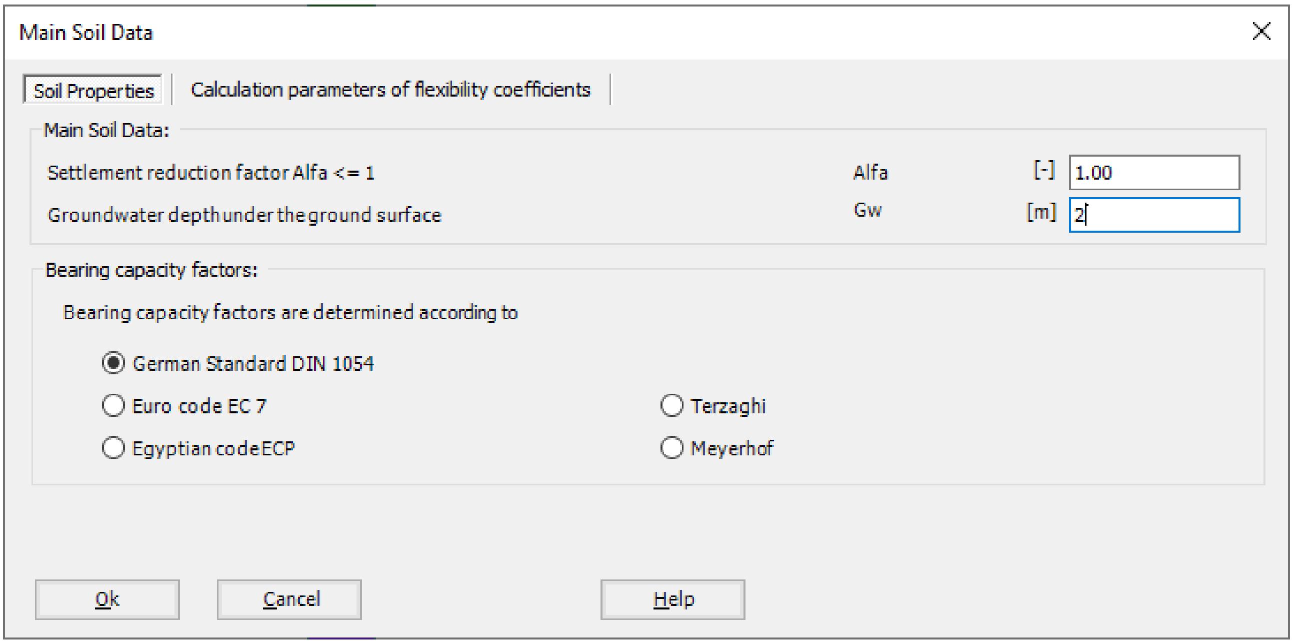


Figure 1.13 "Main Soil Data" dialog box

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

- Choose "Save" command from "File" menu in Figure 1.10 to save the data of boring logs
- Choose "Close" command from "File" menu in Figure 1.10 to close "Soil properties" window and return to *ELPLA* main window

3.5 Shell properties

To define the shell properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 1.14 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, and the unit weight of the shell. Any other data corresponding to shell properties in the program menus are not required for this example.

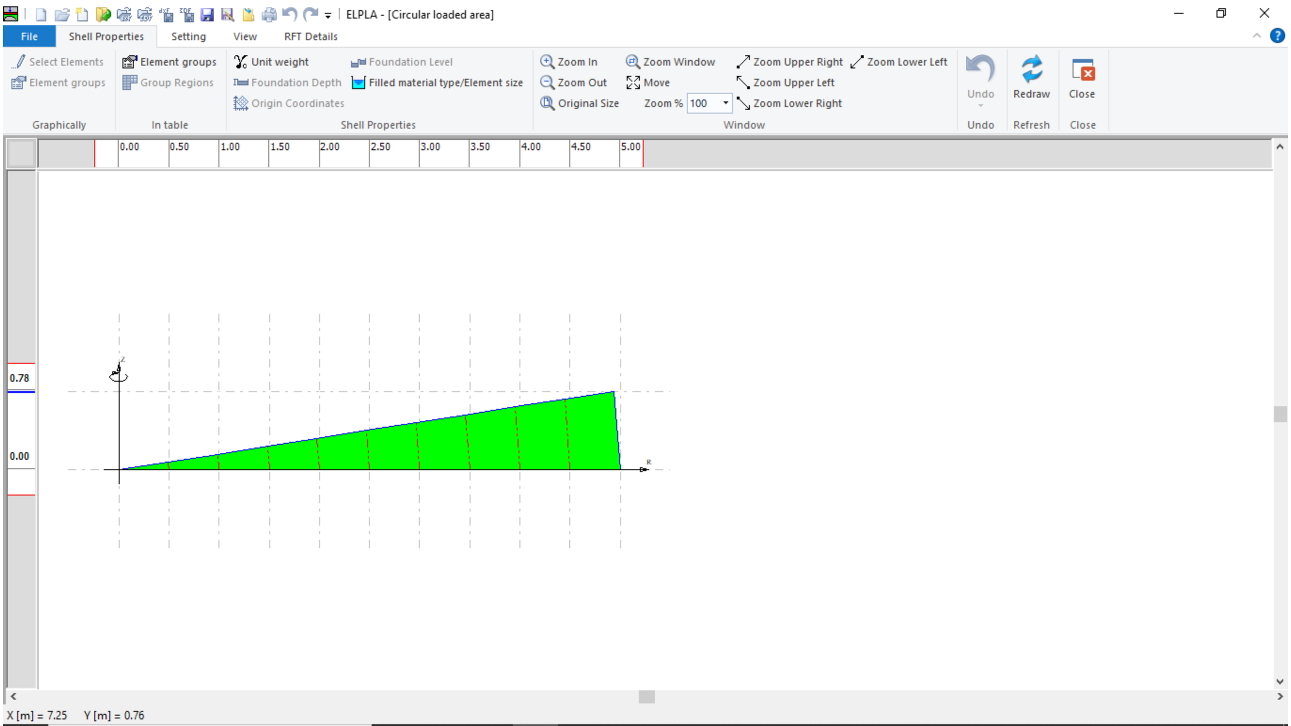


Figure 1.14 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 1.15 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness. Then click "OK" button.

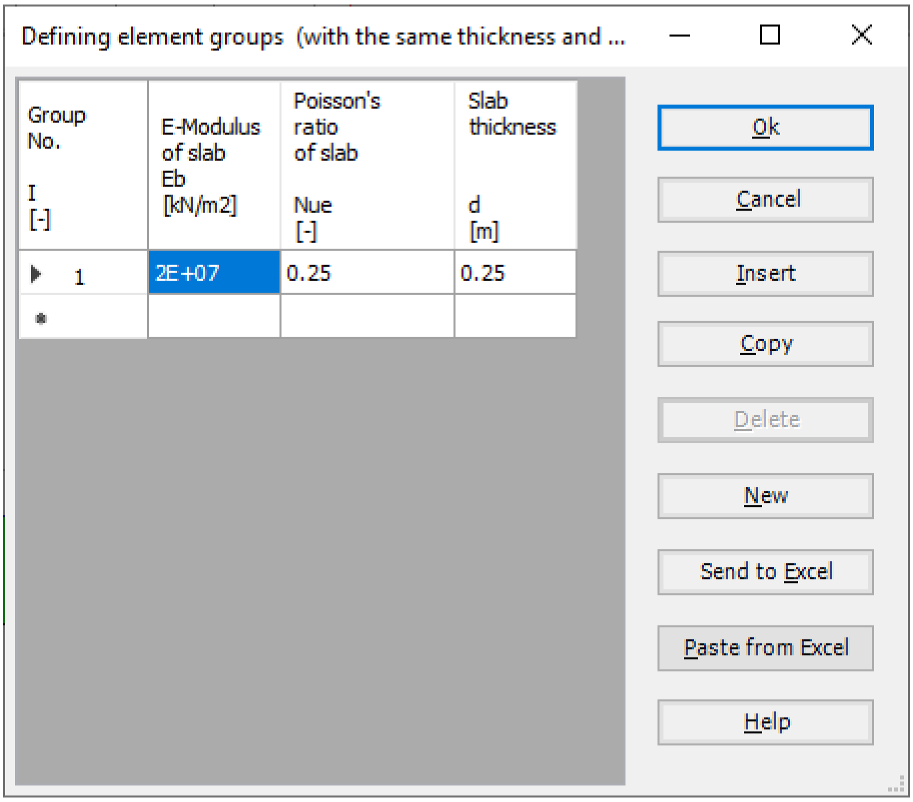


Figure 1.15 "Defining element groups" list box

To enter the unit weight of the shell, choose "Unit weight" command from "Shell Properties" menu in the window of Figure 1.14. The following dialog box in Figure 1.16 with a default unit weight of 25 [kN/m^3] appears. Type 400 in the "Unit weight" edit box, to define the uniform load on the circular area. Then click "OK" button.

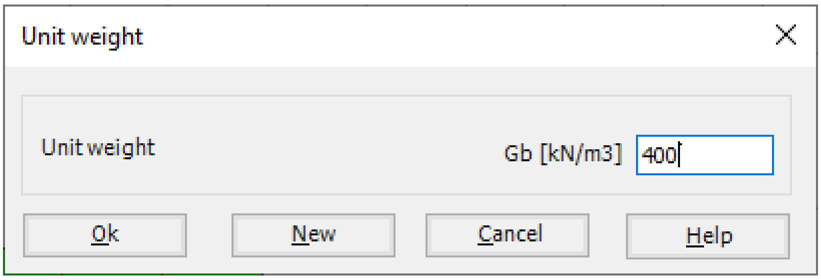


Figure 1.16 "Unit weight" dialog box

After entering the Shell Properties, do the following two steps:

- Choose "Save" command from "File" menu in Figure 1.14 to save the shell properties
- Choose "Close" command from "File" menu in Figure 1.14 to close the "Shell Properties" window and return to *ELPLA* main window

3.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 1.17 appears. In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 1.17. In this example, there is not applied load, as the load has been already defined by the unit weight of the plate.

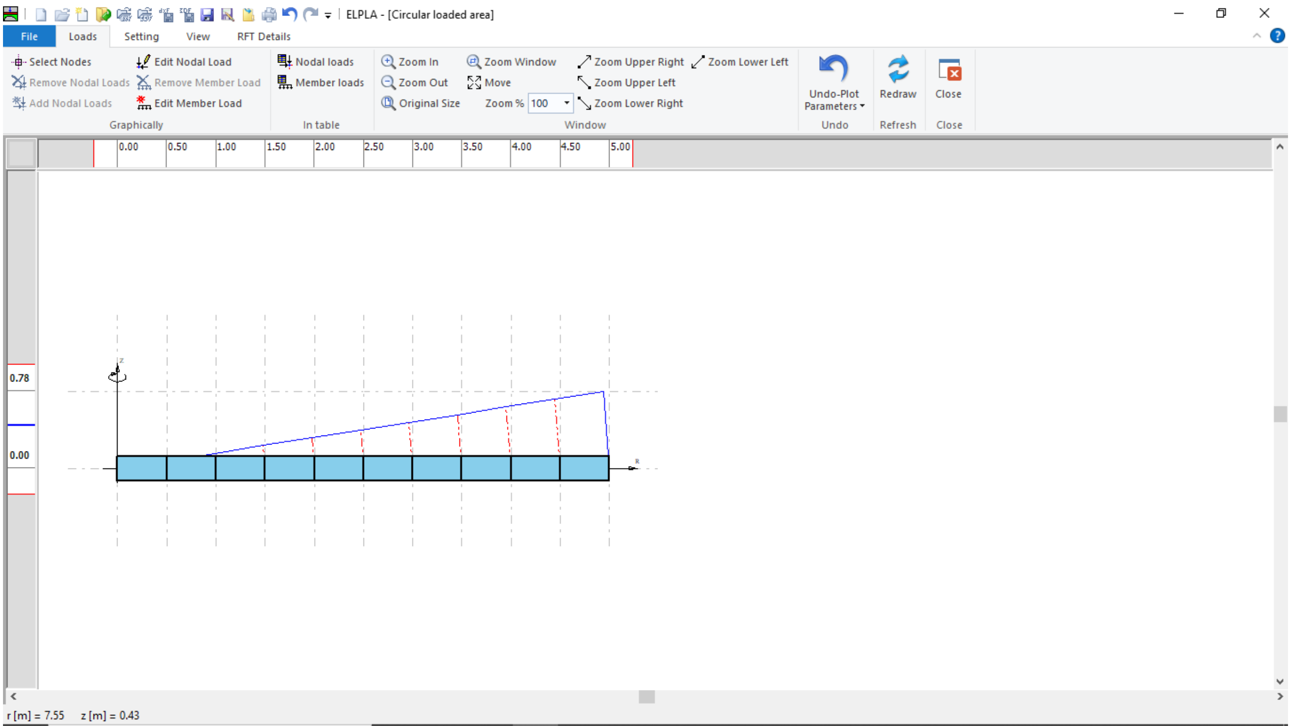


Figure 1.17 "Loads" Window

Do the following two steps:

- Choose "Save" command from "File" menu in Figure 1.17 to save the load data
- Choose "Close" command from "File" menu in Figure 1.17 to close the "Loads" window and return to *ELPLA* main window

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

4 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 1.18.

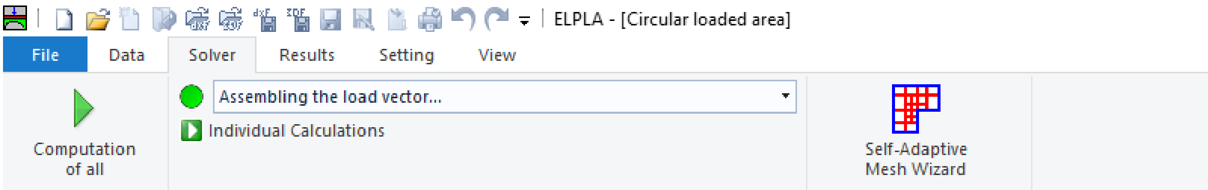


Figure 1.18 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Analysis of the flexible foundation
- Determining deformation, internal forces, contact pressures

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

Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab window.
- The progress of all computations according to the defined method will be carried out automatically with displaying Information through menus and messages.

Analysis progress

Analysis progress menu in Figure 1.19 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

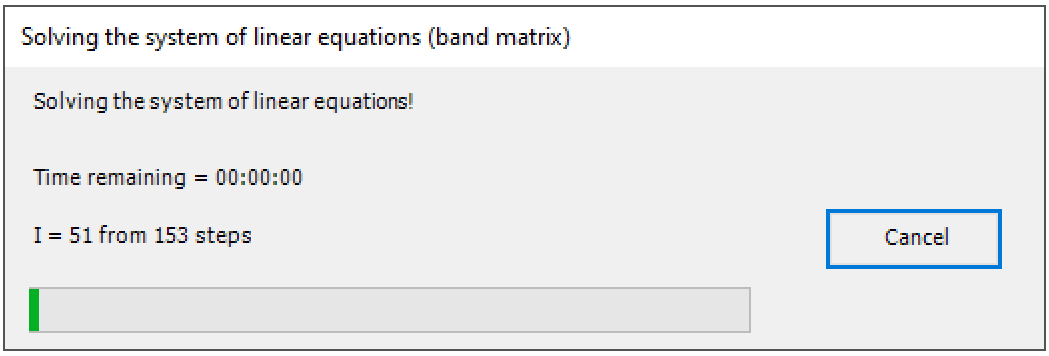


Figure 1.19 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 1.20. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

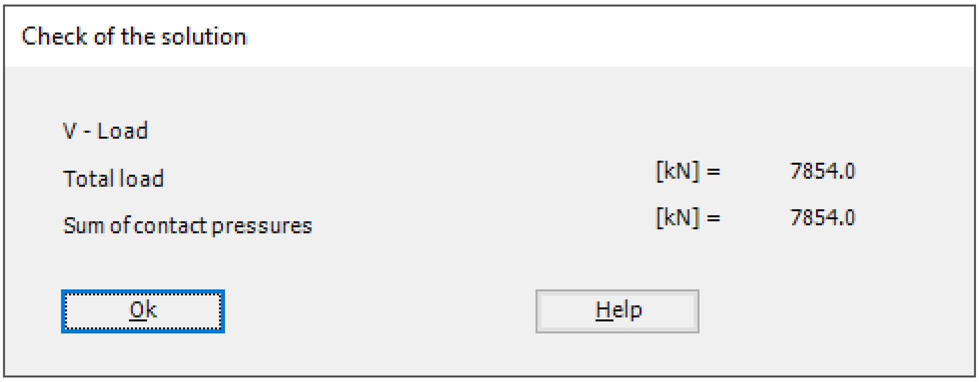


Figure 1.20 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

5 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab.

5.1 Viewing data and result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 1.21).

Figure 1.21 "Results" Tab

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

- Data in the plan
- Boring logs and limit depth
- Sections in the shell base

To view sections in shell base

- Choose "Sections in shell base" command from "Section" menu. The following option box in Figure 1.22 appears
- In the "Sections in shell base" option box, select "Base settlements w" as an example for the results to be displayed
- Click "OK" button

The Settlements are now displayed as shown in Figure 1.23.

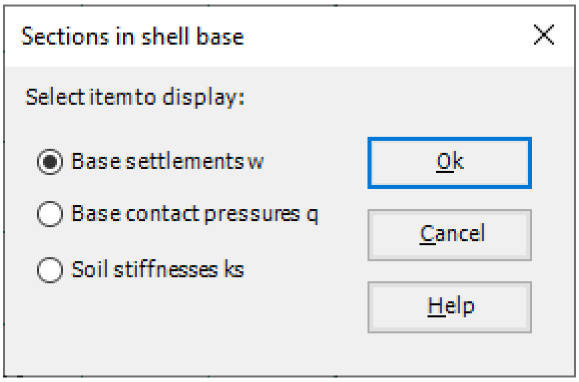


Figure 1.22 "Sections in shell base" option box

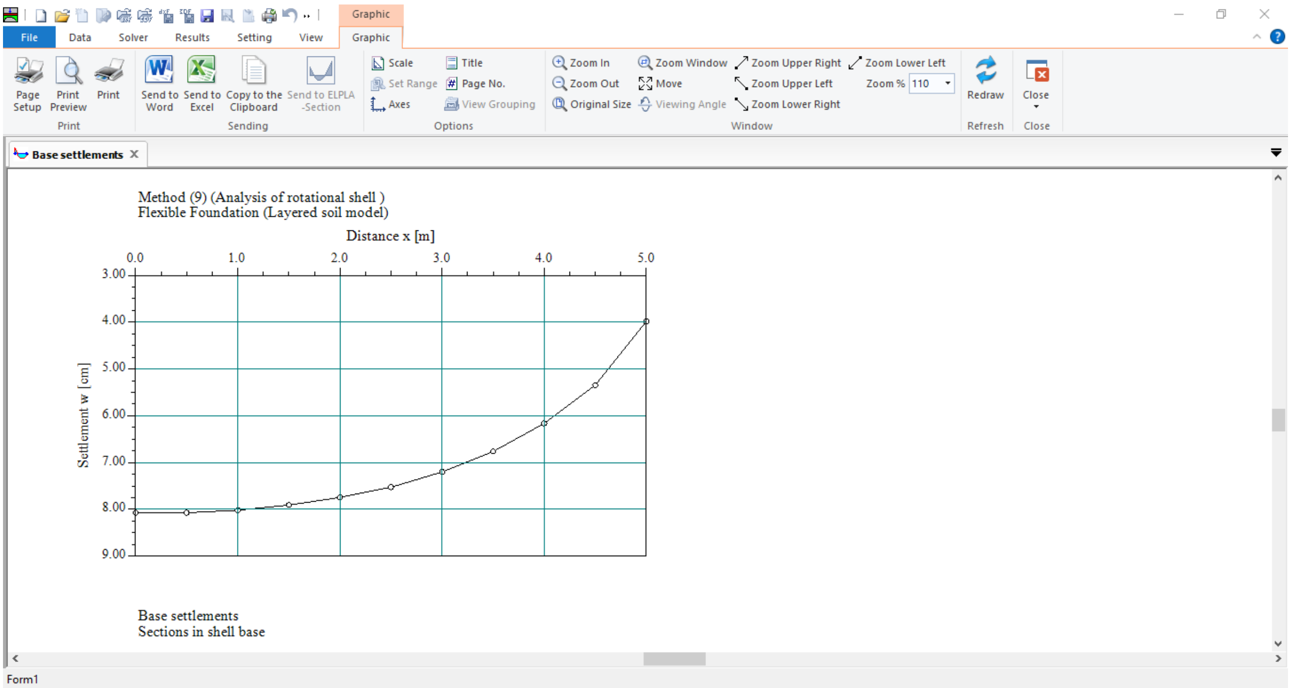


Figure 1.23 Base settlements Sections in shell base

5.2 Listing data and results in tables

The "Result" Tab contains the commands of listing data and results. The commands for listing data and results in tables are:

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

To list results in a table

- Choose "Display tables of Results" command from "List" menu. The following option box in Figure 1.24 appears
- In the "Display Tables of Results" option box, select "Settlements/Contact pressures" as an example for the result to be listed in a table
- Click "OK" button. The loading results are now listed (Figure 1.25)
- Choose "Send to Excel" from "Sending" menu if you wish to export the table to a MS Excel application, Figure 1.26

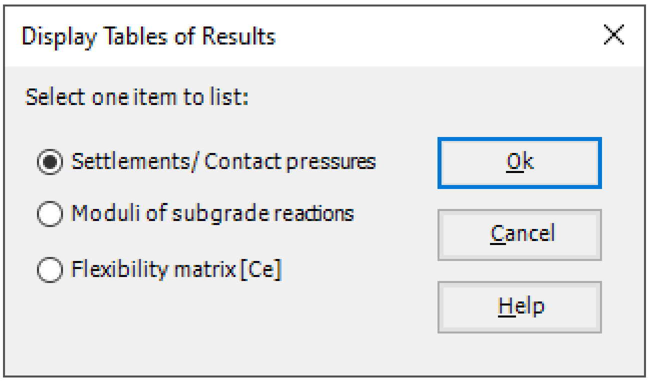
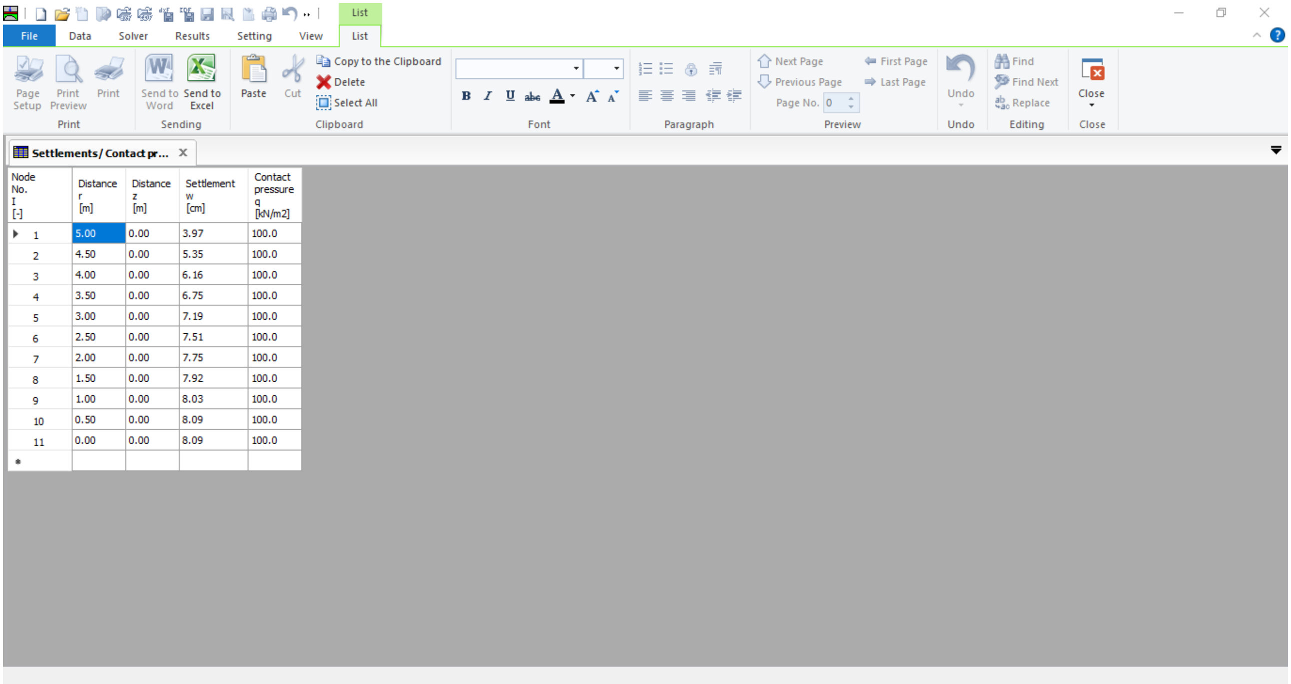


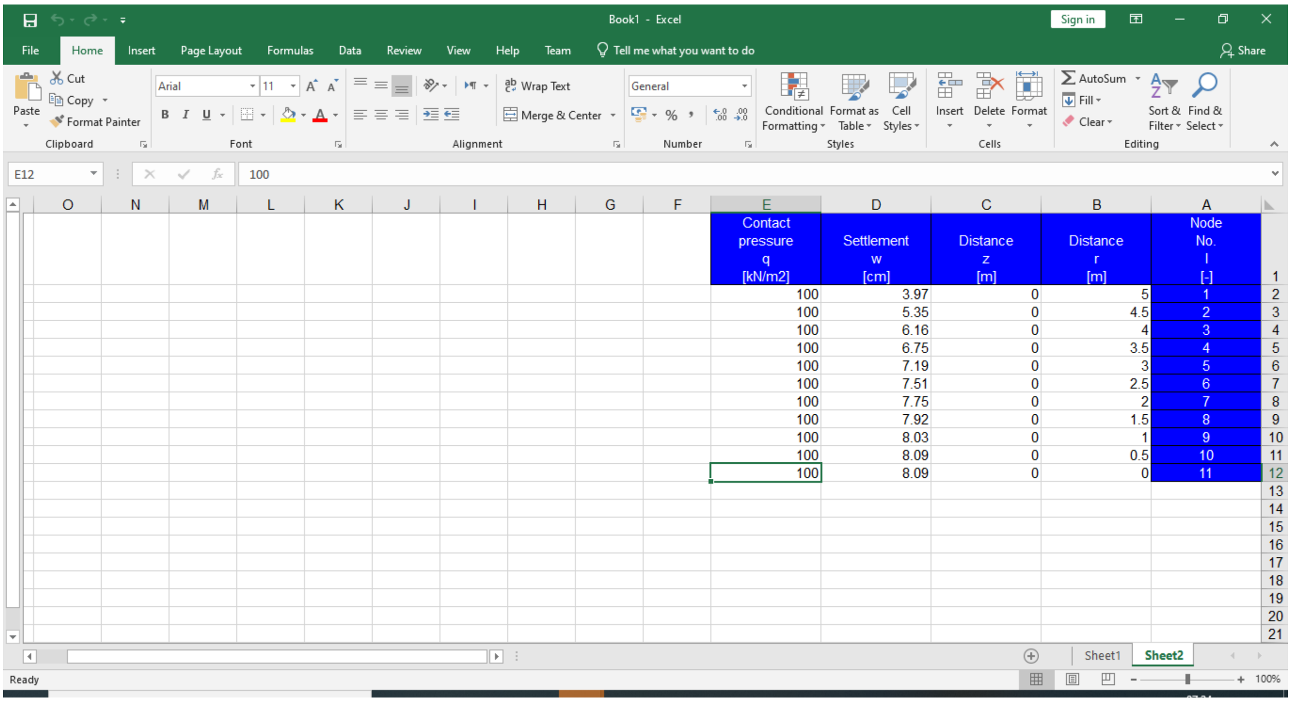
Figure 1.24 "Display Tables of Results" option box



Node No. I [-]	Distance r [m]	Distance z [m]	Settlement w [cm]	Contact pressure q [kN/m2]
1	5.00	0.00	3.97	100.0
2	4.50	0.00	5.35	100.0
3	4.00	0.00	6.16	100.0
4	3.50	0.00	6.75	100.0
5	3.00	0.00	7.19	100.0
6	2.50	0.00	7.51	100.0
7	2.00	0.00	7.75	100.0
8	1.50	0.00	7.92	100.0
9	1.00	0.00	8.03	100.0
10	0.50	0.00	8.09	100.0
11	0.00	0.00	8.09	100.0

Figure 1.25 List of loading data

Example 1



Contact pressure q [kN/m2]	Settlement w [cm]	Distance z [m]	Distance r [m]	Node No. I [-]
100	3.97	0	5	1
100	5.35	0	4.5	2
100	6.16	0	4	3
100	6.75	0	3.5	4
100	7.19	0	3	5
100	7.51	0	2.5	6
100	7.75	0	2	7
100	7.92	0	1.5	8
100	8.03	0	1	9
100	8.09	0	0.5	10
100	8.09	0	0	11

Figure 1.26 Exported data in MS Word

Example 2

Analysis of an annular plate resting on *Winkler's* medium

Contents Page

1 Description of the problem

An example of an annular plate resting on *Winkler's* medium is selected, to illustrate some features of *ELPLA* for analyzing shell elements using circular and annular elements.

2 Geometry and properties

A simply supported annular plate subjected to a uniform load resting on *Winkler's* medium is chosen as shown in Figure 2.1. Load on the plate, plate radii, elastic properties of the soil and the plate are:

Inner radius of the plate	r_1	= 2.5	[m]
Outer radius of the plate	r_2	= 5	[m]
Thickness of the plate	t	= 0.25	[m]
Uniform load on the raft	p	= 200	[kN/m^2]
Modulus of sub grade reaction of the soil	k_s	= 10000	[kN/m^3]
Young's modulus of the plate material	E_c	= 2.7×10^7	[kN/m^2]
Poisson's ratio of the plate material	ν_c	= 0.2	[-]

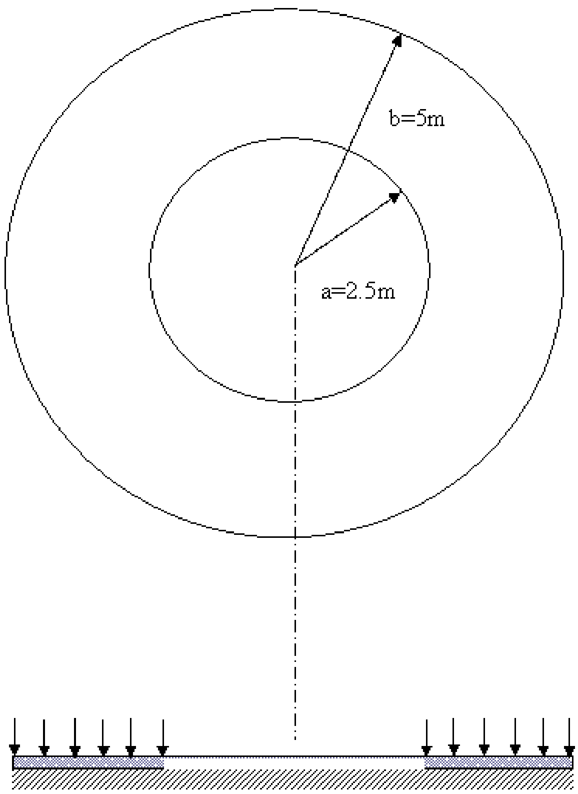


Figure 2.1 Annular plate subjected to a uniform load

3 Analysis of the plate

The available method "Constant Modulus of Subgrade Reaction /2" in *ELPLA* is used here to determine the vertical displacement and moment of the plate on *Winkler's* medium. Figure 2.2 shows the annular plate with 10 annular regions and supports.

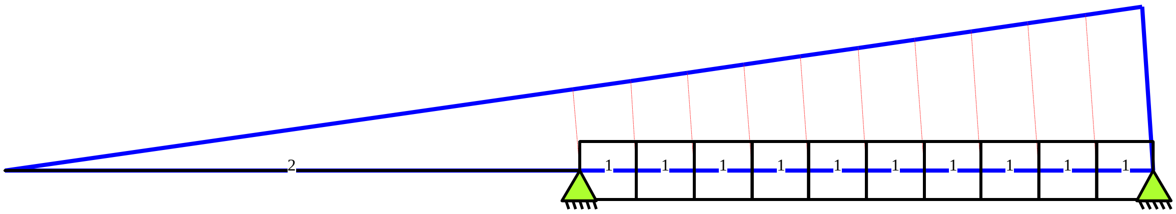


Figure 2.2 Annular plate with 10 annular regions and supports

4 Creating the project

In this section, the user will learn how to create a project for analyzing an annular plate resting on *Winkler's* medium. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 2.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of Forms. The first Form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 2.3).

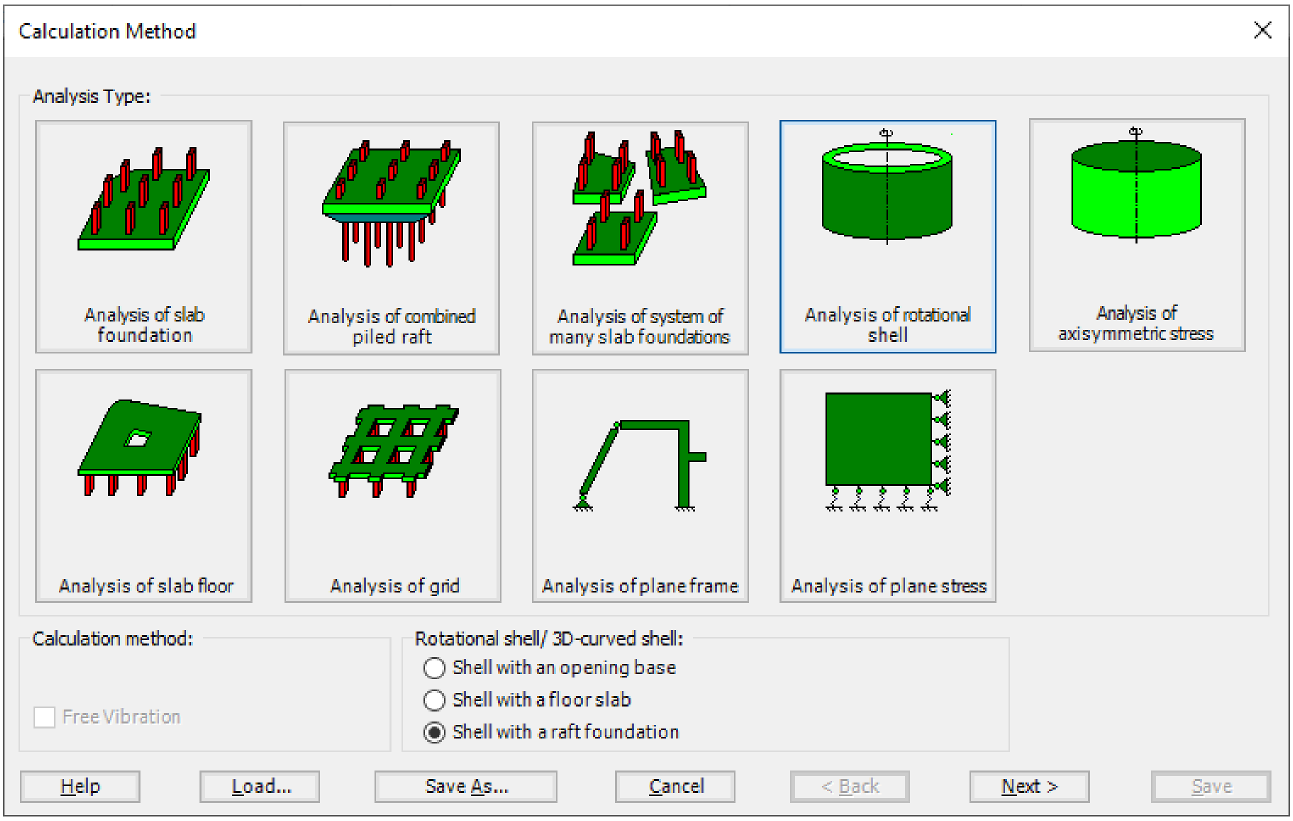


Figure 2.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 2.3, define the analysis type of the problem. As the analysis type is an annular plate, select "Analysis of rotational shell" button, and check "Shell with a raft foundation" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Calculation Methods" Form appears, Figure 2.4.

To define the calculation method:

- Select the calculation method "2/3 Constant/ Variable Modulus of Subgrade Reaction"
- To determine the modulus of subgrade reaction, select "Modulus is defined by the user" option
- Click "Next" button to go to the next Form

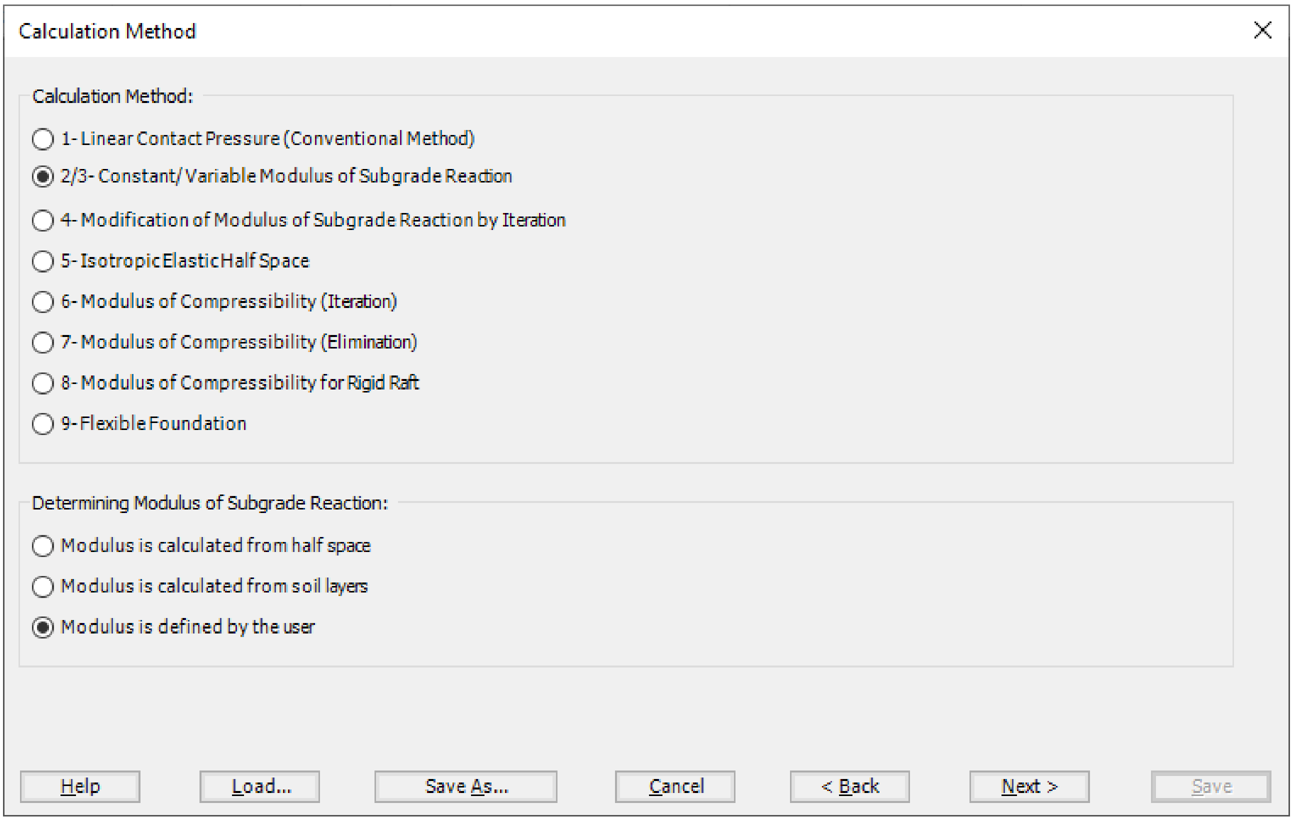


Figure 2.4 "Calculation Methods" Form

The last Form in the wizard is the "Options" Form, Figure 2.5. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Select "Supports/ Boundary Conditions", then click the "Save" button.

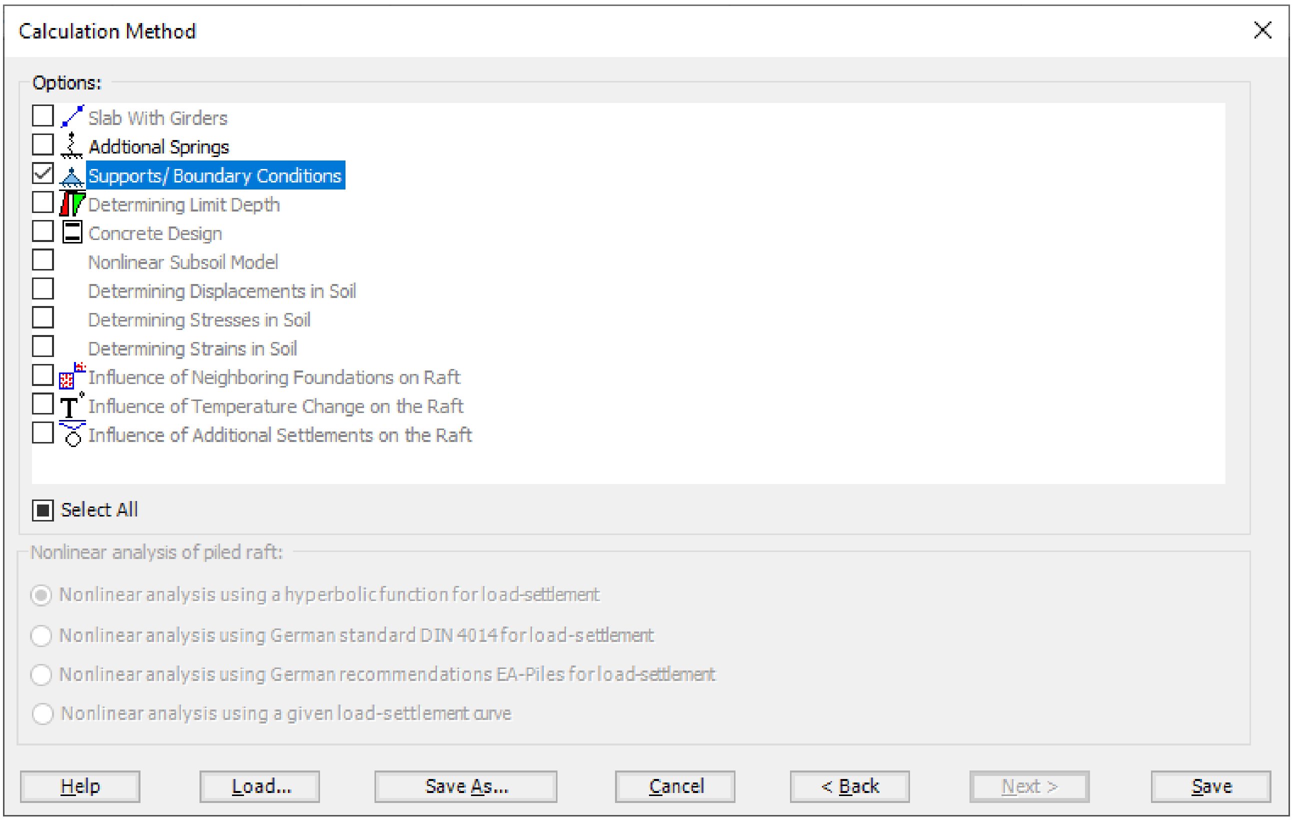


Figure 2.5 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 2.6. In this dialog box type a file name for the current project in "File name" edit box. For example, type "annular plate". *ELPLA* will use automatically this file name in all reading and writing processes.

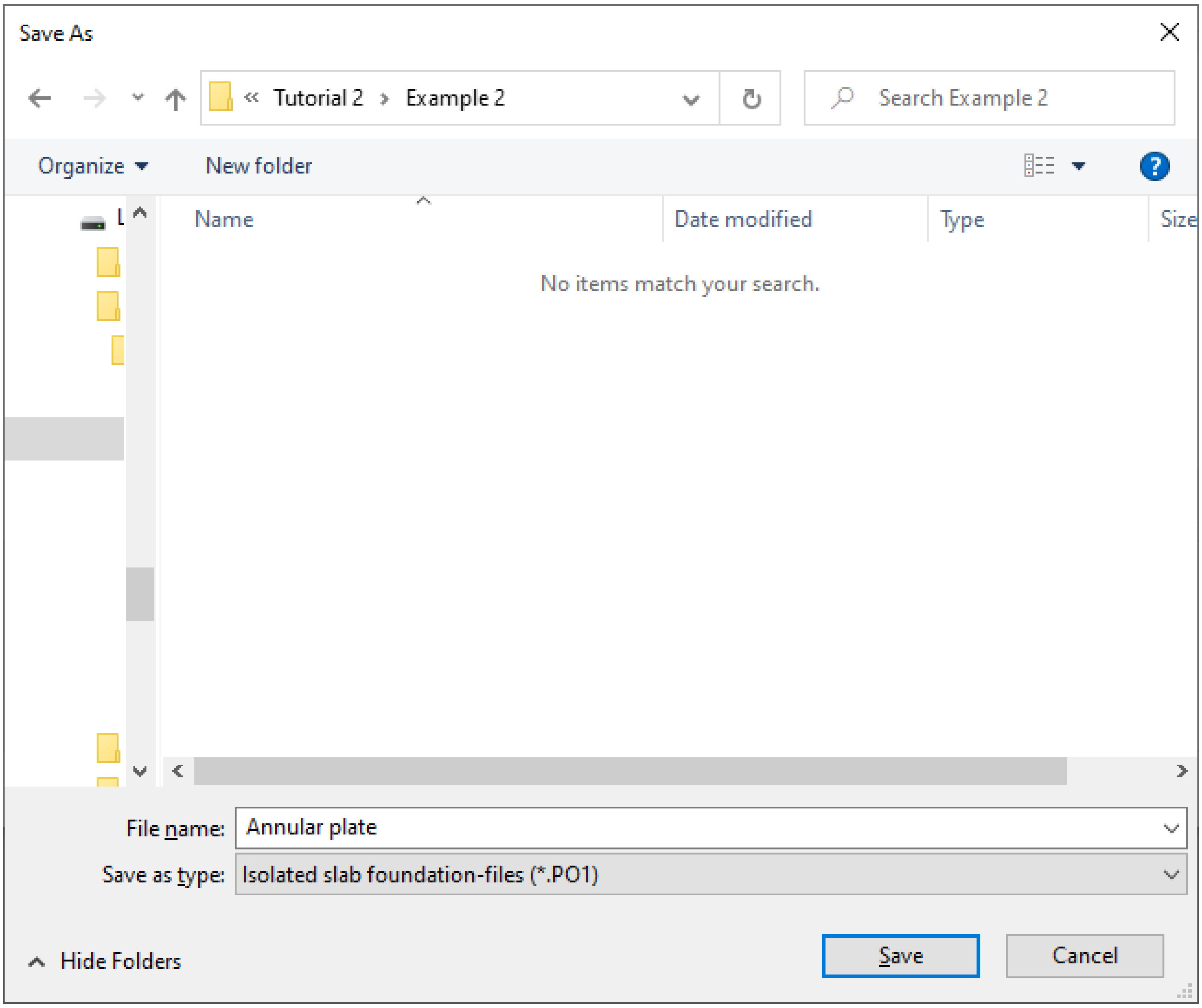


Figure 2.6 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [annular plate] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 2.7 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of an annular plate resting on *Winkler's* medium"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

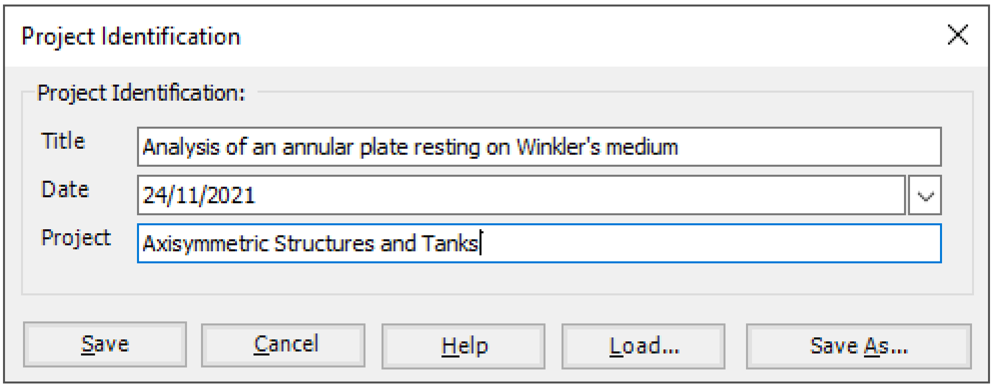


Figure 2.7 "Project Identification" dialog box

4.3 FE-Net data

For the given problem, the shell has an annular shape with an outer radius of b = 5 [m] and an inner radius of a = 2.5 m. To define the FE-Net for this plate, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 2.8. This wizard will guide you through the steps required to generate a FE-Net.

The first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of shells. These net templates are used to generate standard nets.

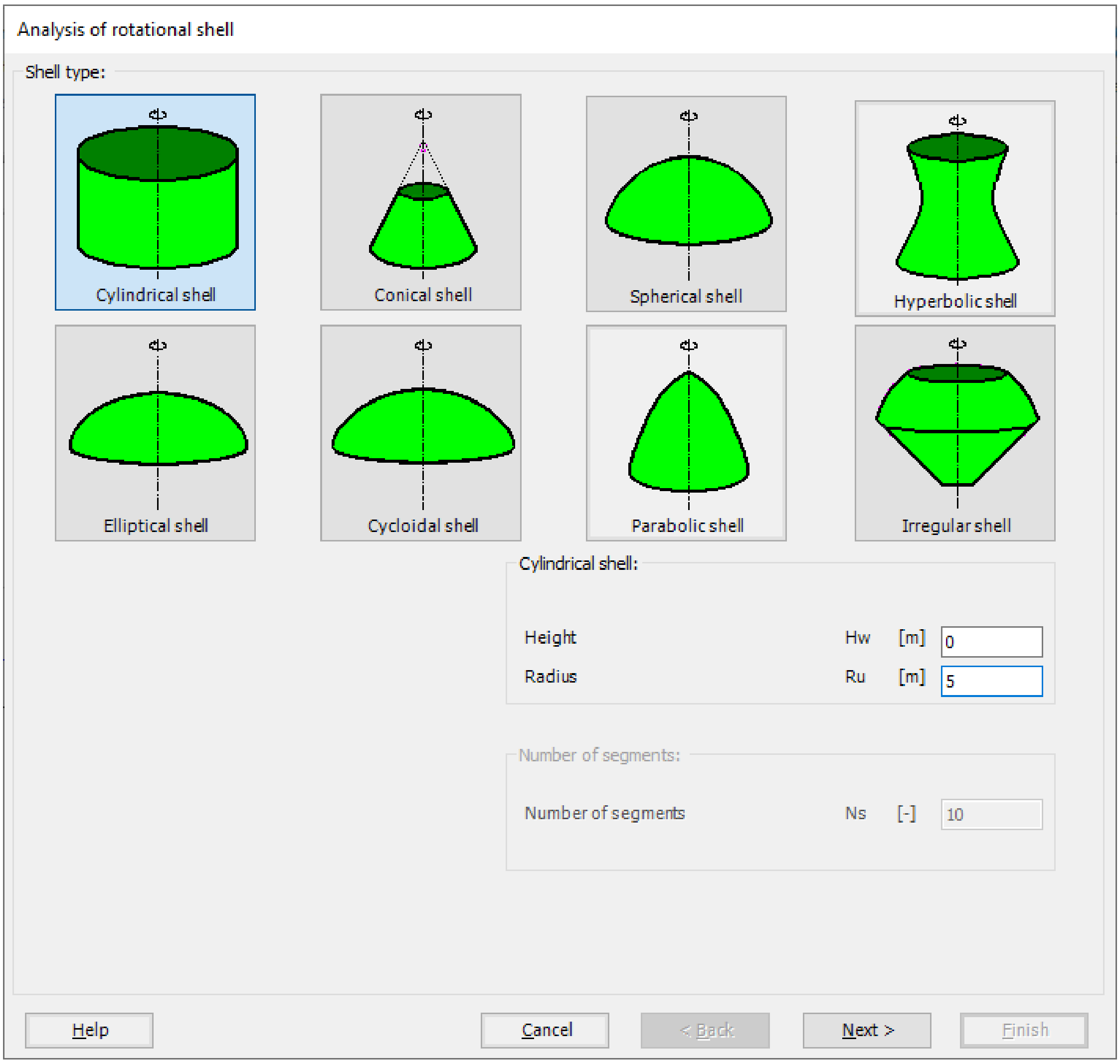


Figure 2.8 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Cylindrical shell" button
- Type 0 in the "Height" edit box, as the example is an annular plate
- Type 5 in the "Radius" edit box, as the outer radius is b = 5 m
- Click "Next" button to go to the next Form

-

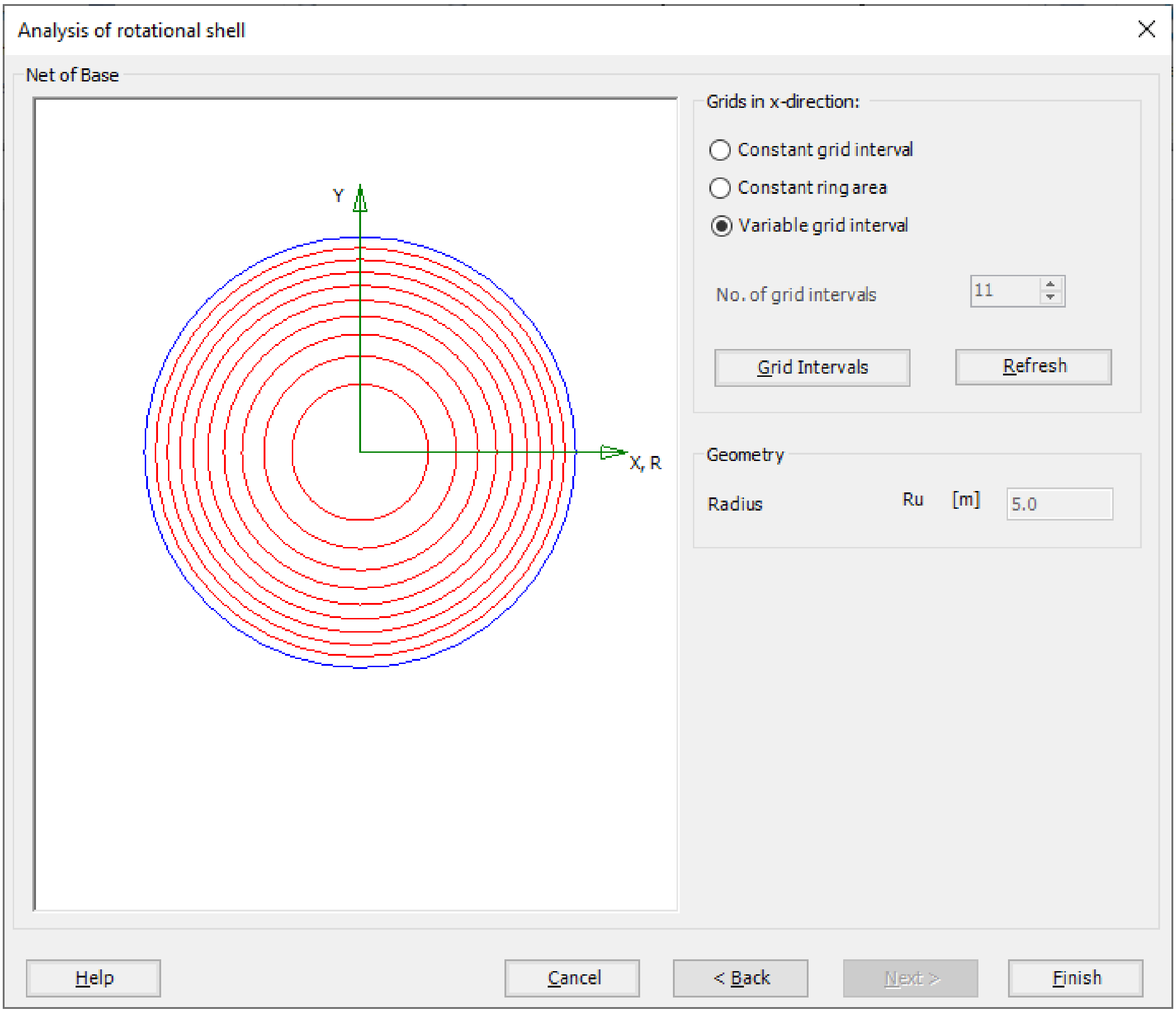


Figure 2.9 "Net of Base" Form

The next Form of the "Analysis of rotational shell" wizard is the "Net of Base" Form Figure 2.9.

To edit the grid spacing in *x*-direction, do the following steps in "Grid in *x*-direction" frame:

- Choose "Variable grid interval" check box.
- Click "Grid Intervals" button, the following "Grid intervals in *x*-direction" form appears Figure 2.10
- Define the grid intervals as the following

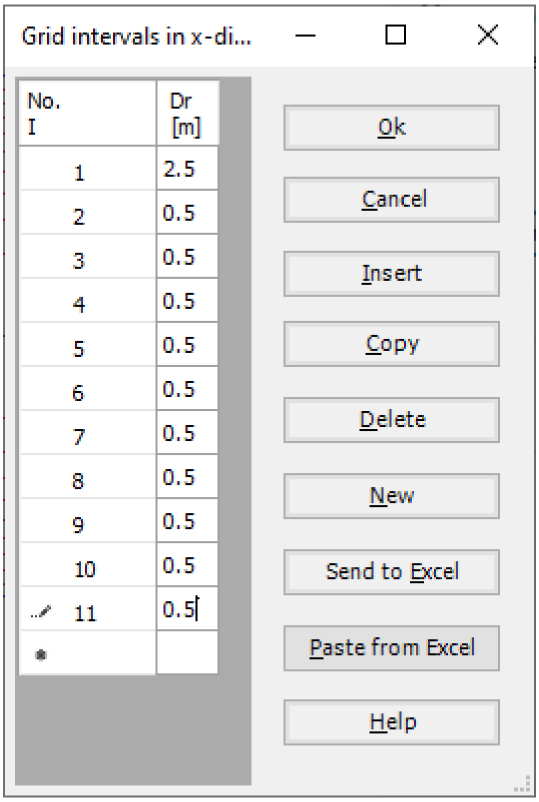


Figure 2.10 "Grid intervals in *x*-direction" Form

ELPLA will generate a sector from the annular area with 11 circular elements. The following Window in Figure 2.11 appears with the generated net.

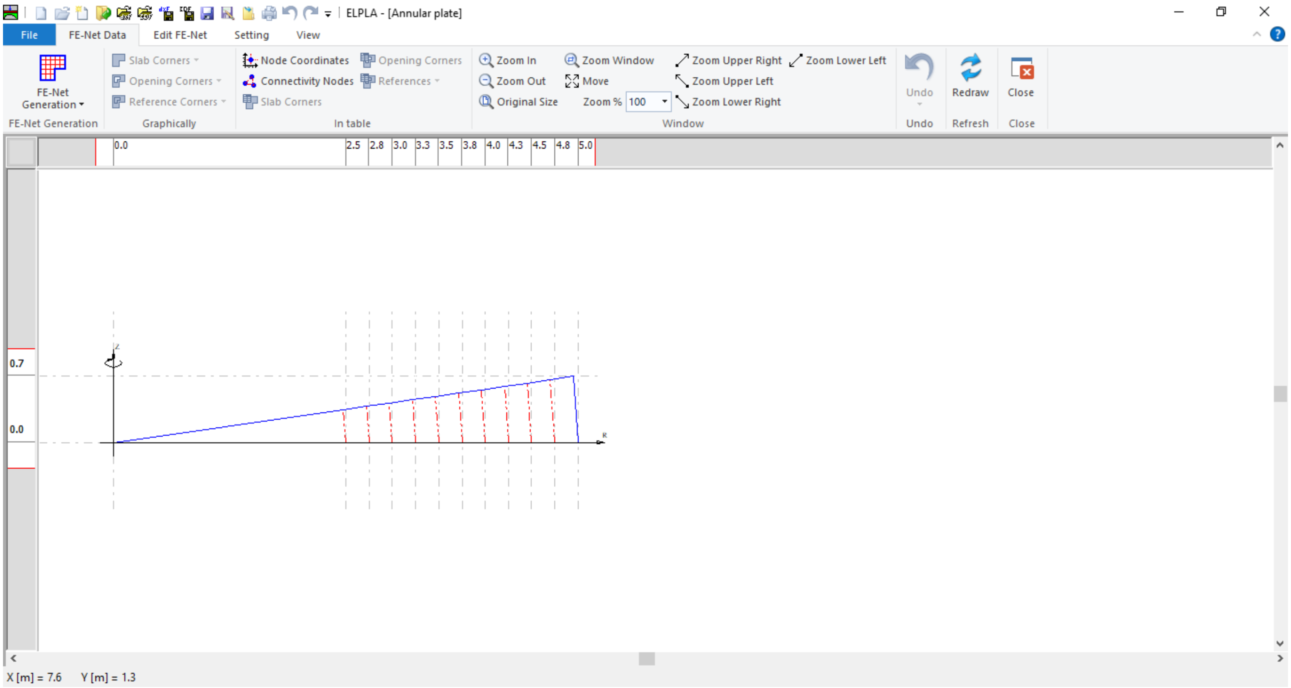


Figure 2.11 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 2.11 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 2.11 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Soil Properties

To define the soil properties, choose "Soil Properties" command from “Data” Tab. The following "Soil Properties" form in Figure 2.12 appears, enter the modulus of subgrade reaction of the soil and the ground water depth under the ground surface. Other data for this example is not required.

Soil data

Modulus of subgrade reaction	k_s	= 10 000	[kN/ m^3]
Ground water depth under the surface	Gw	= 1	[m]

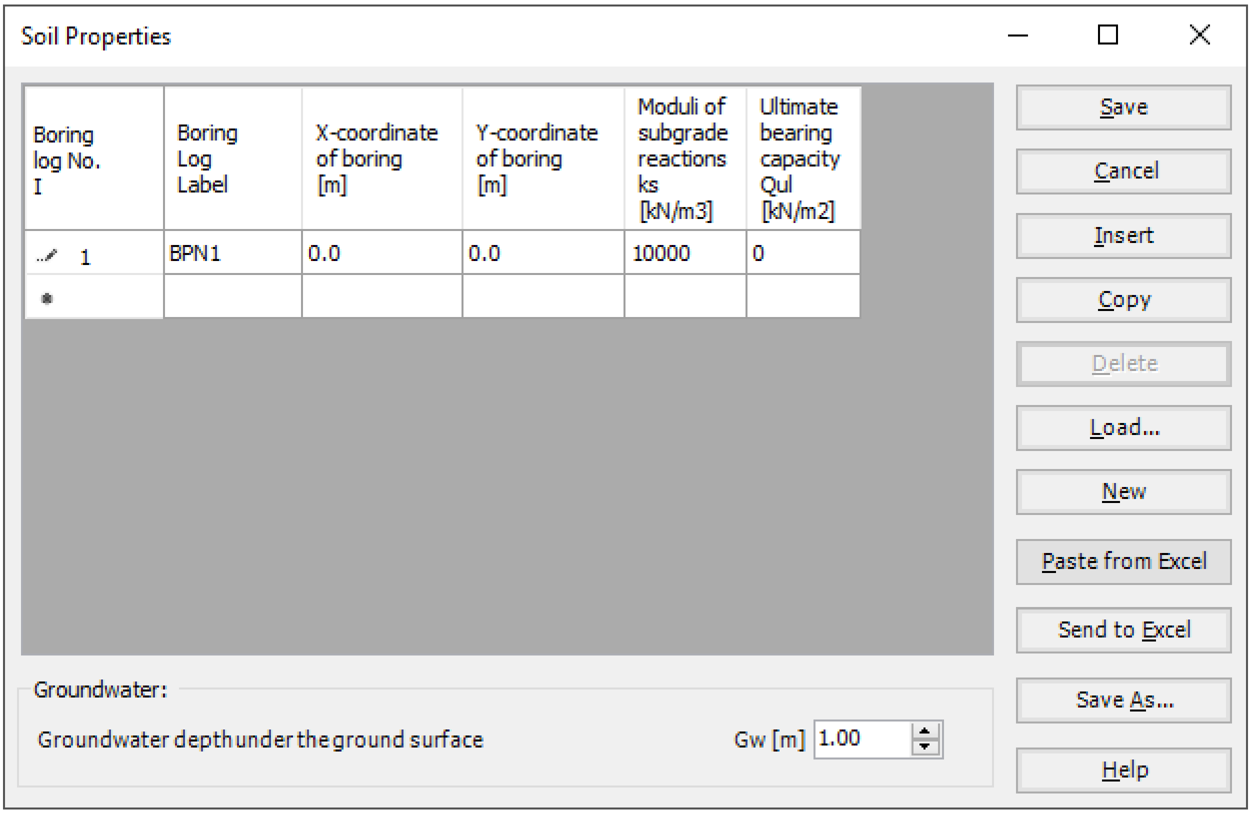


Figure 2.12 "Soil Properties" form

4.5 Shell properties

To define the annular plate properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 2.13 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups and unit weight of the plate. Any other data corresponding to the shell properties in the program menus are not required for this example.

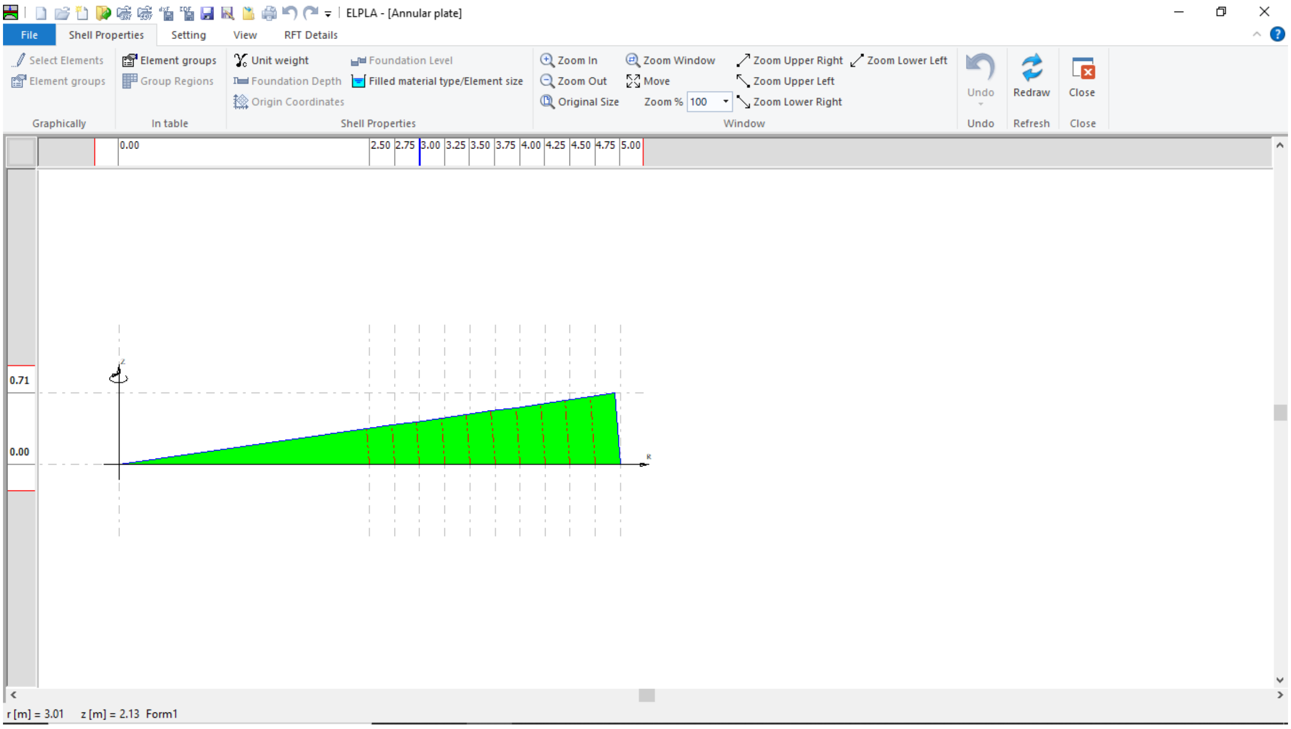


Figure 2.13 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 2.14 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness, the thickness of the inner ring is eliminated by defining its slab thickness by zero. Then click "OK" button.

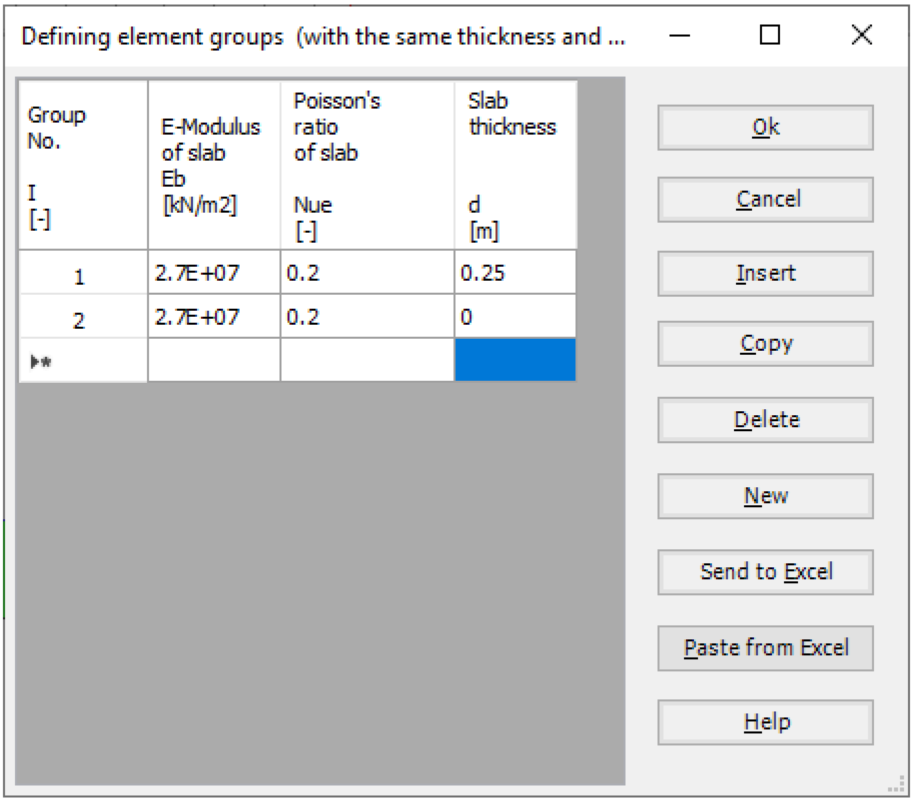


Figure 2.14 "Defining element groups" list box

Choose "Group Regions" command from "In table" menu. The following list box in Figure 2.15 appears. As the inner radius is a = 2.5m, and the elements of the plate differ in thickness, Edit the "Group no" value for each element as the following. Then click "OK" button.

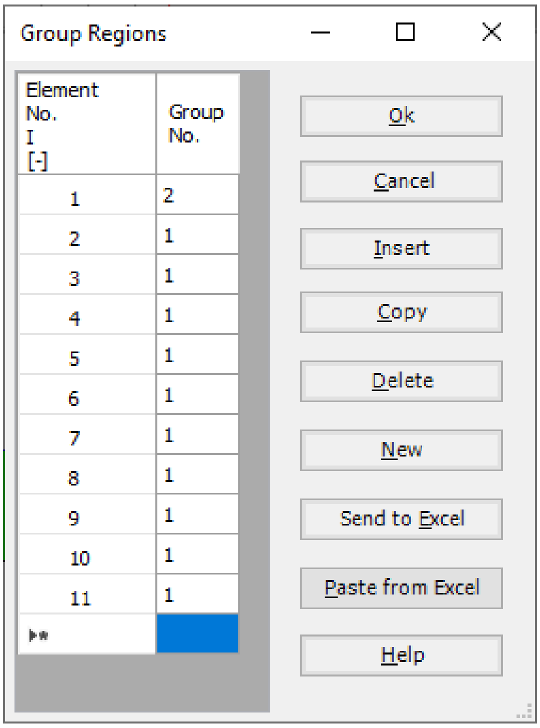


Figure 2.15 "Group Regions" Form

To enter the unit weight of the plate, choose "Unit weight" command from "Shell Properties" menu in the window of Figure 2.13. The following dialog box in Figure 2.16 with a default unit weight of 25 [kN/m^3] appears. Type 800 in the "Unit weight" edit box, note that the unit weight of the plate material is used to determine the uniform load q [kN/m^2] on the annular area, which is equal to $\gamma_b \times d$. Click "OK" button.

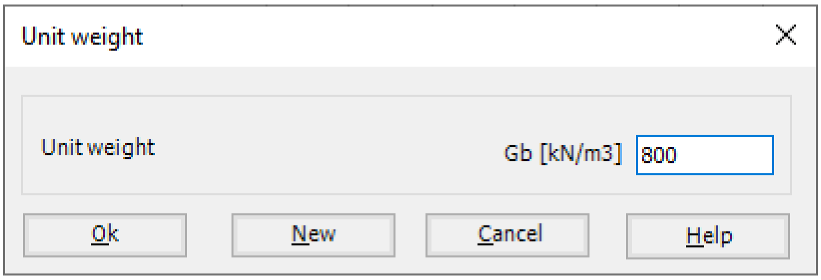


Figure 2.16 "Unit weight" dialog box

Now the shell properties have been entered and the sector of the plate appears as follows in Figure 2.17.

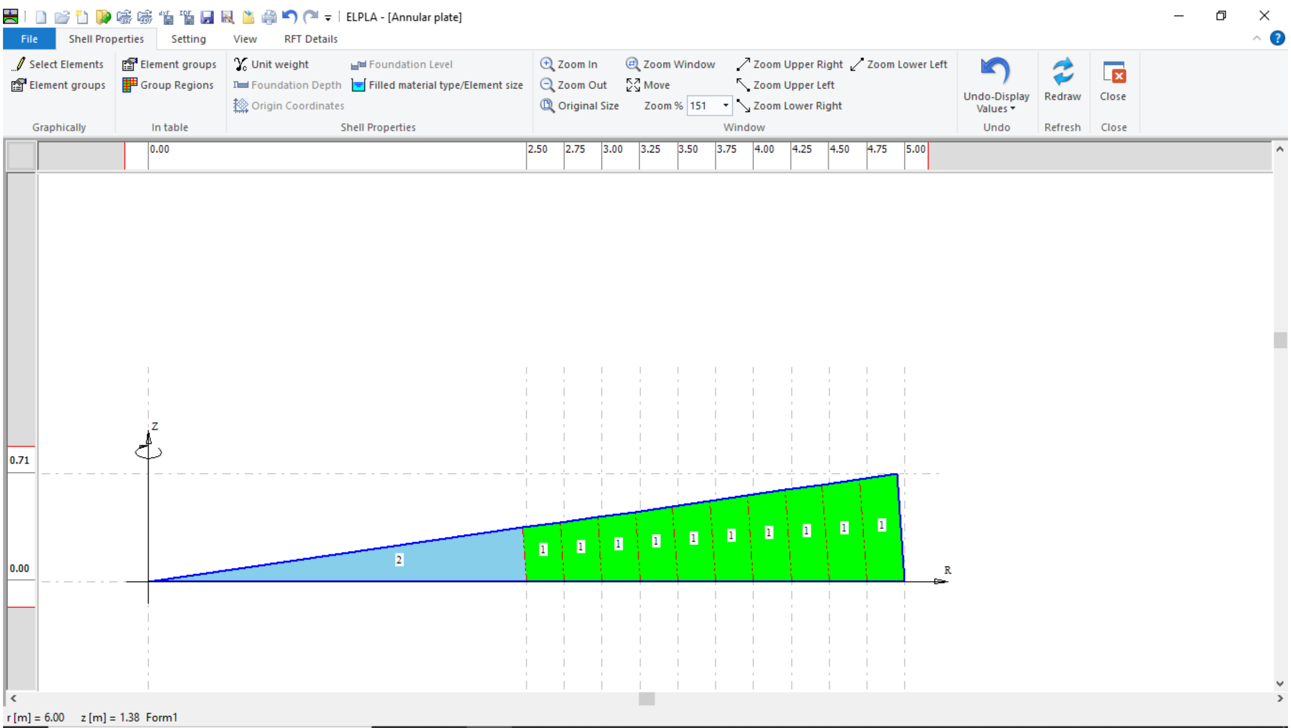


Figure 2.17 "Shell Properties" Window

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

- Choose "Save" command from "File" menu in Figure 2.17 to save the shell properties
- Choose "Close" command from "File" menu in Figure 2.17 to close the "Shell Properties" window and return to *ELPLA* main window

4.6 Supports/ boundary conditions

To define supports choose "Supports/ Boundary Conditions" command from “Data” Tab. The following Tab in Figure 2.18 appears.

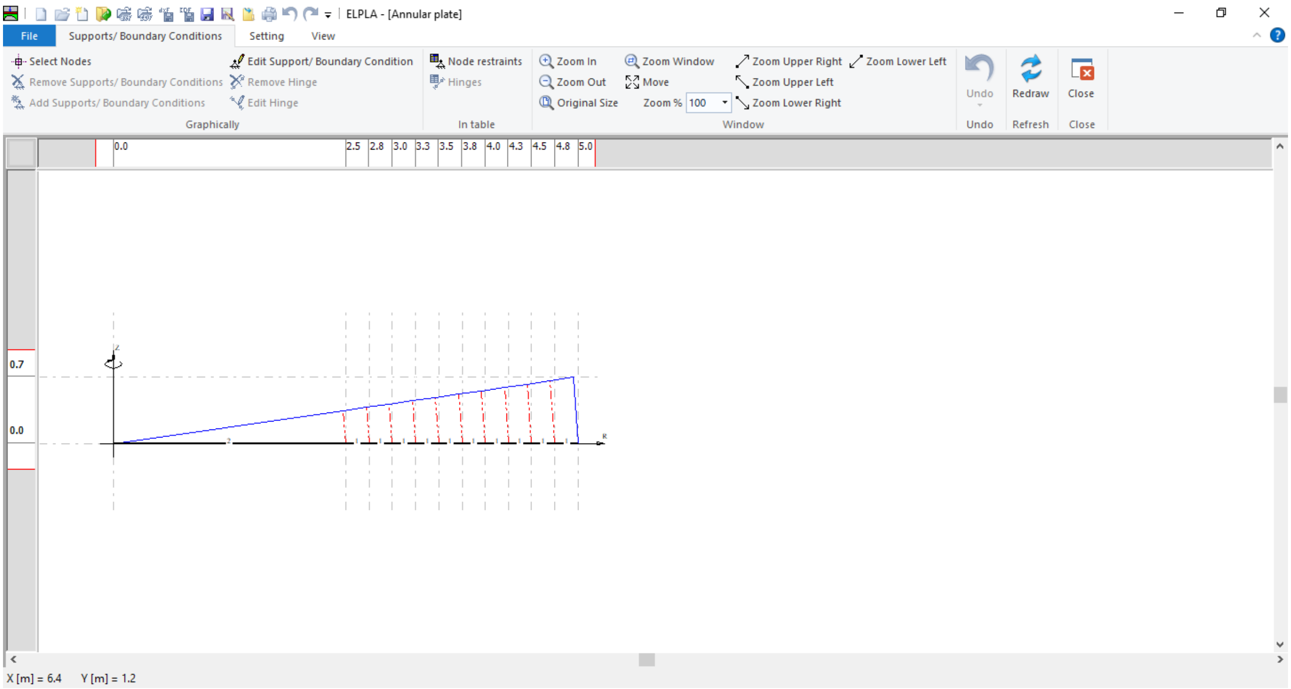


Figure 2.18 "Supports/ Boundary Conditions" Window

To define the supports

- Choose "Select Nodes" command from "Graphically" menu in Figure 2.18. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nodes that have supports as shown in Figure 2.19
- After selecting nodes of supports, choose "Add Supports/ Boundary Conditions" command from "Graphically" menu (Figure 2.18). The "Supports/ Boundary Conditions" dialog box in Figure 2.20 appears

Example 2

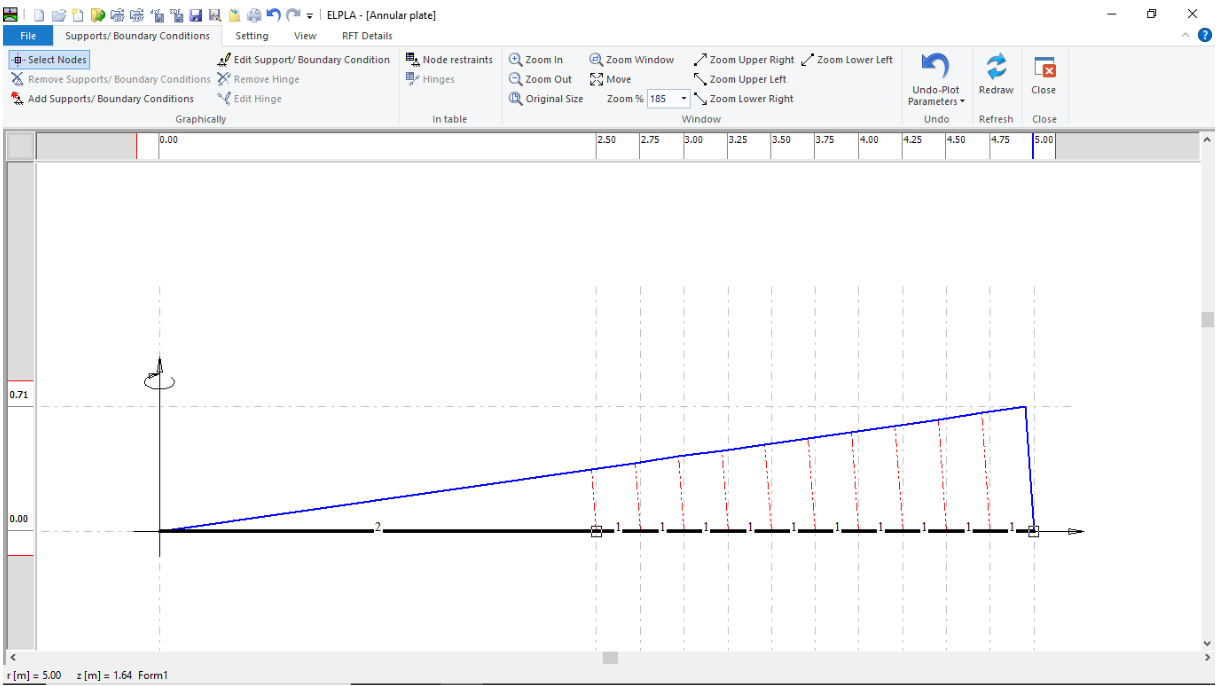


Figure 2.19 Selection of nodes that have supports

In this dialog box

- Type 0 in the "Displacement w" edit box to define the vertical supports, as the annular plate is prevented from moving in the vertical direction at its ends
- Click "OK" button

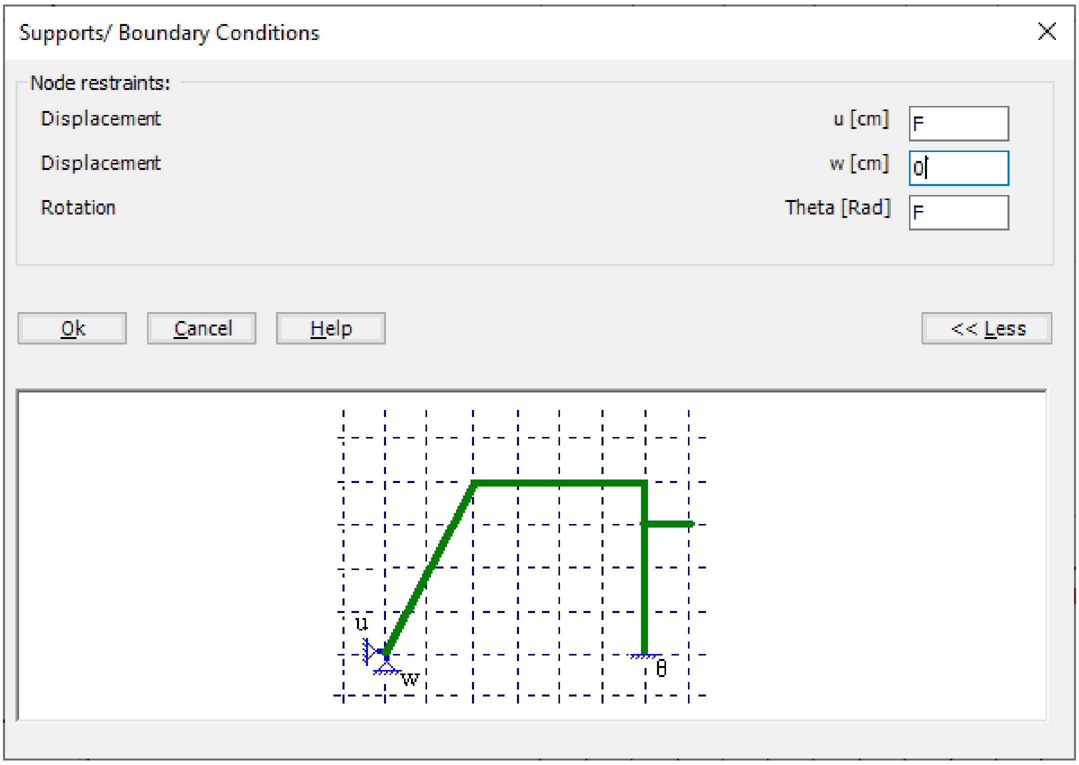


Figure 2.20 "Supports/ Boundary Conditions" dialog box

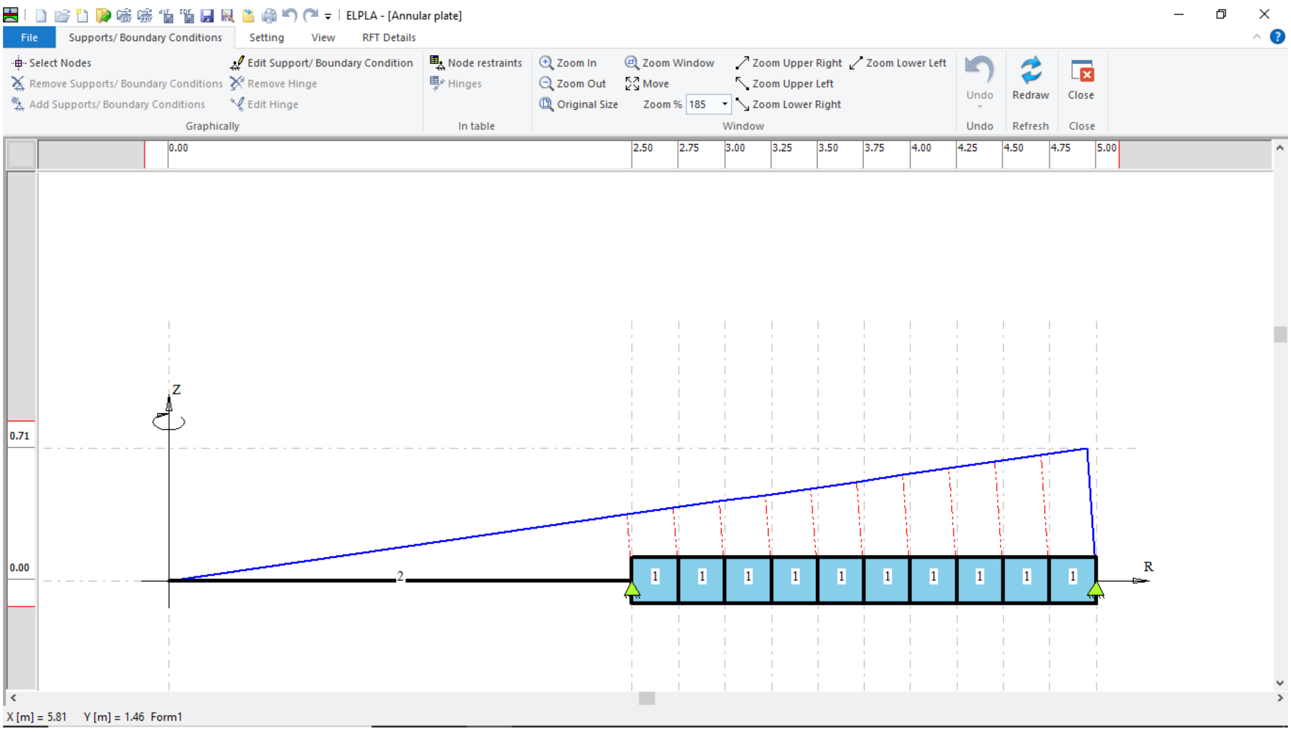


Figure 2.21 Supports on the screen

After entering supports, do the following two steps

- Choose "Save " command from "File" menu in Figure 2.21 to save the data of supports
- Choose "Close" command from "File" menu in Figure 2.21 to close the "Supports/ Boundary conditions" window and return to the main window

4.7 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following Window in Figure 2.22 appears. In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 2.22. For this example, there is not applied load, as the load has been already defined by the unit weight of the plate.

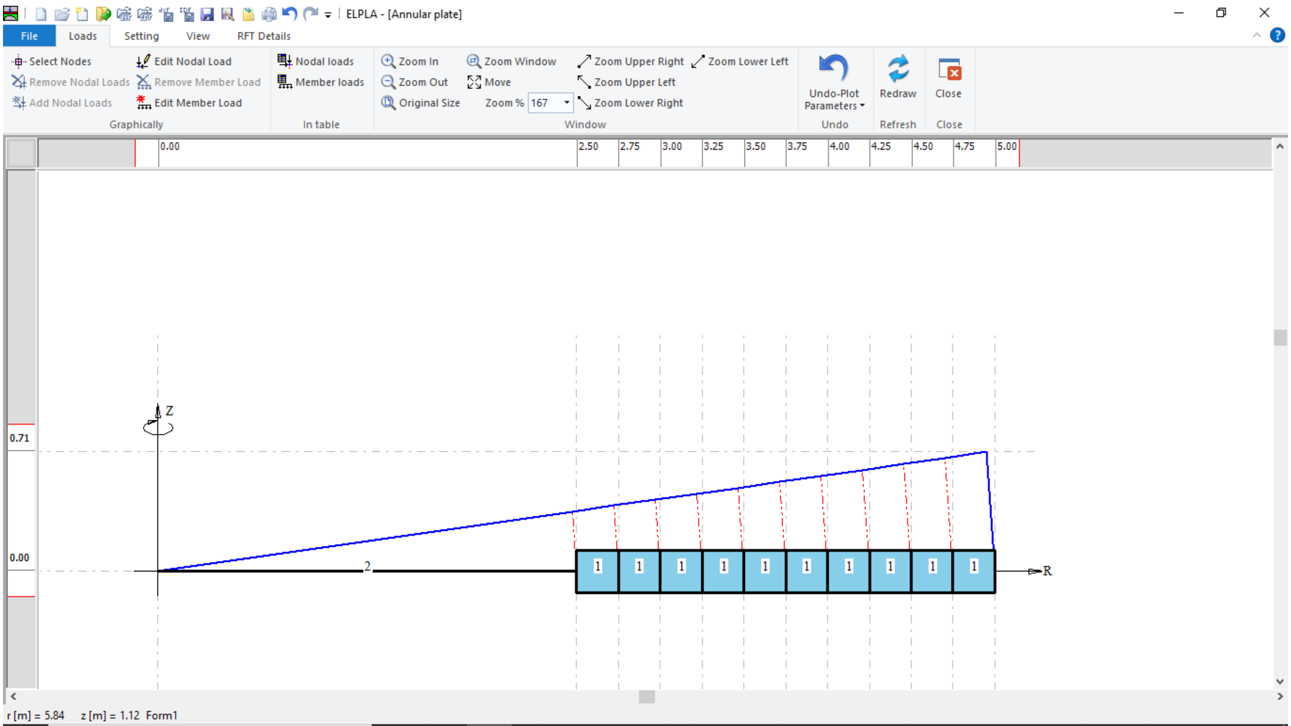


Figure 2.22 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 2.22 to save the load data
- Choose "Close" command from "File" menu in Figure 2.22 to close the "Loads" window and return to *ELPLA* main window

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

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 2.23.

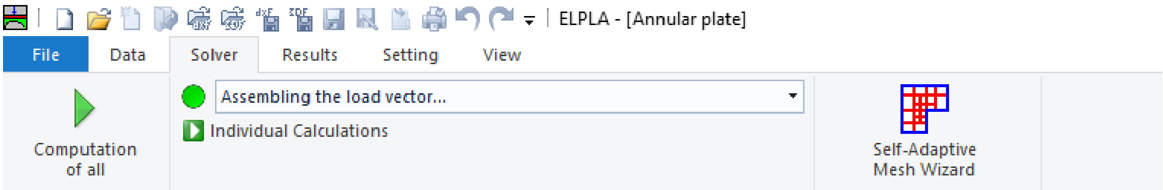


Figure 2.23 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

- Assembling the load vector
- Determining the modulus of subgrade reaction
- Assembling the slab stiffness matrix
- Solving the system of linear equations (band matrix)
- Determining deformation, internal forces, contact pressures

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

Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab Window.

Analysis progress

Analysis progress menu in Figure 2.24 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

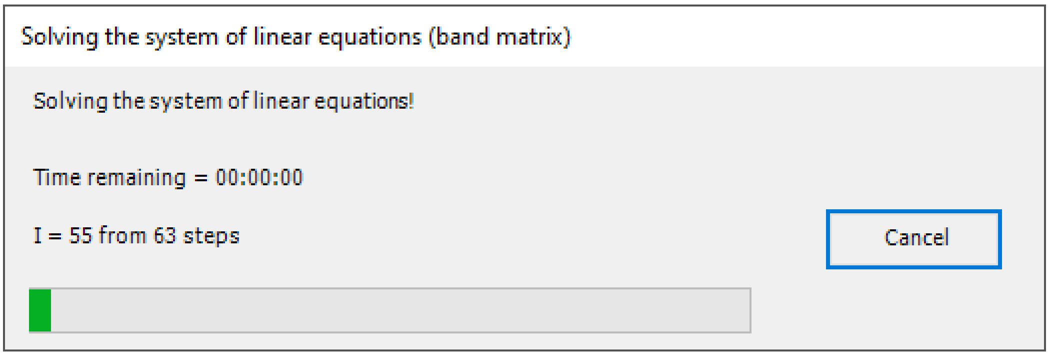


Figure 2.24 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 2.25. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

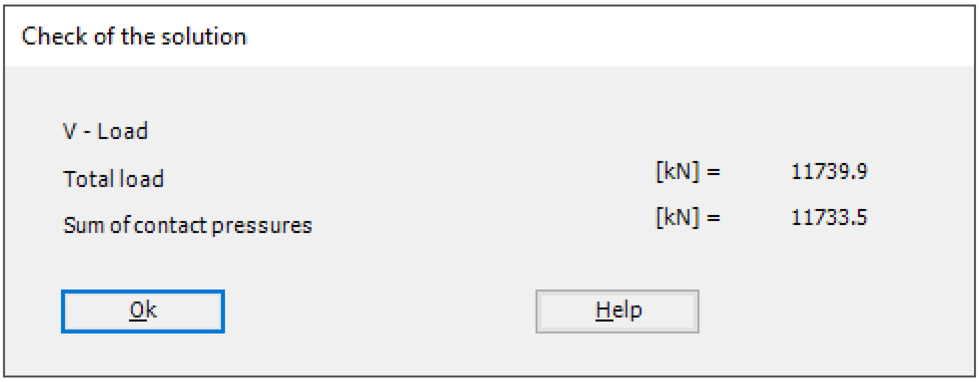


Figure 2.25 Menu "Check of the solution"

Ignore the elimination of the negative contact pressure which appears at the inner ring.

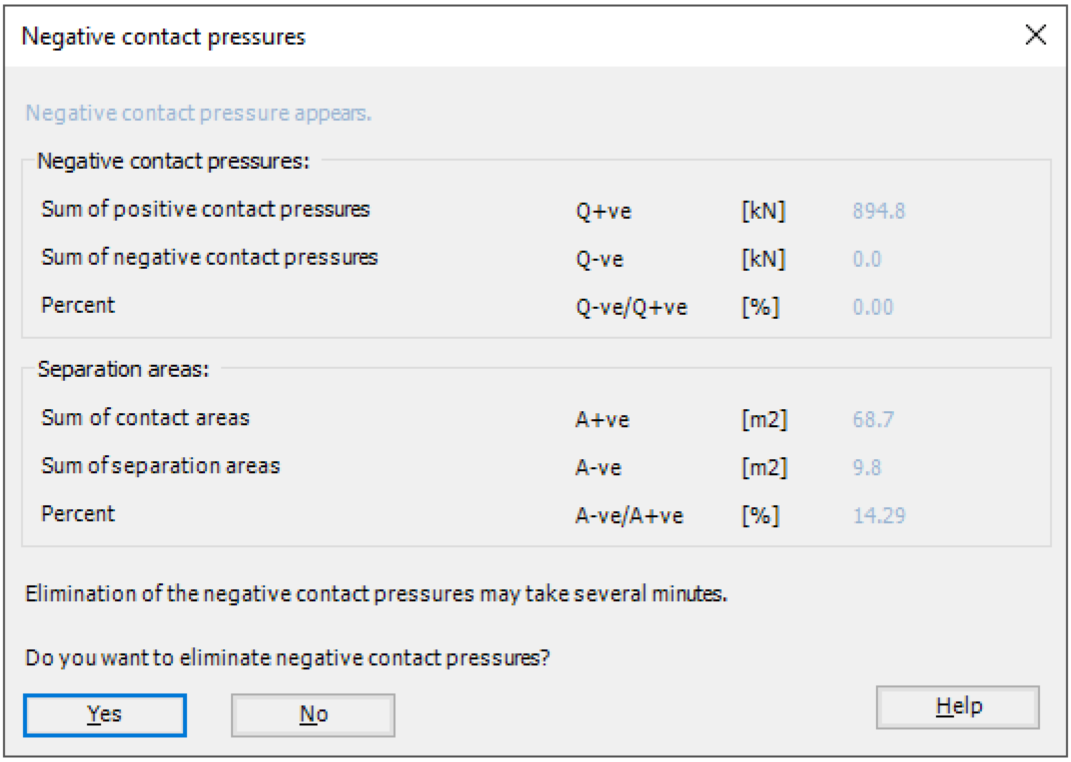


Figure 2.26 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab. To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 2.27).

Figure 2.27 "Results" Tab

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

- Data in the plan
- Support reactions
- Rotational shell results
- Sections in shell wall
- Sections in shell base
- Display tables of data
- Display tables of results

To view the meridional moments in the shell base

- Choose "Sections in shell base" command from "Section" menu. The following option box in Figure 2.28 appears
- In the "Sections in shell base" option box, select "Meridional moments *My*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 2.29.

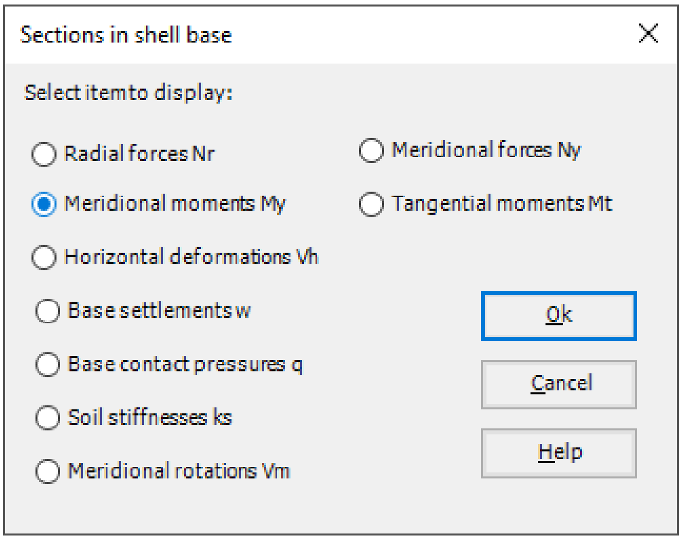


Figure 2.28 "Sections in shell base" option box

Example 2

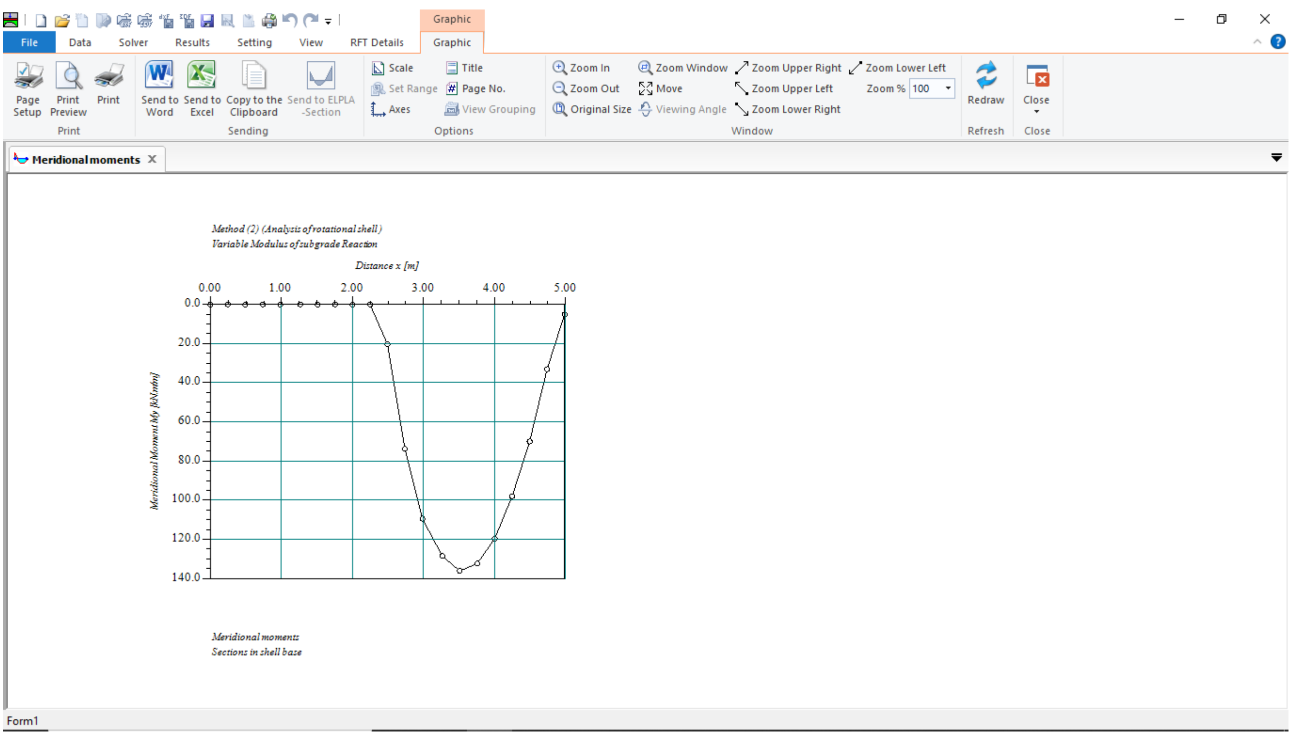


Figure 2.29 Meridional moments in shell base

To view element groups of the plate

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 2.30 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

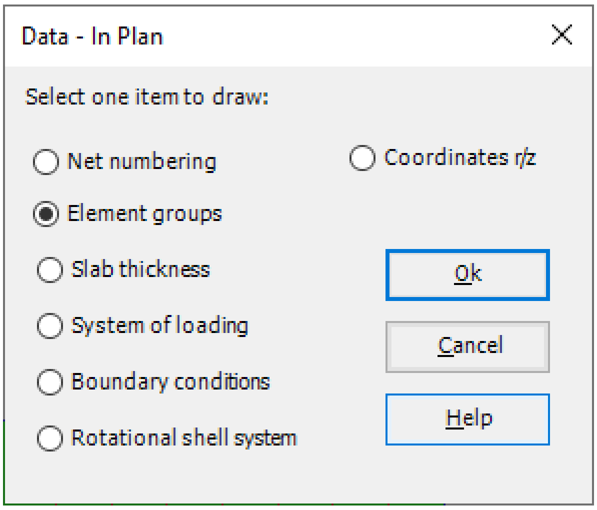


Figure 2.30 "Data – In Plan" option box

To draw the thickness of the annular plate

- Choose "Plot Parameters" command from "Setting" Tab. The "Plot Parameters" dialog box in Figure 2.31 appears
- In the "FE-Net" Tab, check the "Draw girder thickness" check box in the "Rotational shell system" dialog box
- Click "OK" button

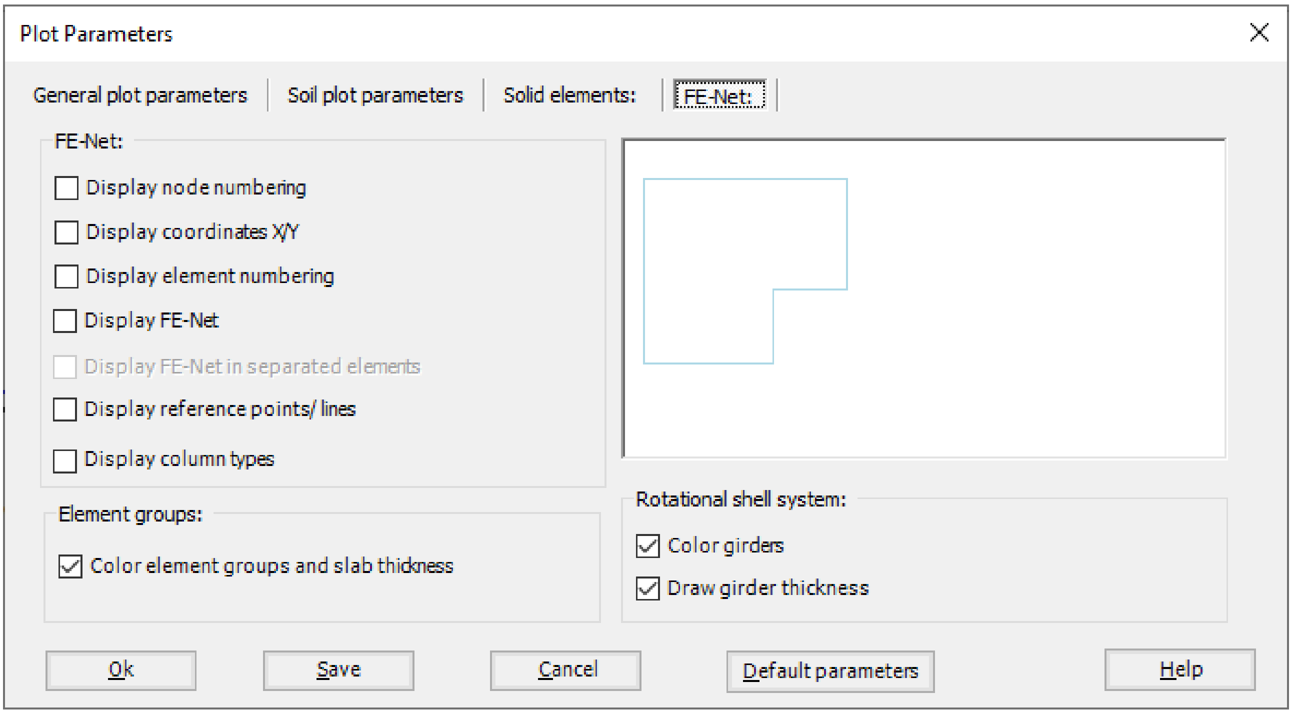


Figure 2.31 "Plot Parameters" dialog box

To view the reactions of the supports on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 2.32 appears
- In this check group box, check "Supports /Boundary Conditions" and "Supports Reactions RV" check boxes
- The user can choose any other data to be displayed
- Click "OK" button

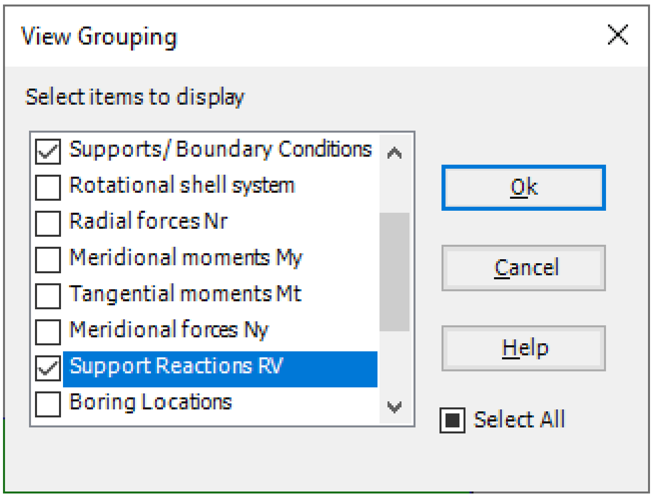


Figure 2.32 "View Grouping" check group box

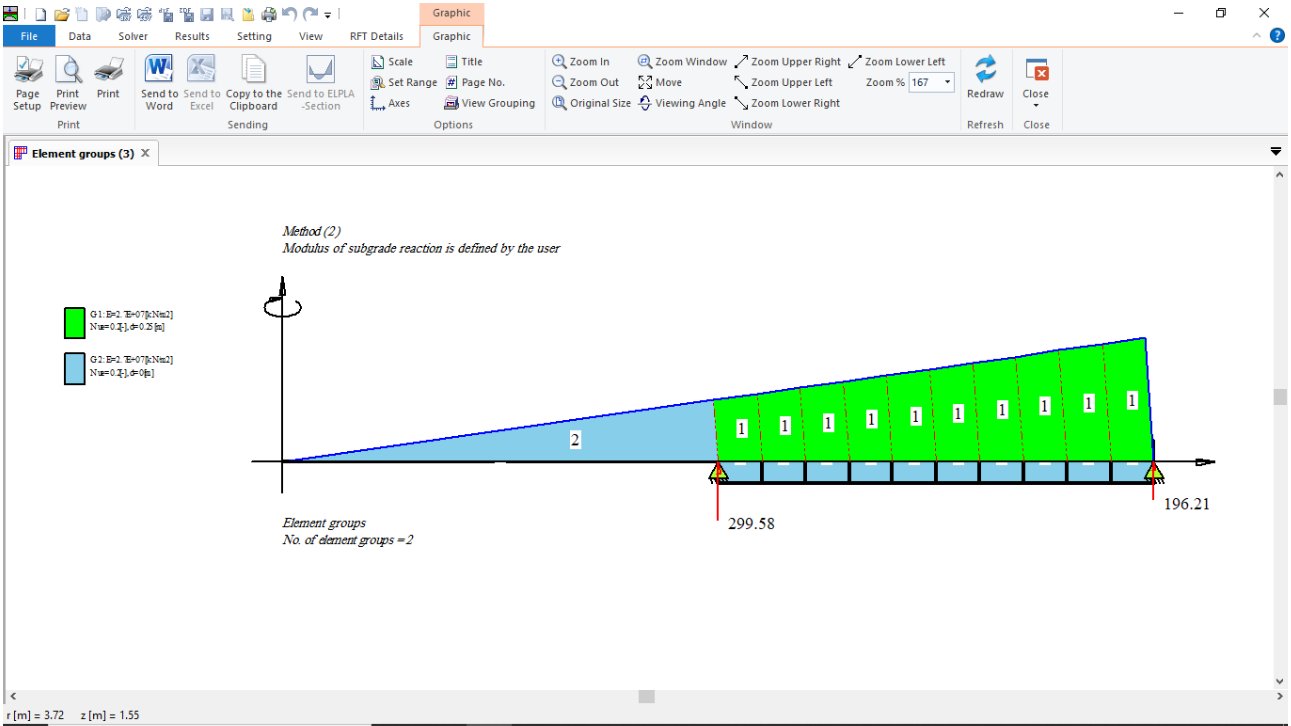


Figure 2.33 Element groups of the annular plate

Example 3

Analysis of a tank with a fixed base

Contents Page

1 Description of the problem

An example of an axi-symmetrically circular cylindrical tank with a fixed base is selected to illustrate some features of *ELPLA* for analyzing shell elements.

2 Tank geometry and properties

A circular cylindrical tank of a radius of a = 7 [m] and a height of H = 5 [m] is considered as shown in Figure 3.1. Thickness of the tank wall is t = 0.25 [m]. The tank is filled with water. The lower edge of the tank is clamped. Figure 3.1 shows the circular cylindrical tank with its dimensions, while the tank material and unit weight of the water are listed in Table 3.1.

Table 3.1 Tank material and water unit weight

Modulus of Elasticity of the tank material	E_c	= 2×10^7^ [kN/m²]
Poisson's ratio of the tank material	ν_c	= 0.15 [-]
Unit weight of the water	γ_w	= 10 [kN/m³]

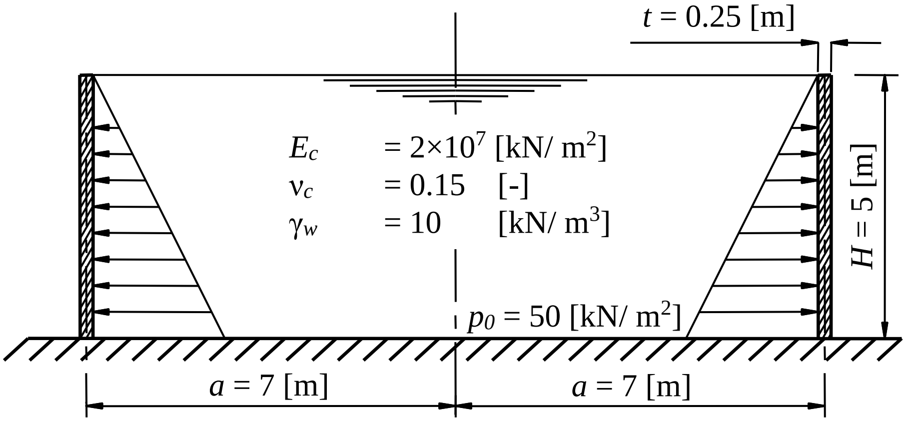


Figure 3.1 Cylindrical circular tank with dimensions

3 Numerical Analysis

The analysis of circular cylindrical shell tank is carried out using the finite element method. In the analysis, the height of the tank is divided into 50 equal segments. In each segment, element size is 0.1 [cm] as shown in Figure 3.2.

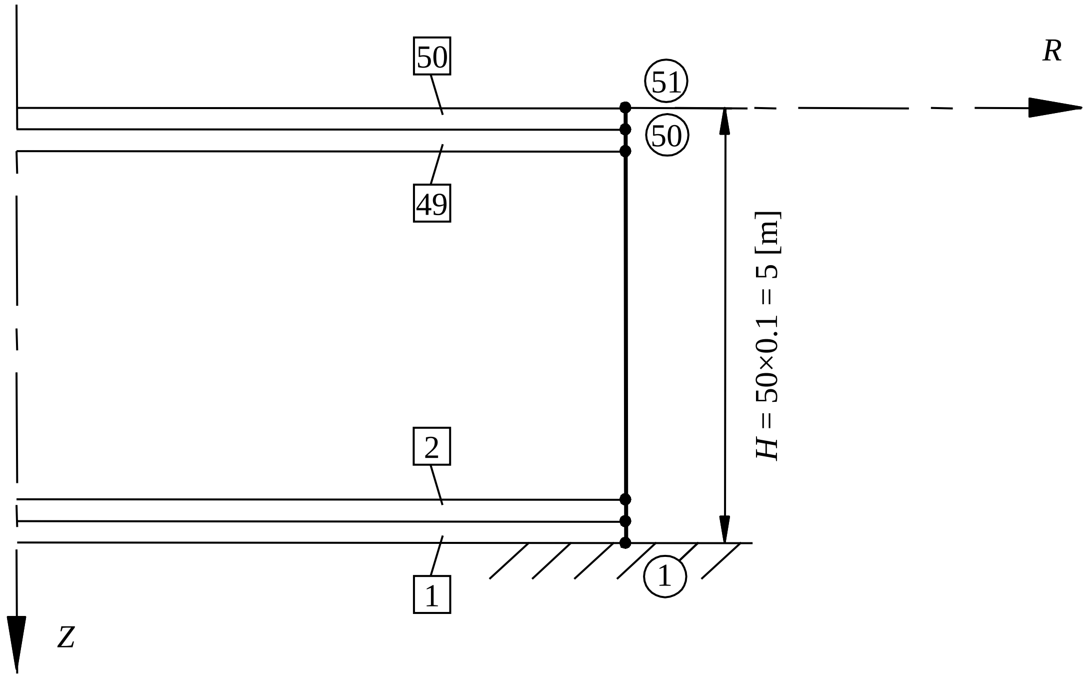


Figure 3.2 Finite element mesh of the tank

4 Creating the project

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

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 3.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of Forms. The first Form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 3.3).

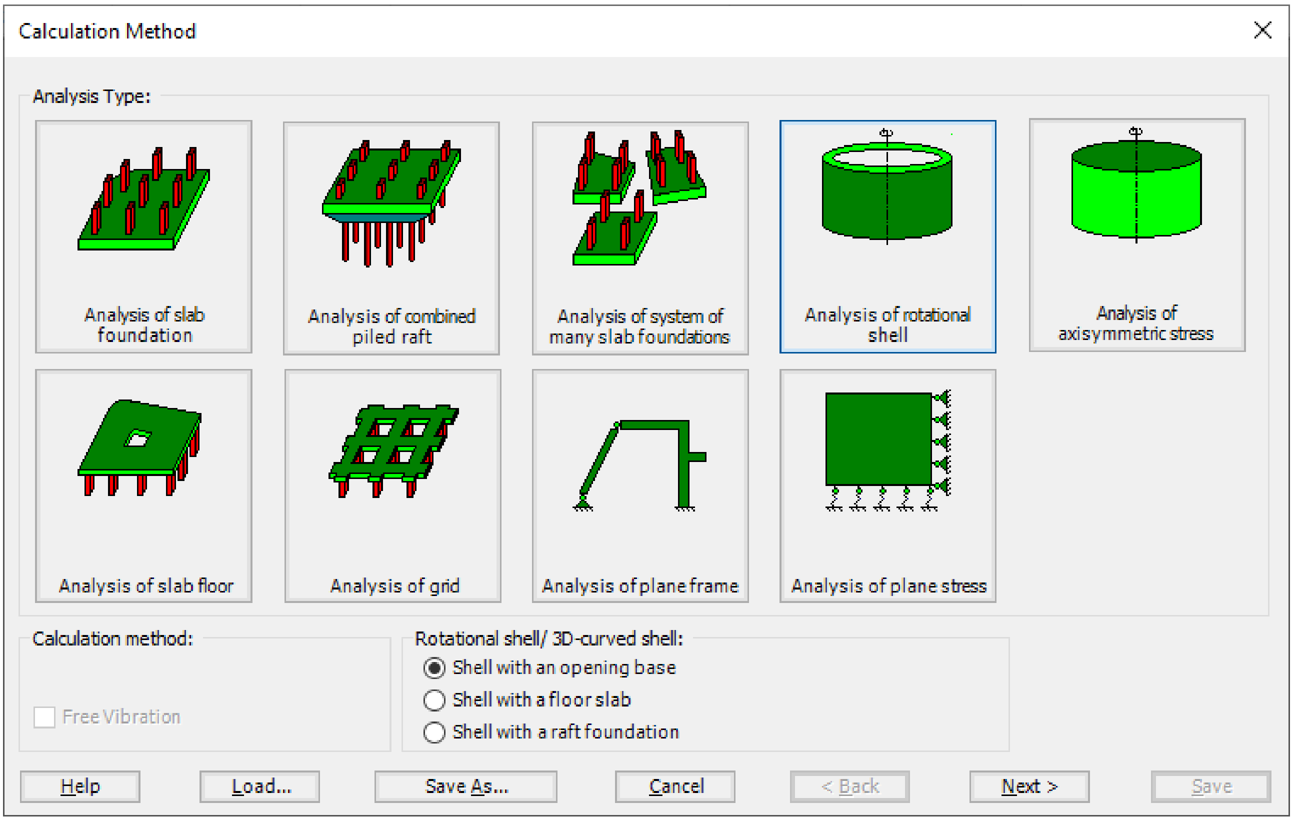


Figure 3.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 3.3, define the analysis type of the problem. As the analysis type is a tank with a fixed base problem, select "Analysis of rotational shell" button, and check "Shell with an opening base" option then click "Next" button to go to the next Form.

The last Form in the wizard is the "Options" Form, Figure 3.4. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Select "Supports/ Boundary Conditions", then click the "Save" button.

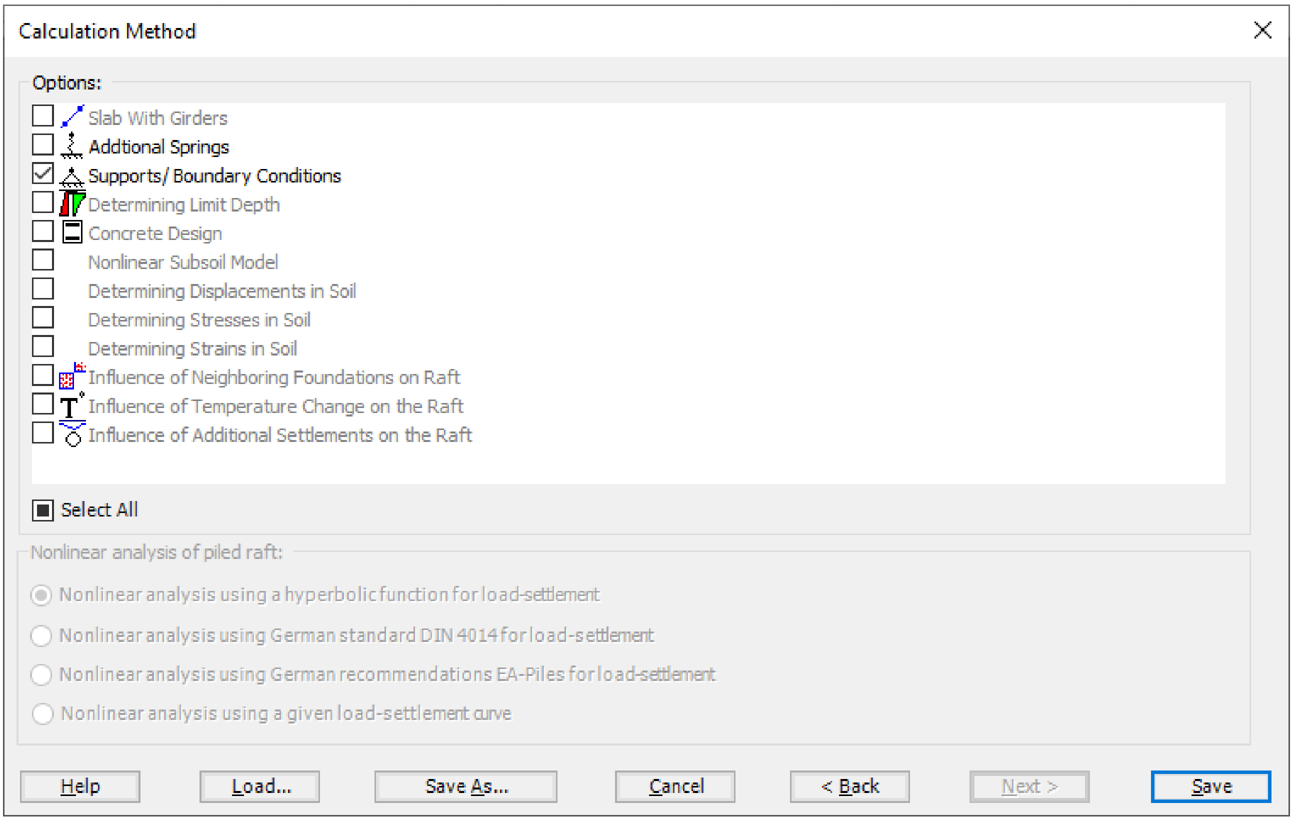


Figure 3.4 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 3.5. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Tank with fixed base". *ELPLA* will use automatically this file name in all reading and writing processes.

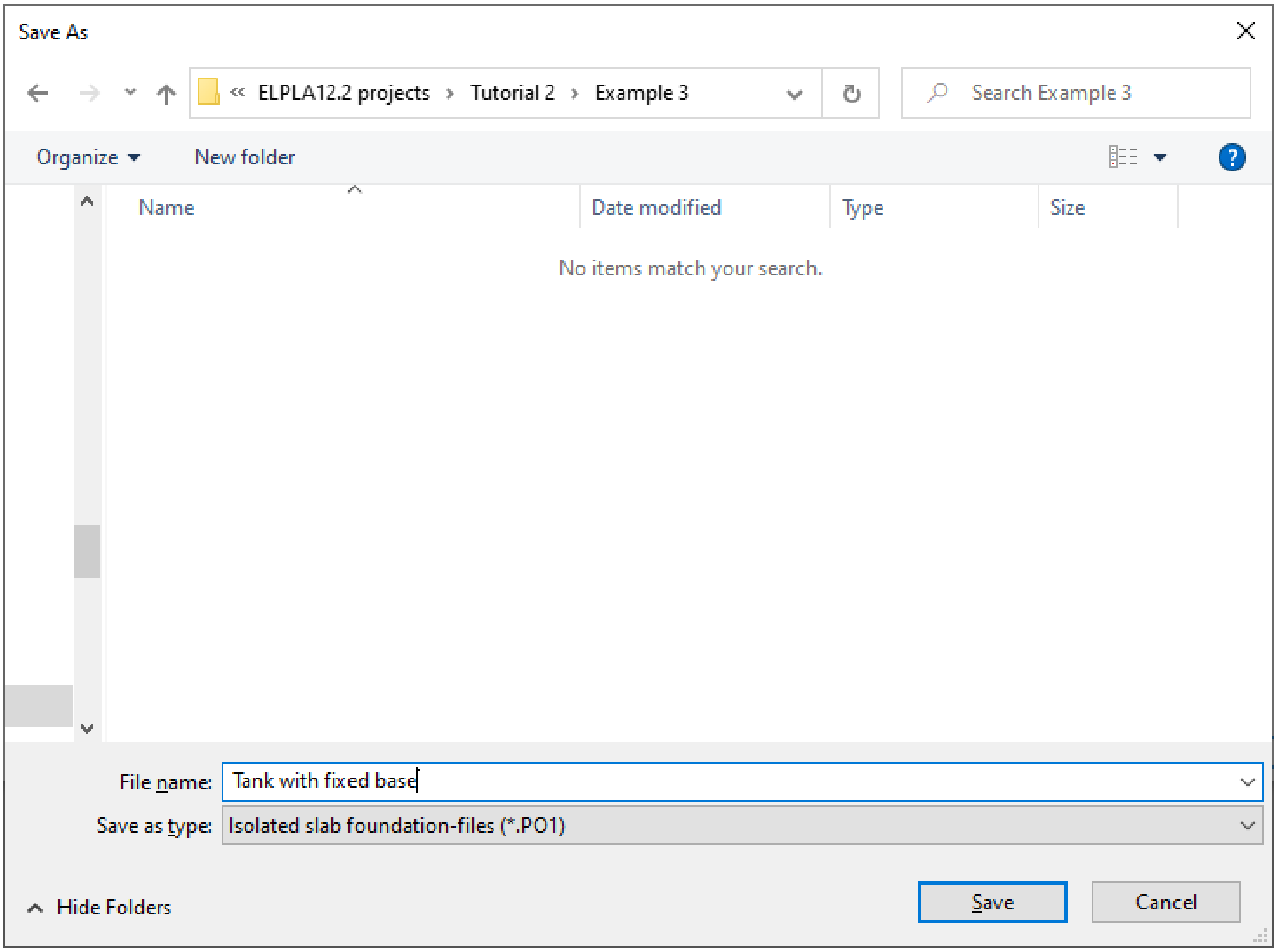


Figure 3.5 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Tank with fixed base] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 3.6 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a tank with a fixed base"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

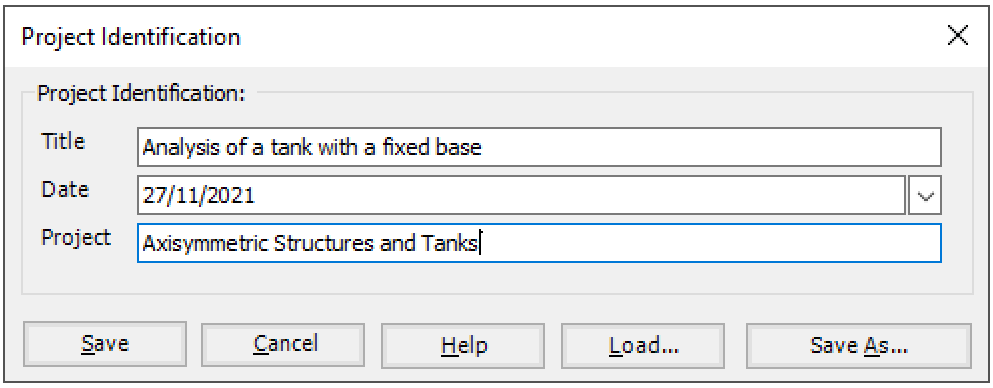


Figure 3.6 "Project Identification" dialog box

4.3 FE-Net data

For the given problem, the tank has a radius of a = 7 [m] and a height of H = 5 [m], the height of the tank is divided into 50 equal segments. Each segment is 10 [cm] size. To define the FE-Net for this tank, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 3.7. This wizard will guide you through the steps required to generate a FE-Net, the first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of shells. These net templates are used to generate standard nets.

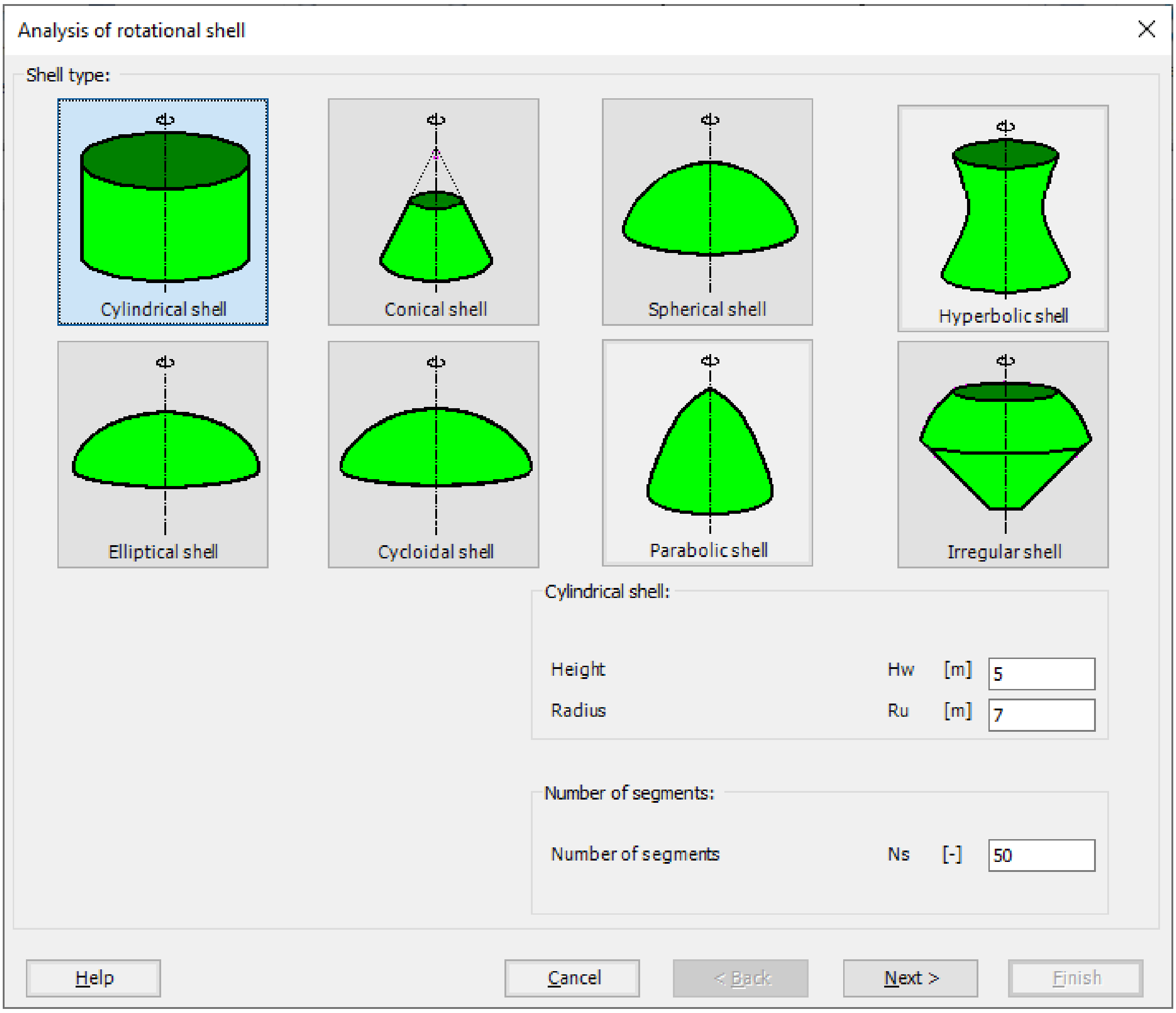


Figure 3.7 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Cylindrical shell" button
- Type 5 in the "Height" edit box,
- Type 7 in the "Radius" edit box,
- Type 50 in the "Number of segments" edit box
- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Cylindrical shell" Form appears, Figure 3.8. *ELPLA* divides the height of the tank into 50 equal segments, the user can edit the data of the segments individually by using "Modify" button, or all of them by using "In Table" button, if it is necessary.

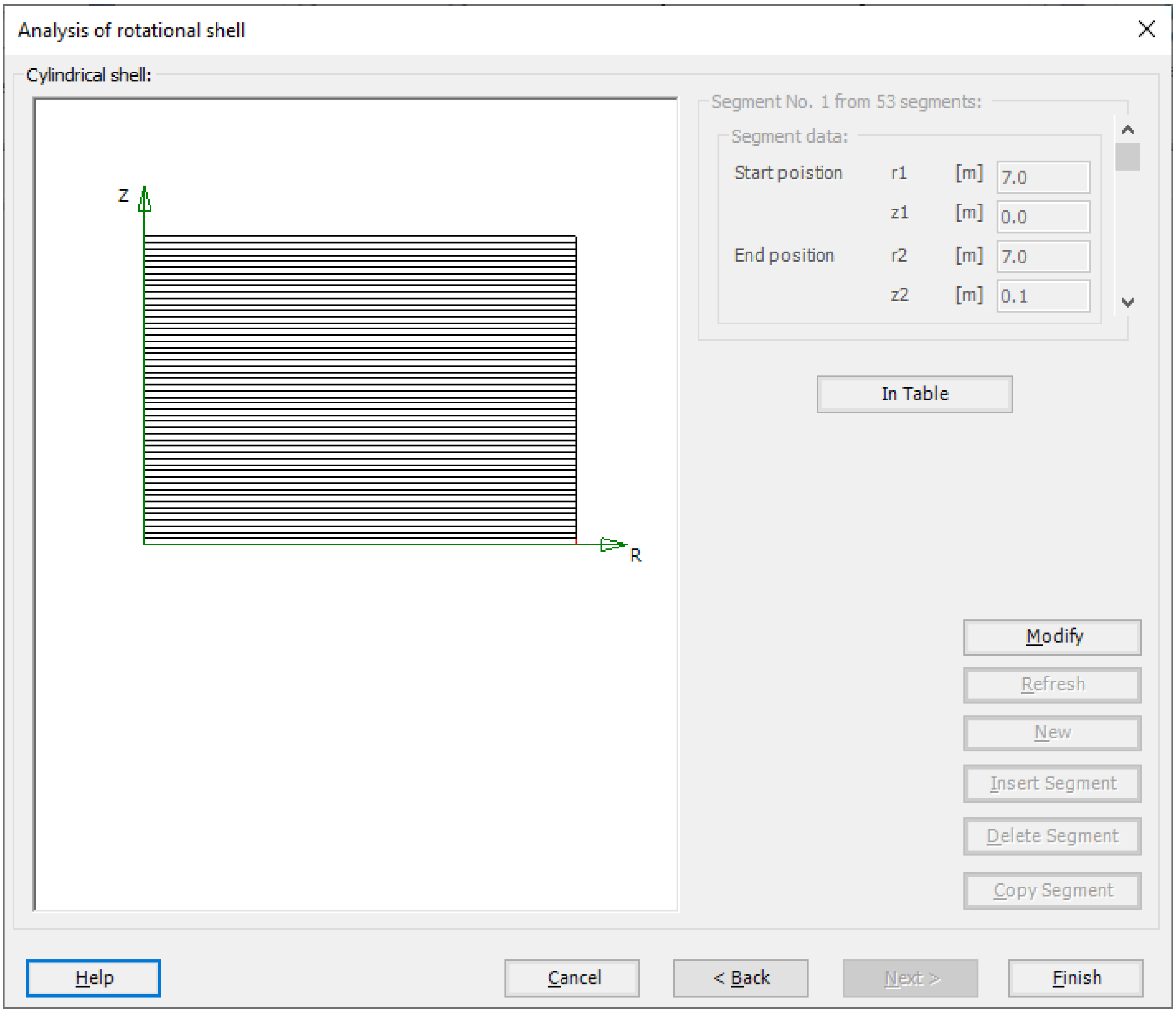


Figure 3.8 "Cylindrical shell" Form

Click "Finish" button, the generated FE-Net appears in Figure 3.9.

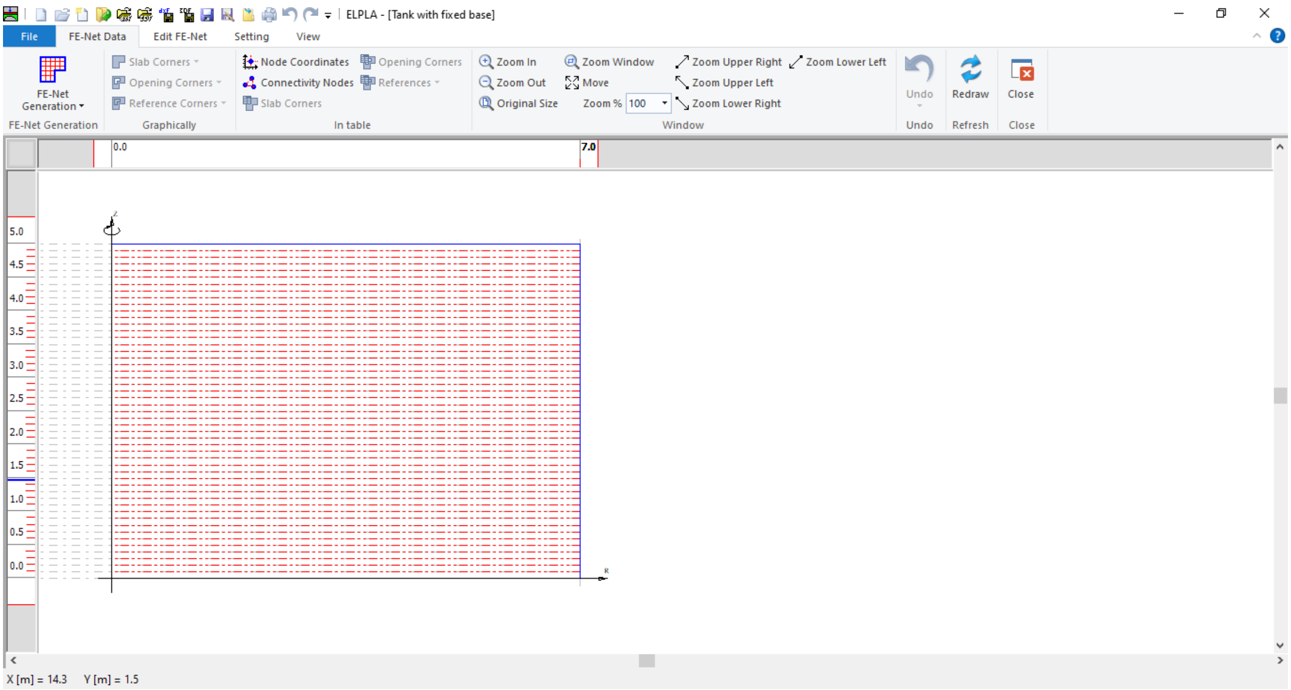


Figure 3.9 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 3.9 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 3.9 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Shell properties

To define the tank properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 3.10 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, unit weight of the shell and the filled material properties.

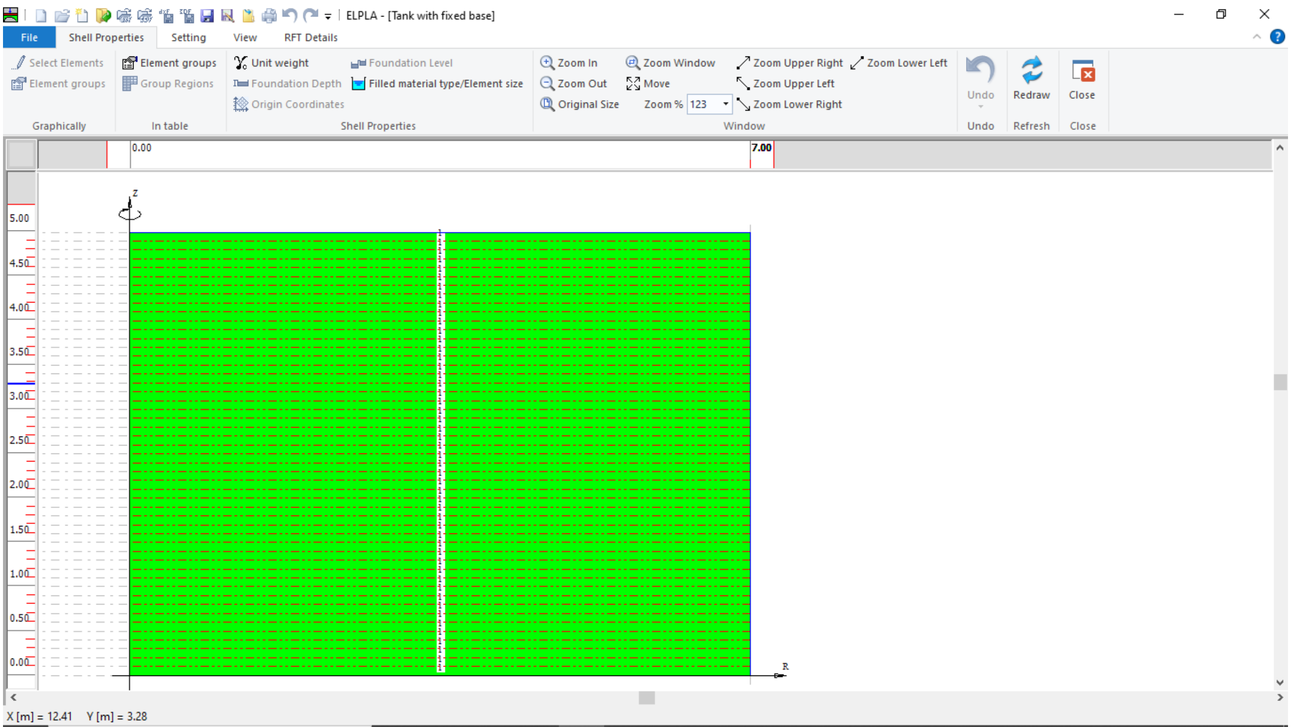


Figure 3.10 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 3.11 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness. Then click "OK" button.

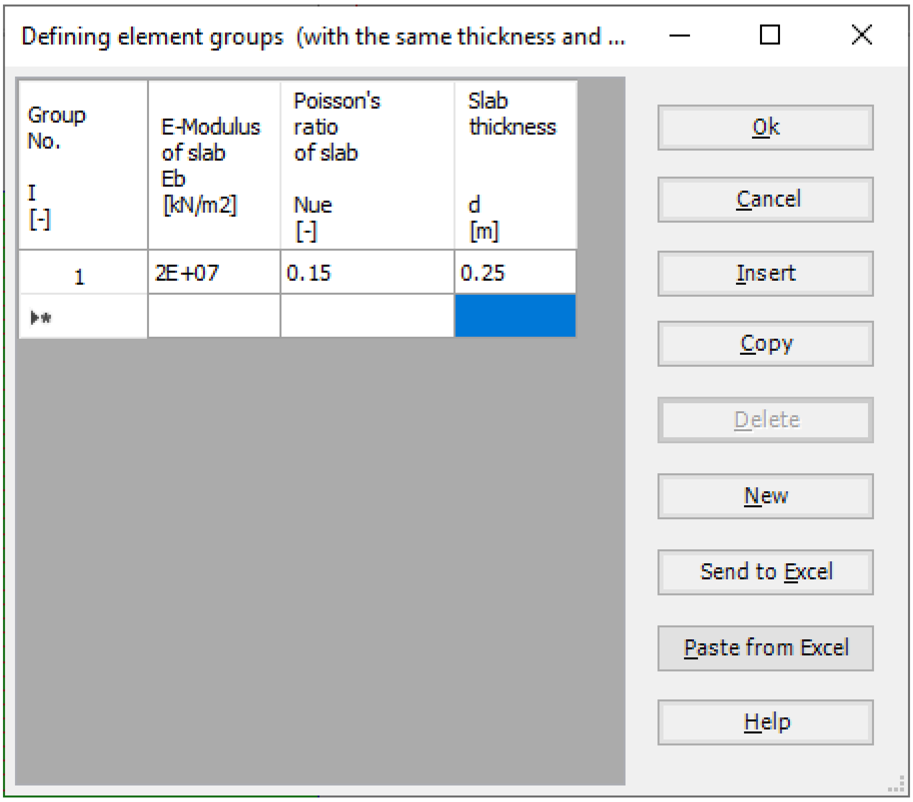


Figure 3.11 "Defining element groups" list box

To enter the unit weight of the shell, choose "Unit weight" command from "Shell Properties" menu in the window of Figure 3.10. The following dialog box in Figure 3.12 with a default unit weight of 25 [kN/m^3] appears, Click "OK" button.

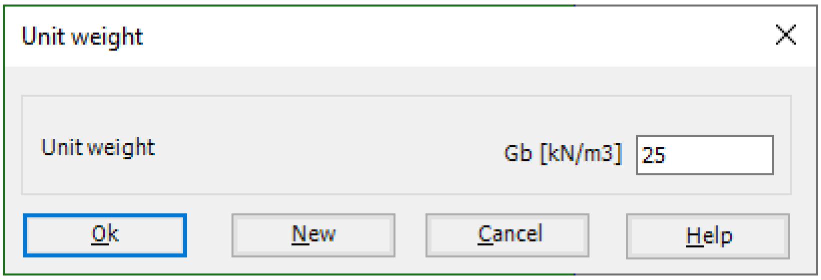


Figure 3.12 "Unit weight" dialog box

To define the liquid properties of the shell, choose "Filled material type/Element size" command from "Shell Properties" menu in the window of Figure 3.10. The following form in Figure 3.13.

To define the filled material type of the tank:

- Select the "Liquid container" check box,
- Type 5 in the "Height of the liquid" edit box,
- Type 10 in the "Unit weight of the liquid" edit box,

To define the element size of the tank:

- Check the "Constant element sizes in z-direction" check box,
- Type 0.2 in the "Element size in each shell segment" edit box,
- Click "OK" button

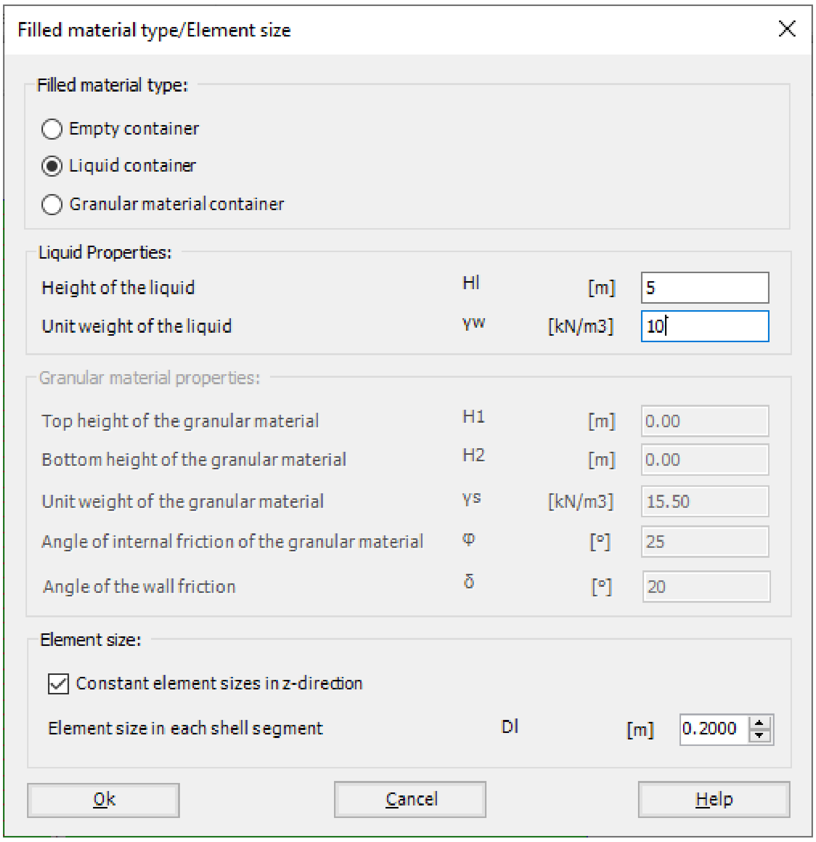


Figure 3.13 "Filled material type/Element size" Form

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

- Choose "Save" command from "File" menu in Figure 3.10 to save the shell properties
- Choose "Close" command from "File" menu in Figure 3.10 to close the "Shell properties" window and return to *ELPLA* main window

4.5 Supports/ boundary conditions

To define the fixed support, choose "Supports/ Boundary Conditions" command from “Data” Tab. The following window in Figure 3.14 appears.

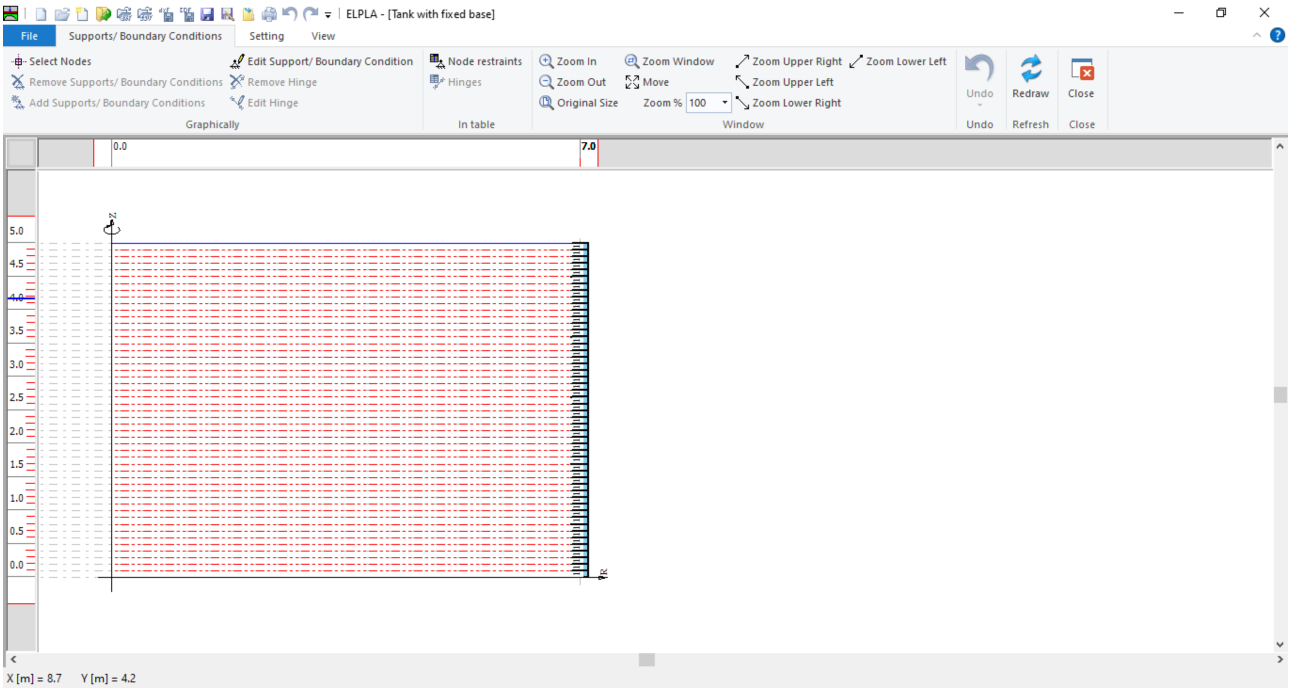


Figure 3.14 "Supports/ Boundary Conditions" Window

To define supports on the net:

- Choose "Select Nodes" command from "Graphically" menu in Figure 3.14. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the node that have the fixed support as shown in Figure 3.15
- After selecting the node, choose "Add Supports/ Boundary Conditions" command from "Graphically" menu (Figure 3.14). The "Supports/ Boundary Conditions" dialog box in Figure 3.16 appears.

-

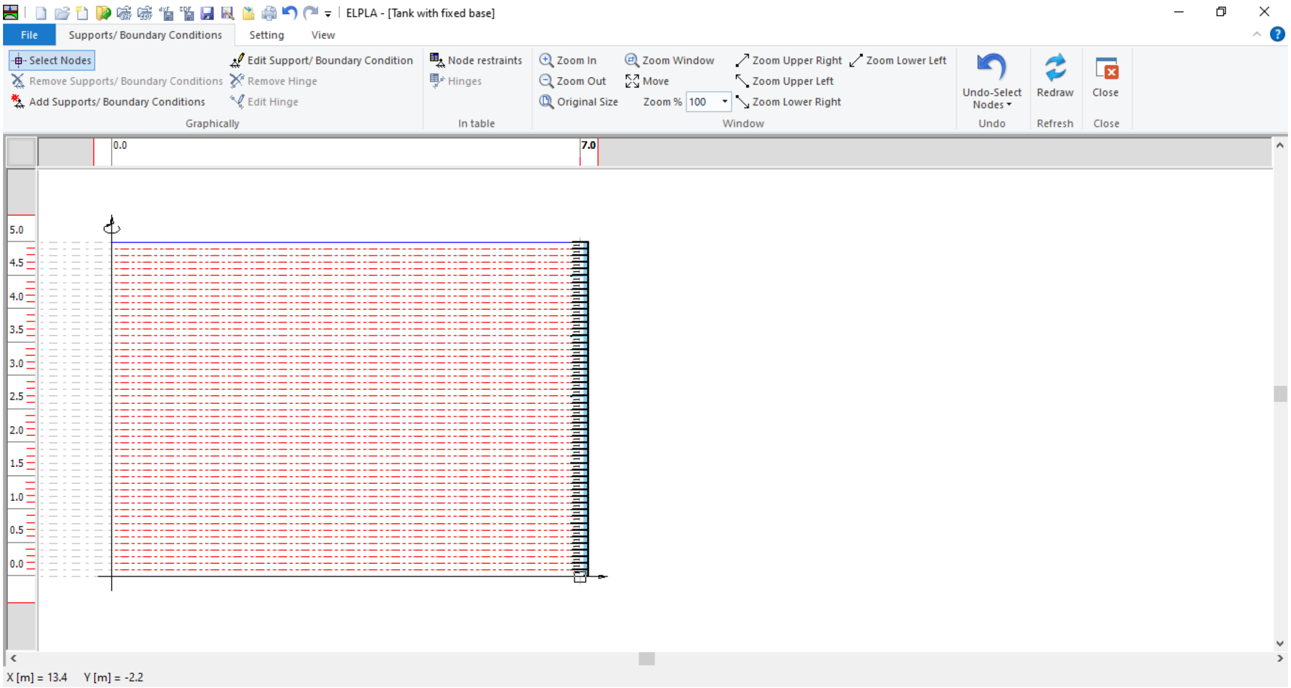


Figure 3.15 Selection of node that has a fixed support

In this dialog box

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

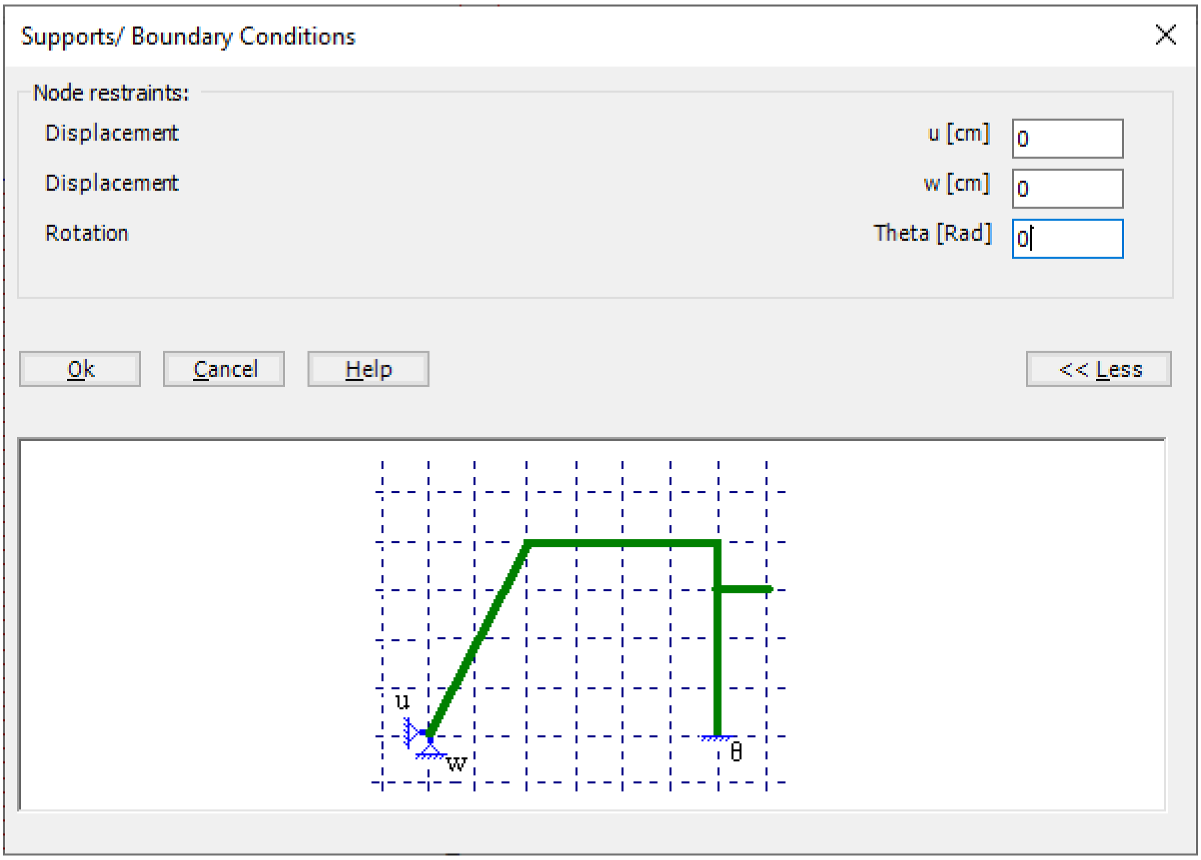


Figure 3.16 "Supports/ Boundary Conditions" dialog box

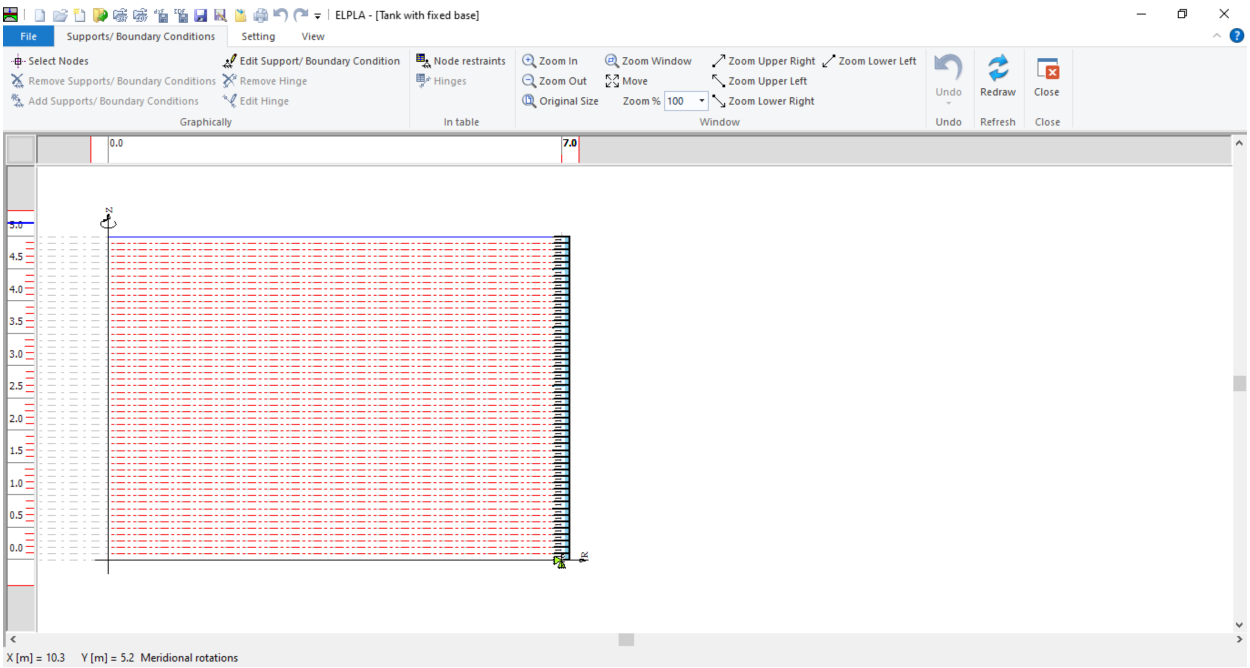


Figure 3.17 Supports on the screen

After defining the supports, do the following two steps

- Choose "Save " command from "File" menu in Figure 3.17 to save the data of supports
- Choose "Close" command from "File" menu in Figure 3.17 to close the "Supports/ Boundary conditions" window and return to the main window

4.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 3.18 appears. In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 3.18. In this example, there is not applied load, as the vertical load has been already defined by the unit weight of the tank material, while the hydrostatic pressure on the tank is defined by the unit weight of water.

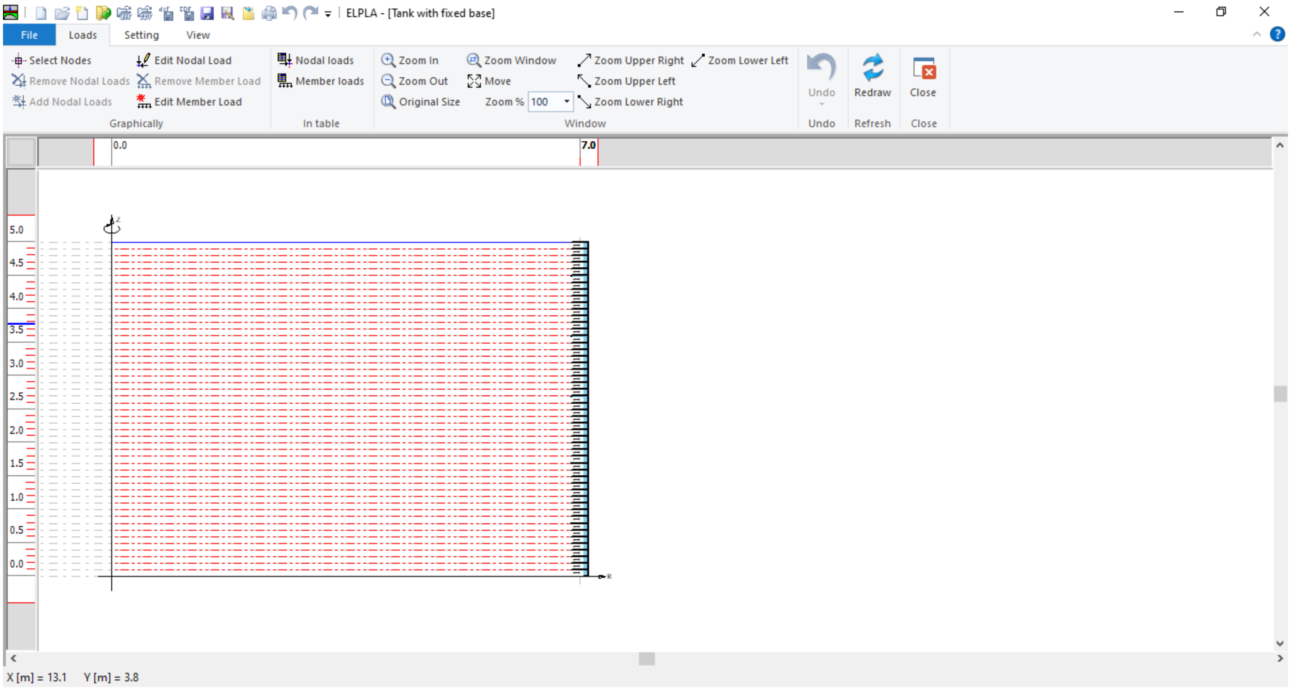


Figure 3.18 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 3.18 to save the load data
- Choose "Close" command from "File" menu in Figure 3.18 to close the "Loads" window and return to *ELPLA* main window

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

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 3.19.

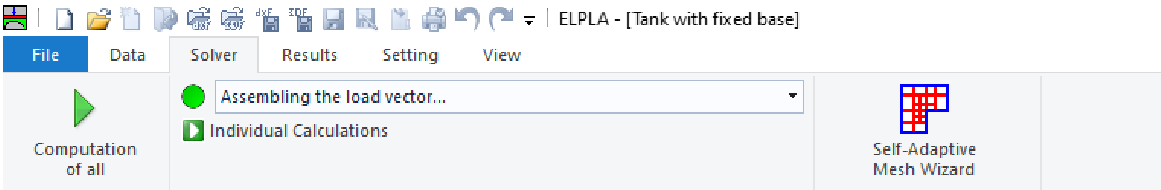


Figure 3.19 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

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

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

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab Window.

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

Analysis progress

Analysis progress menu in Figure 3.20 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

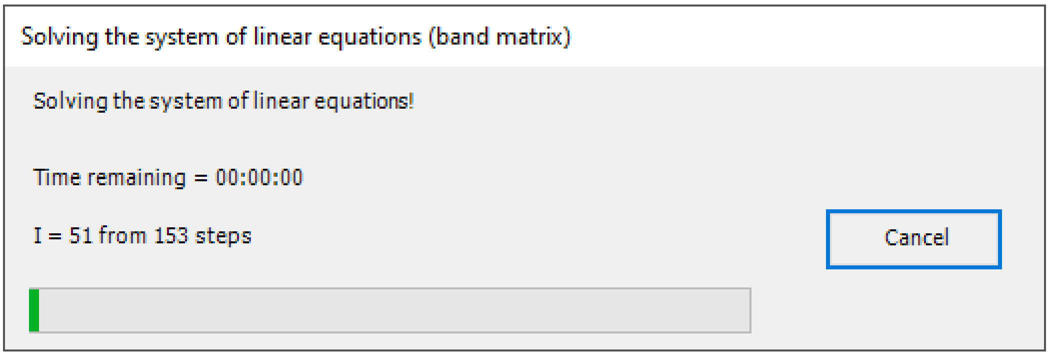


Figure 3.20 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 3.21. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

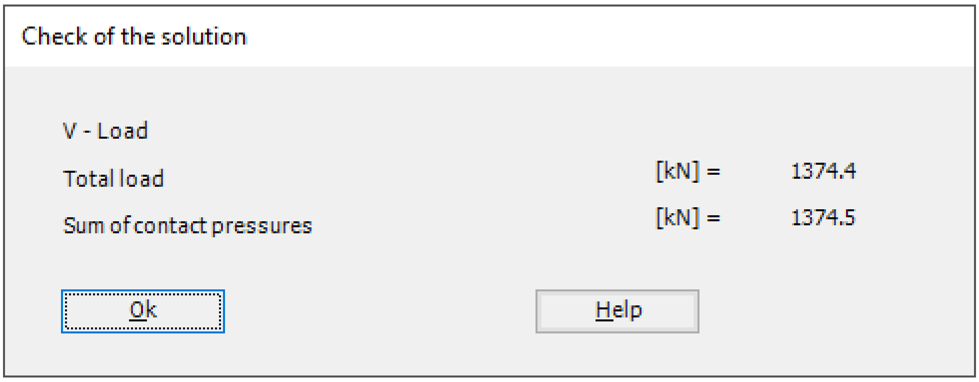


Figure 3.21 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab.

To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 3.22).

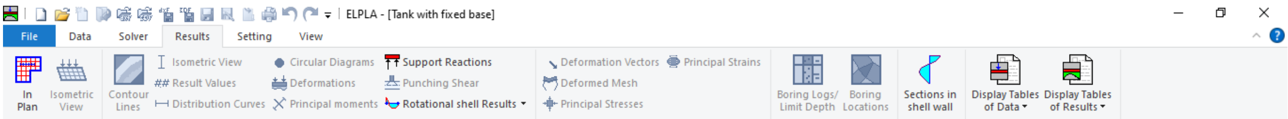


Figure 3.22 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Support Reactions
- Sections in shell wall
- Display tables of data
- Display tables of results

To view the meridional moments in the shell wall

- Choose "Sections in shell wall" command from "Section" menu. The following option box in Figure 3.23 appears
- In the "Sections in shell wall" option box, select "Meridional moments *My*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 3.24.

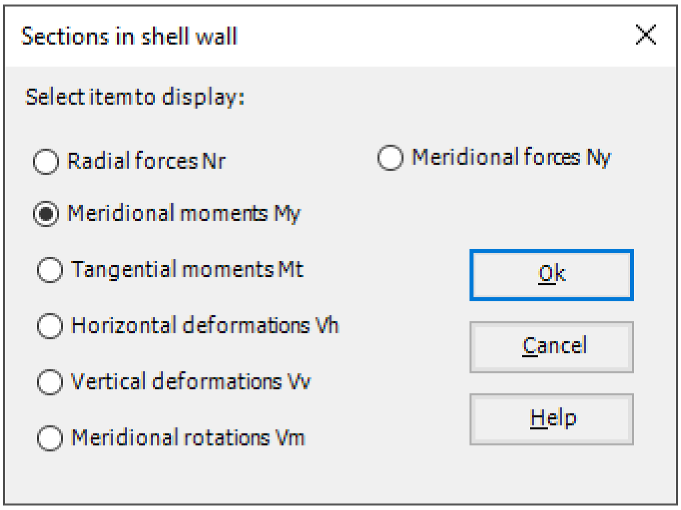


Figure 3.23 "Sections in shell wall" option box

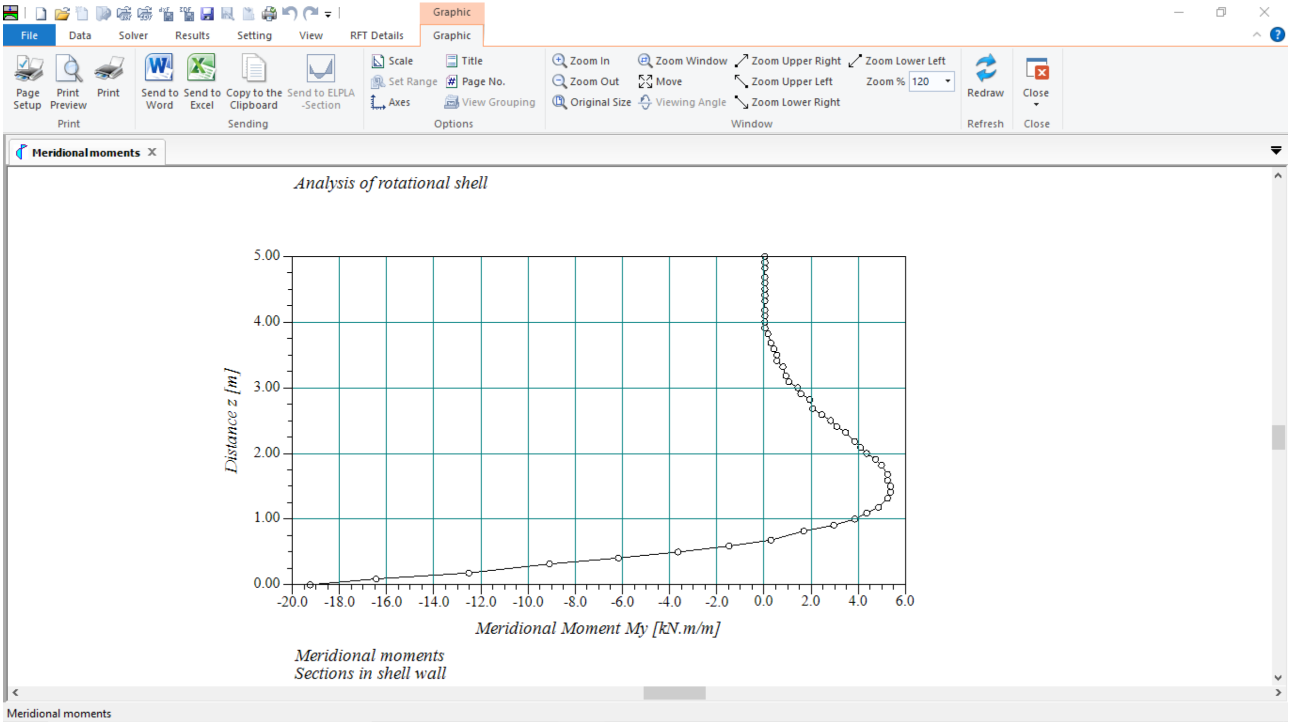


Figure 3.24 Meridional moments sections in shell wall

To view the radial forces on the shell wall

- From "Rotational shell results" command in the "Results" menu, choose "In Plan" command, the following option box in Figure 3.25 appears
- In the "Distribution of Internal Forces" option box, select "Radial forces Nr" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 3.26.

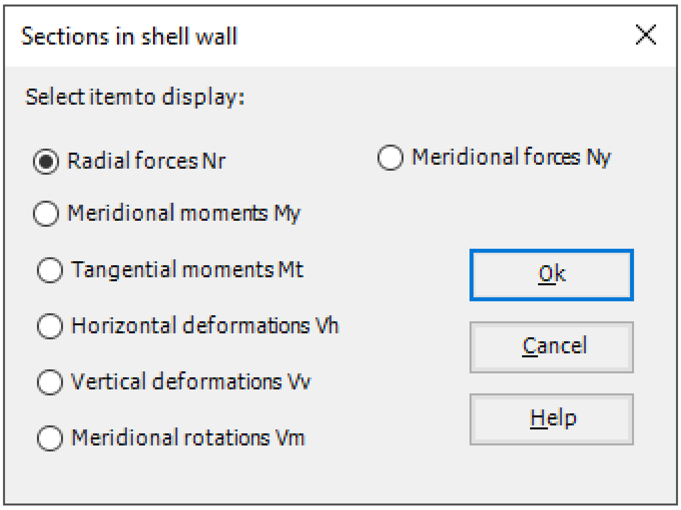


Figure 3.25 "Distribution of Internal Forces" option box

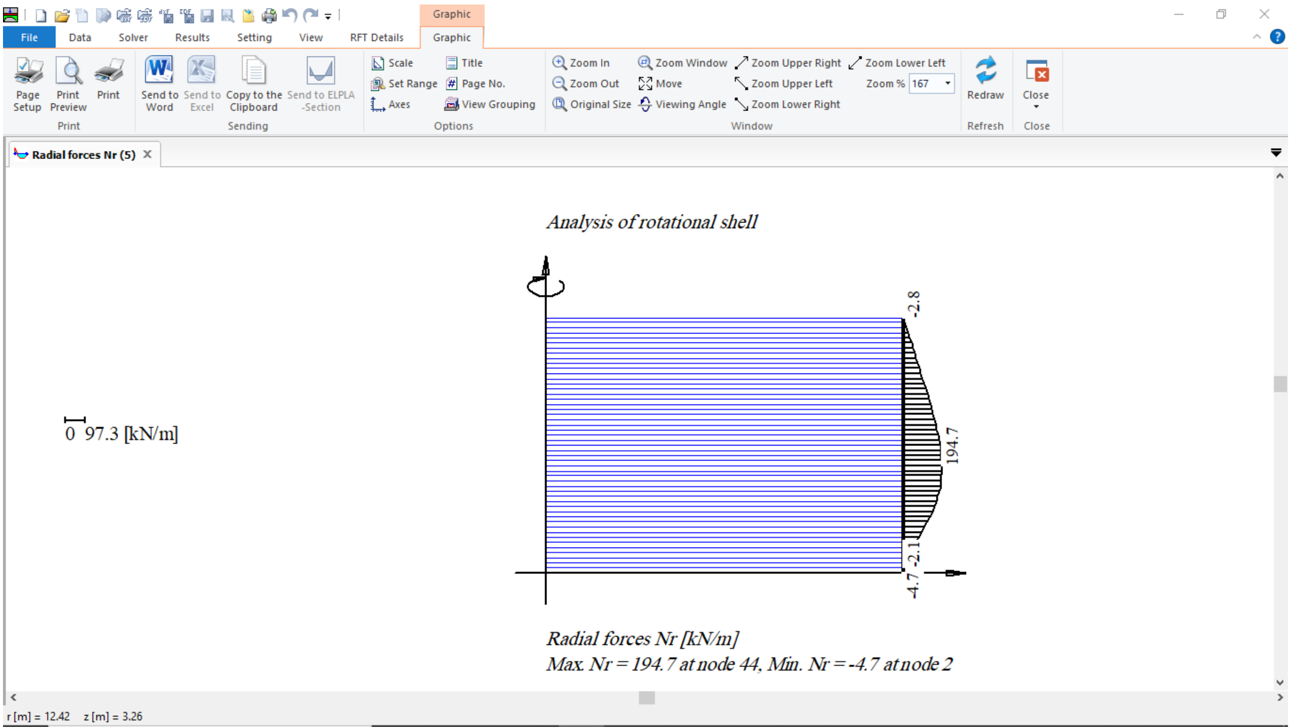


Figure 3.26 Radial forces on the shell wall

To view element groups of the tank

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 3.27 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

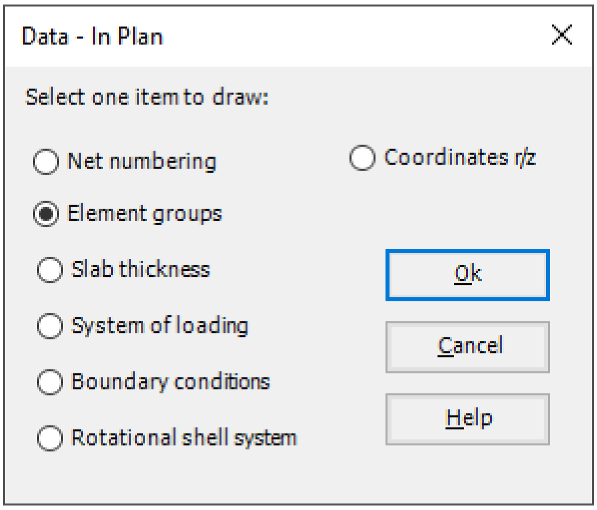


Figure 3.27 "Data – In Plan" option box

To view the supports / boundary conditions on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 3.28 appears
- In this check group box, check "Supports Reactions *RV*" check box
- The user can choose any other data to be displayed
- Click "OK" button

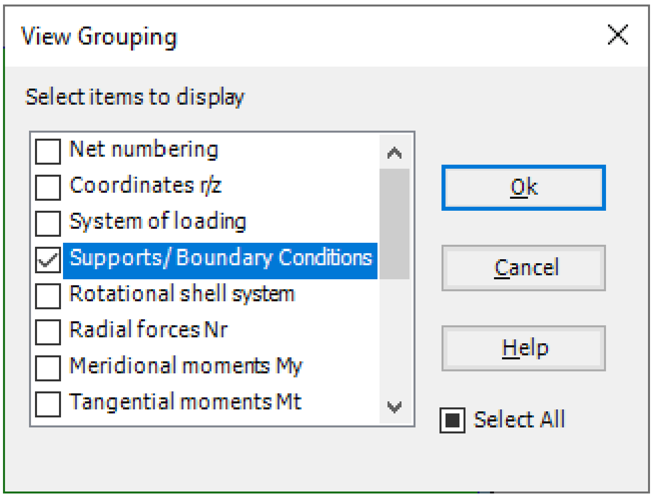


Figure 3.28 "View Grouping" check group box

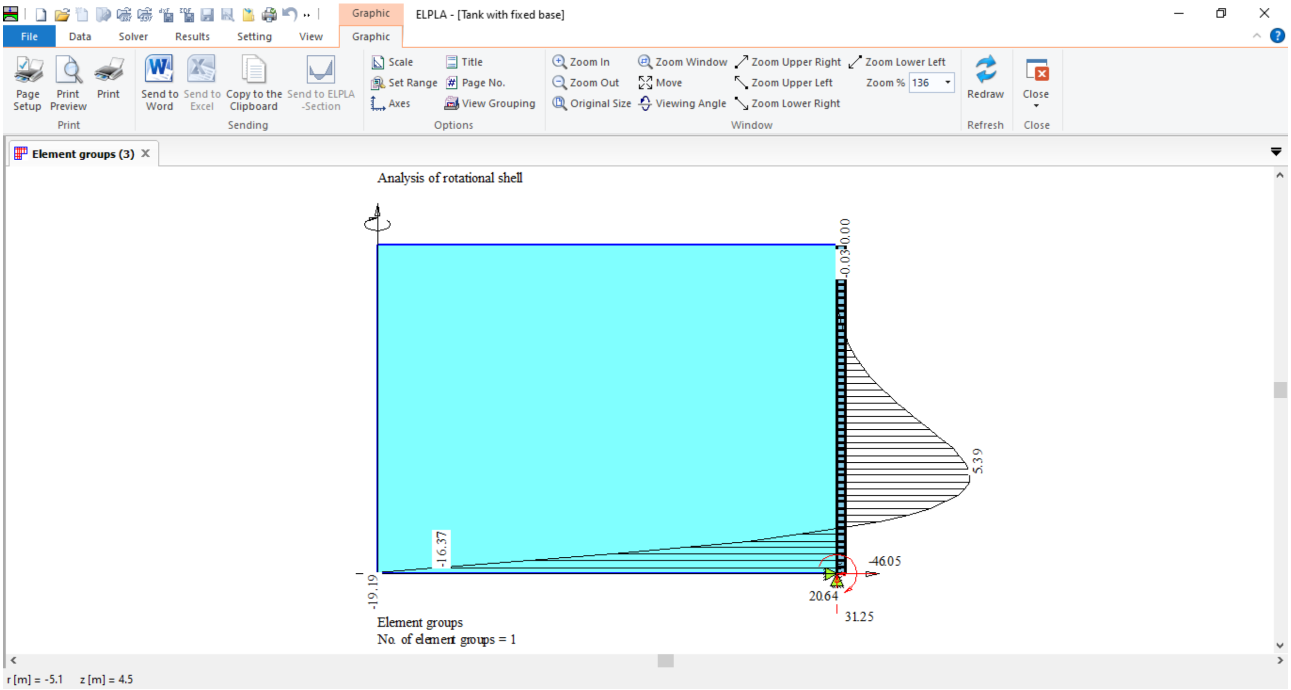


Figure 3.29 Element groups

Example 4

Analysis of a reservoir wall with a variable wall thickness

Contents Page

1 Description of the problem

An example of a reservoir with a variable wall thickness is selected to illustrate some features of *ELPLA* for analyzing shell elements.

2 Geometry and properties

A reservoir wall of a radius a = 100 [m] and a height H = 100.1 [m] is considered as shown in Figure 4.1. The wall of the reservoir has a variable thickness, at the base the thickness is h_{11} = 13.3 [m], while at the top the thickness is h_0 = 4 [m], thickness in between h [m] can be obtained from the following equation:

$$h = 4e^{\frac{1.2x}{100}}$$

where x is the distance from the top in [m].

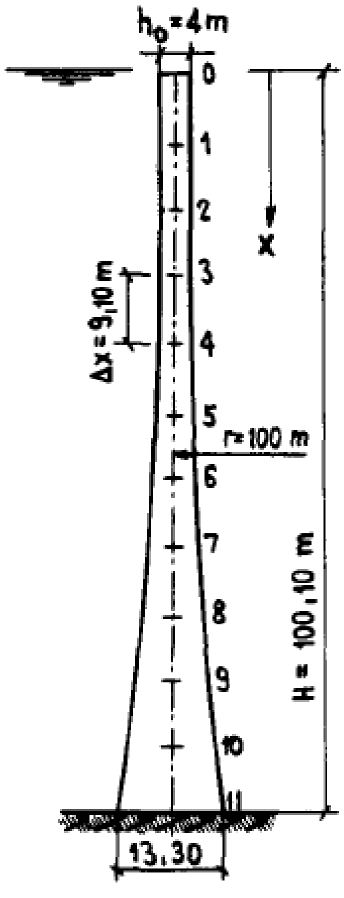


Figure 4.1 reservoir wall with dimensions

The reservoir wall is exposed to a hydrostatic water pressure and is fixed at the base. The wall material and unit weight of the water are listed in Table 4.1.

Table 4.1 Wall material and water unit weight

Modulus of Elasticity of the reservoir wall material	E_c	$= 2.1\times10^7$	[kN/m^2]
Poisson's ratio of the reservoir wall material	ν_c	= 0	[-]
Unit weight of the water	γ_w	= 10	[kN/m^3]

3 Analysis of the reservoir wall

In the analysis, the total height of the wall is divided into 11 segments with a constant length; each is (Figure 4.2):

$$\Delta x = \frac{100.10}{11} = 9.10 \ [m]$$

Figure 4.2 Finite element mesh of the reservoir wall with wall thickness

4 Creating the project

In this section, the user will learn how to create a project for analyzing cylindrical shells with variable wall thickness. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 4.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of Forms. The first Form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 4.3).

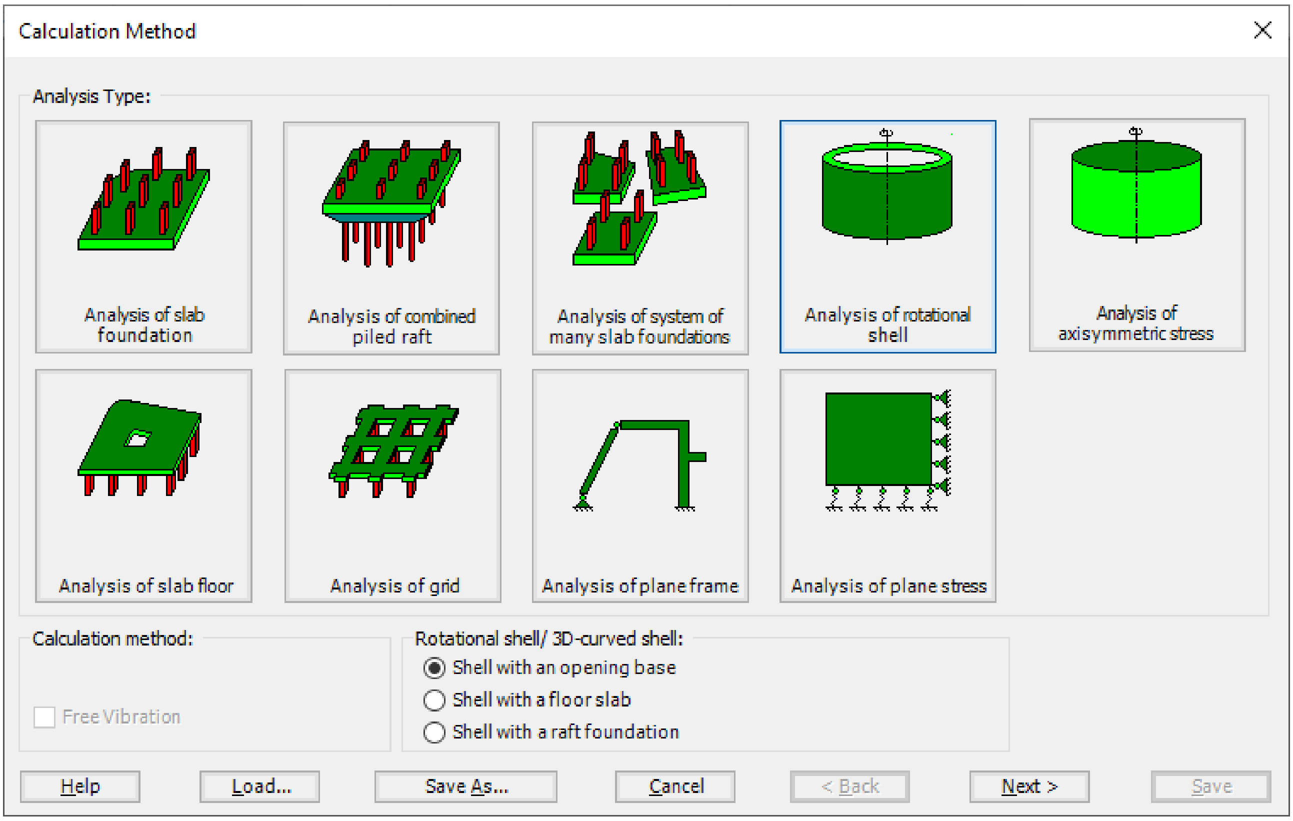


Figure 4.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 4.3, define the analysis type of the problem. As the analysis type is a cylindrical shell with a variable wall thickness problem, select "Analysis of rotational Shell" button, and check "Shell with an opening base" option, then click "Next" button to go to the next Form.

The last Form in the wizard is the "Options" Form, Figure 4.4. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Select "Supports/ Boundary Conditions", then click the "Save" button.

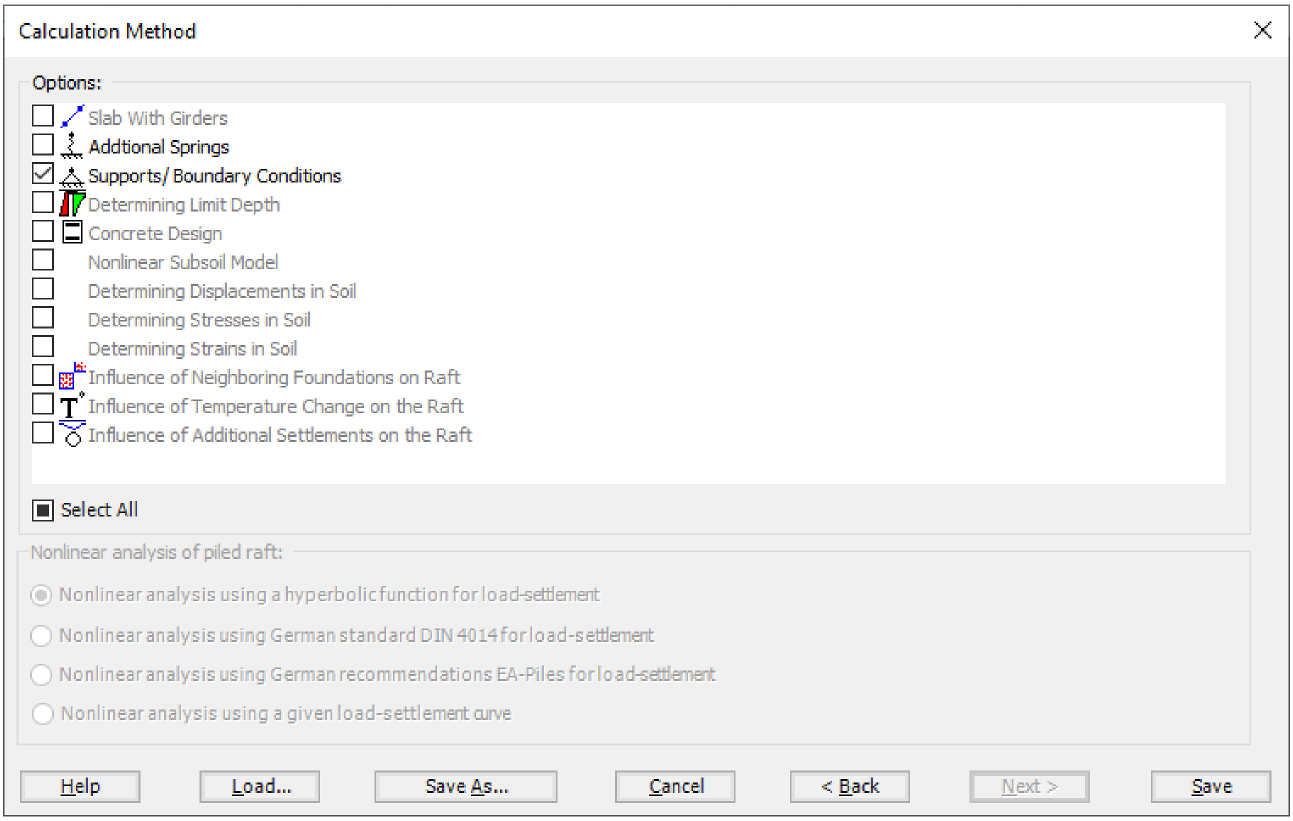


Figure 4.4 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 4.5. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Reservoir wall". *ELPLA* will use automatically this file name in all reading and writing processes.

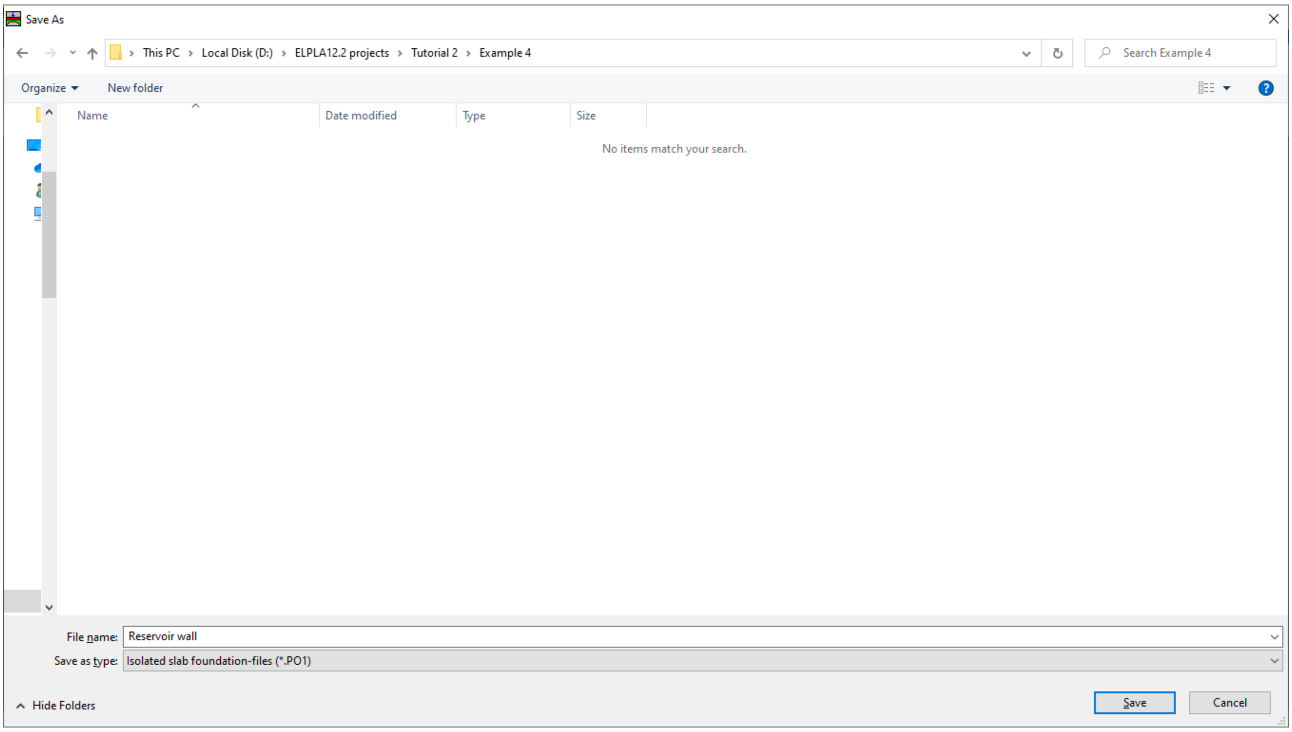


Figure 4.5 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Reservoir wall] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 4.6 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a reservoir wall with a variable wall thickness"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

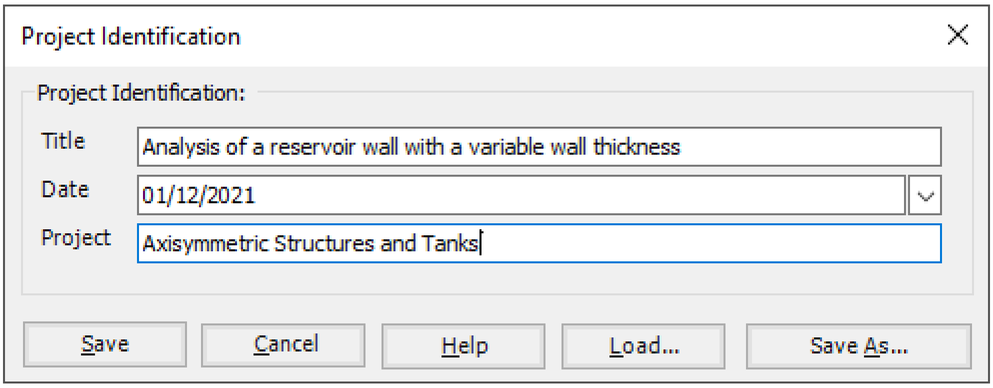


Figure 4.6 "Project Identification" dialog box

4.3 FE-Net data

A reservoir wall of a radius a = 100 [m] and a height H = 100.1 [m] is considered as shown in Figure 4.1. The wall of the reservoir has a variable thickness, at the base the thickness is h_{11} = 13.3 [m], while at the top the thickness is h_0 = 4 [m], the total height of the wall is divided into 11 segments with a constant length, each 9.10 [m]. To define the FE-Net for this shell, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 4.7. This wizard will guide you through the steps required to generate a FE-Net.

The first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

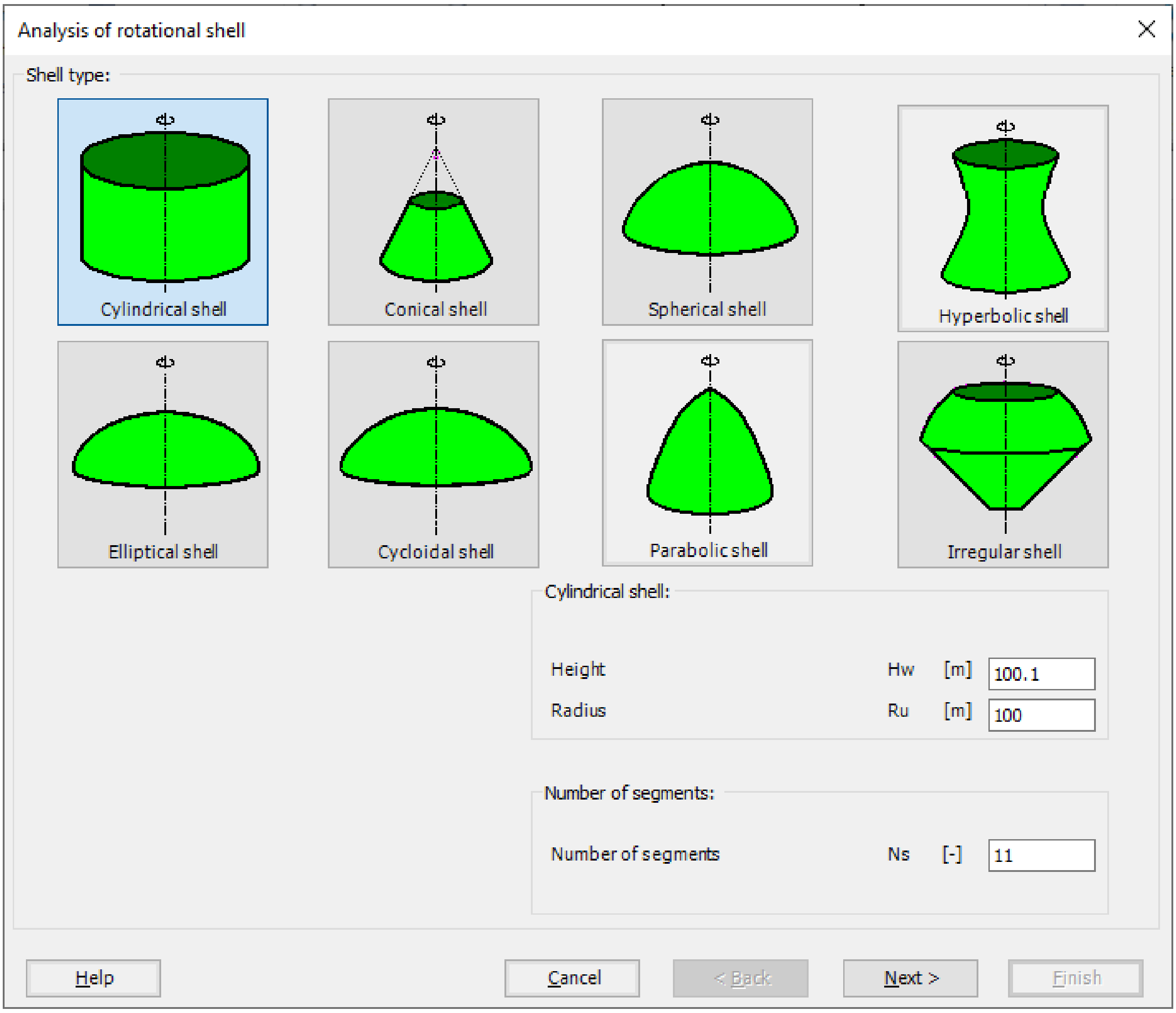


Figure 4.7 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Cylindrical shell" button
- Type 100.1 in the "Height" edit box,
- Type 100 in the "Radius" edit box,
- Type 11 in the "Number of segments" edit box
- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Cylindrical shell" Form appears, Figure 4.8. *ELPLA* divides the height of the reservoir wall into 11 equal segments, the user can edit the data of the segments individually by using "Modify" button, or all of them by using "In Table" button, if it is necessary.

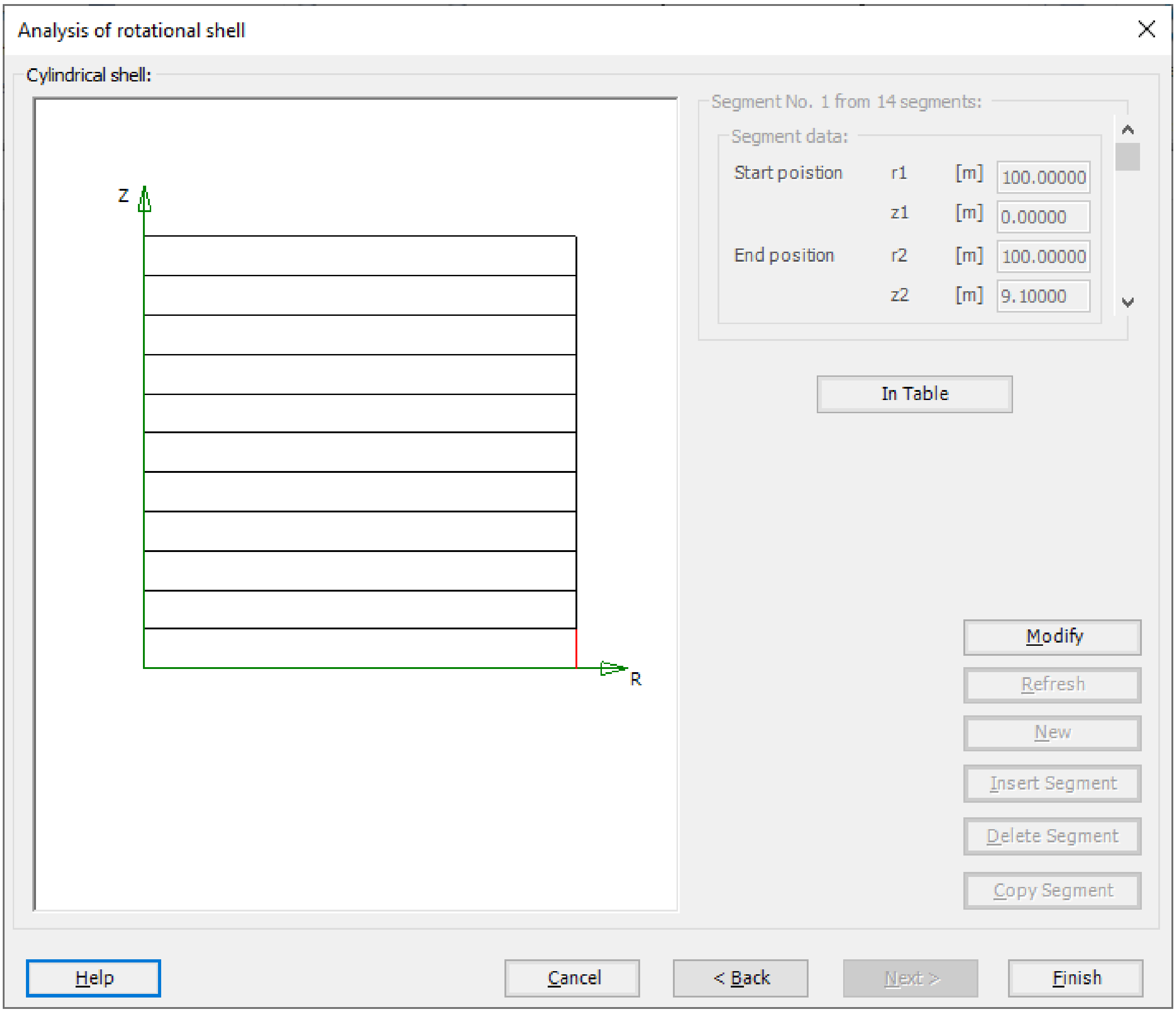


Figure 4.8 "Cylindrical shell" Form

Click "Finish" button, the FE-Net appears in Figure 4.9

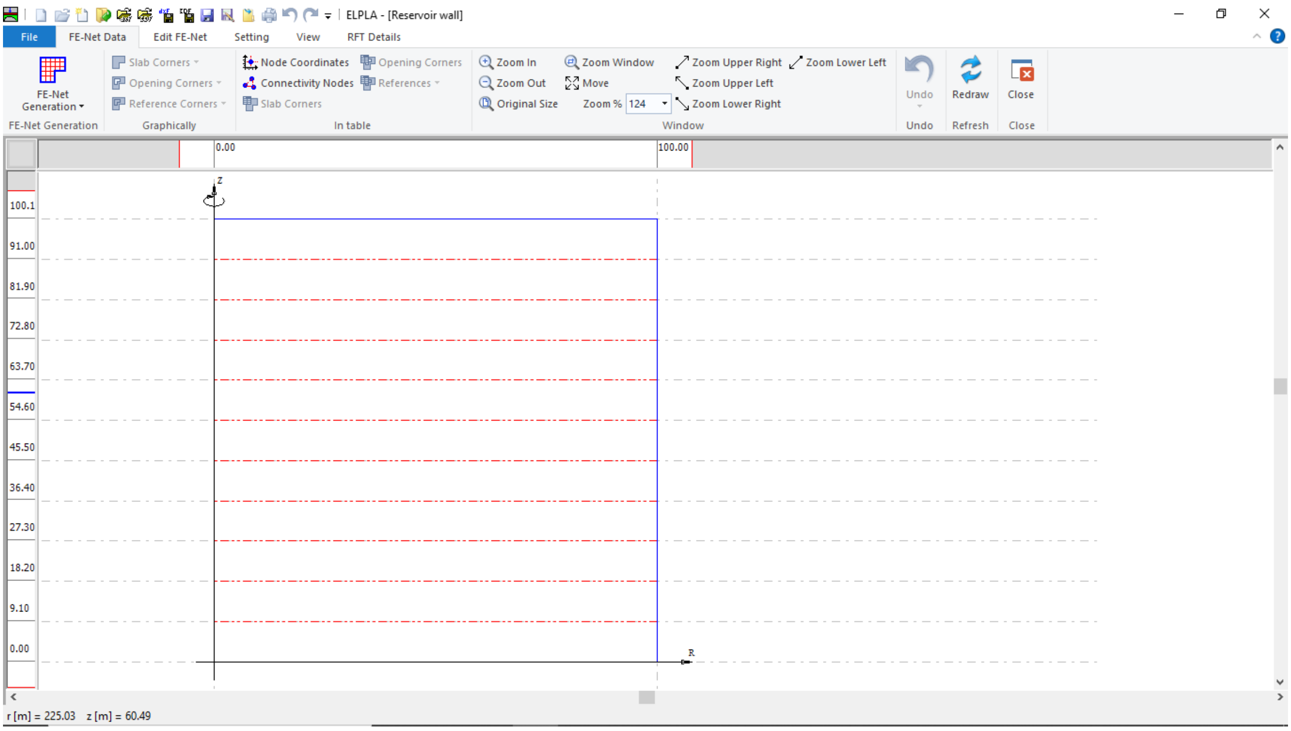


Figure 4.9 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 4.9 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 4.9 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Shell properties

To define the reservoir properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 4.10 appears with default shell properties. The data of reservoir properties for the current example, which are required to be defined, are element groups, group regions, unit weight of the reservoir and liquid properties.

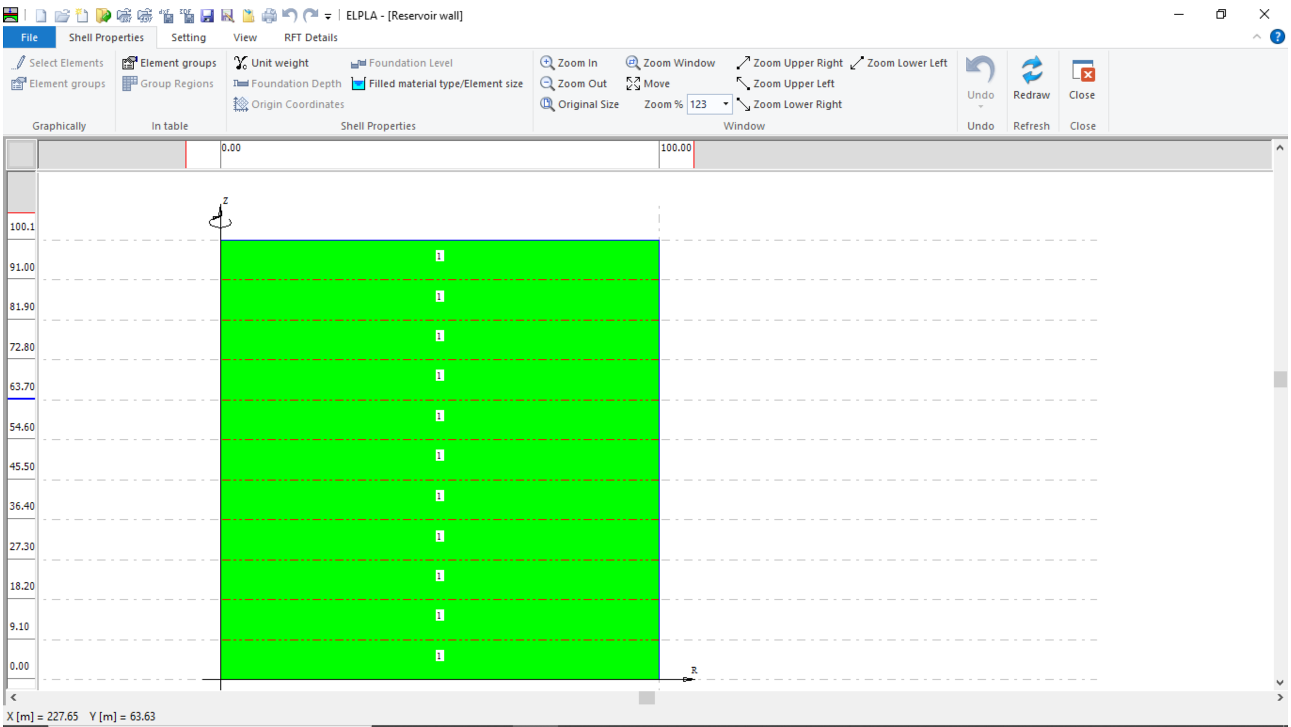
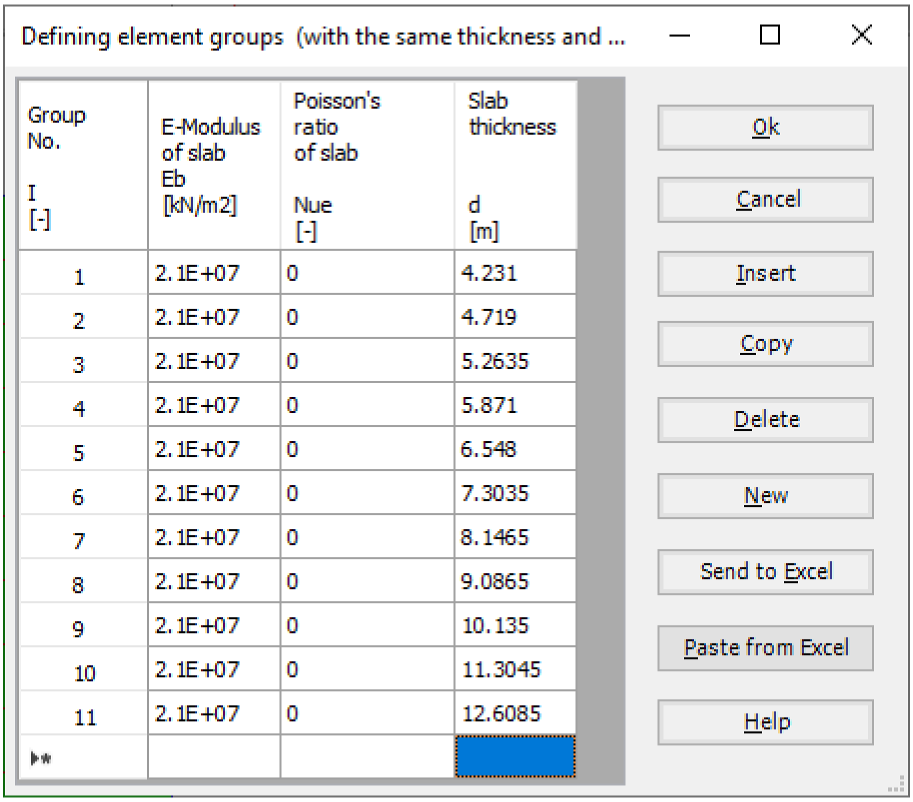


Figure 4.10 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 4.11 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thicknesses of the segments as the following list. Then click "OK" button.



Group No. I [-]	E-Modulus of slab Eb [kN/m2]	Poisson's ratio of slab Nue [-]	Slab thickness d [m]
1	2.1E+07	0	4.231
2	2.1E+07	0	4.719
3	2.1E+07	0	5.2635
4	2.1E+07	0	5.871
5	2.1E+07	0	6.548
6	2.1E+07	0	7.3035
7	2.1E+07	0	8.1465
8	2.1E+07	0	9.0865
9	2.1E+07	0	10.135
10	2.1E+07	0	11.3045
11	2.1E+07	0	12.6085
*			

Figure 4.11 "Defining element groups" list box

Choose "Group Regions " command from "In table" menu. The following list box in Figure 4.12 appears. In this list box, edit the "Group No." value for each segment. Then click "OK" button.

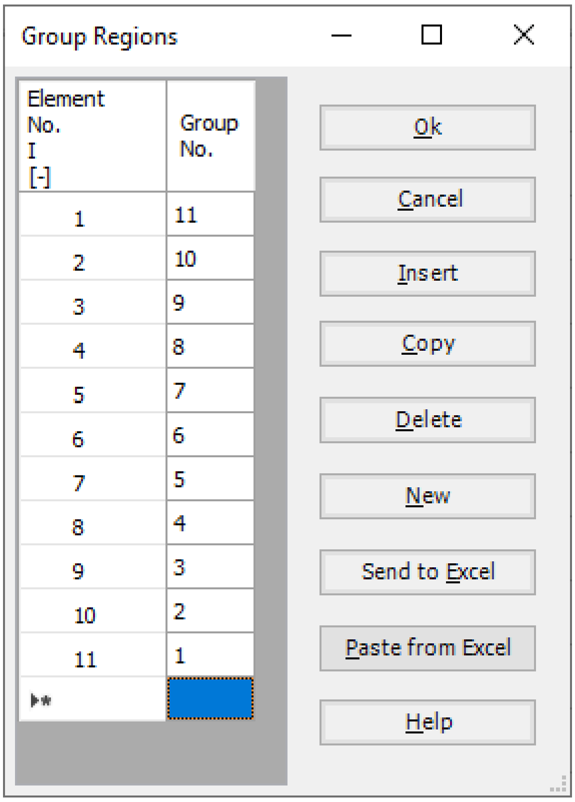


Figure 4.12 "Group Regions" list box

To enter the unit weight of the reservoir, choose "Unit weight" command from "Shell Properties" menu in the window of Figure 4.10. The following dialog box in Figure 4.13 with a default unit weight of 25 [kN/m^3] appears, type 0 in the "Unit weight" edit box to neglect the wall weight in the analysis, then click "OK" button.

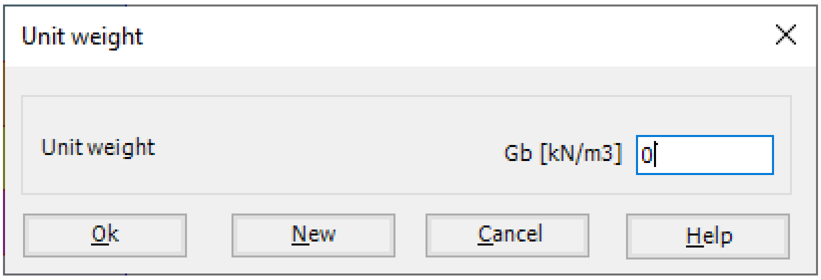


Figure 4.13 "Unit weight" dialog box

To define the liquid properties of the reservoir, choose "Filled material type/Element size" command from "Shell Properties" menu in the window of Figure 4.10. The following form in Figure 4.14.

To define the filled material properties of the reservoir wall:

- Select the "Liquid container" check box,
- Type 100.1 in the "Height of the liquid" edit box,
- Type 9.81 in the "Unit weight of the liquid" edit box,

To define the element size of the ring wall:

- Check the "Constant element sizes in z-direction" check box,
- Type 9.1 in the "Element size in each shell segment" edit box,
- Click "OK" button

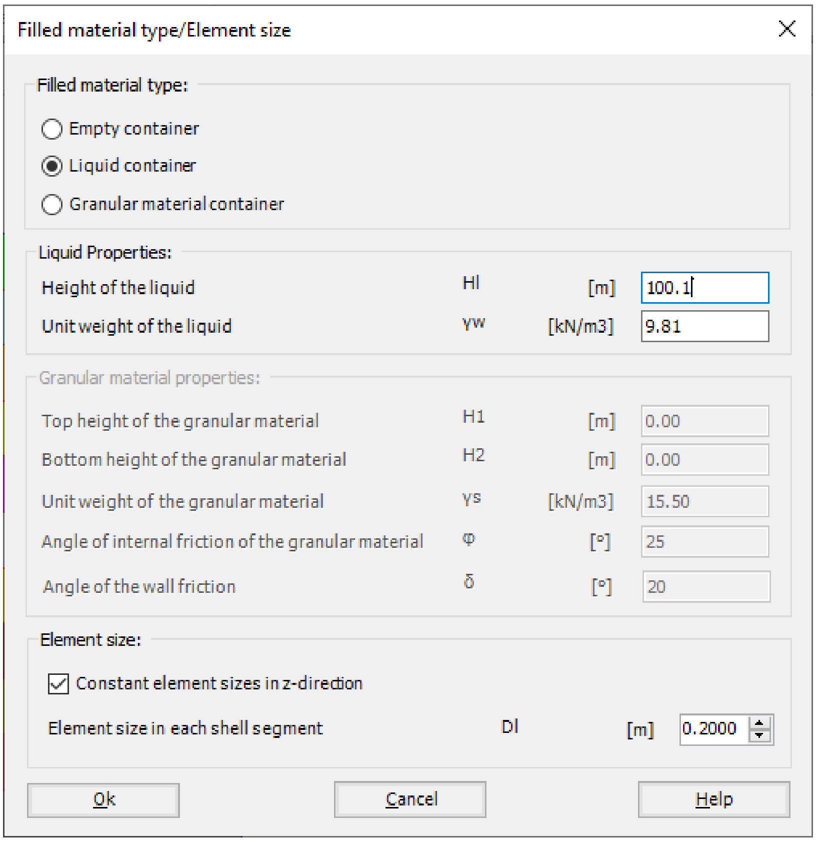


Figure 4.14 "Liquid properties/Element size" Form

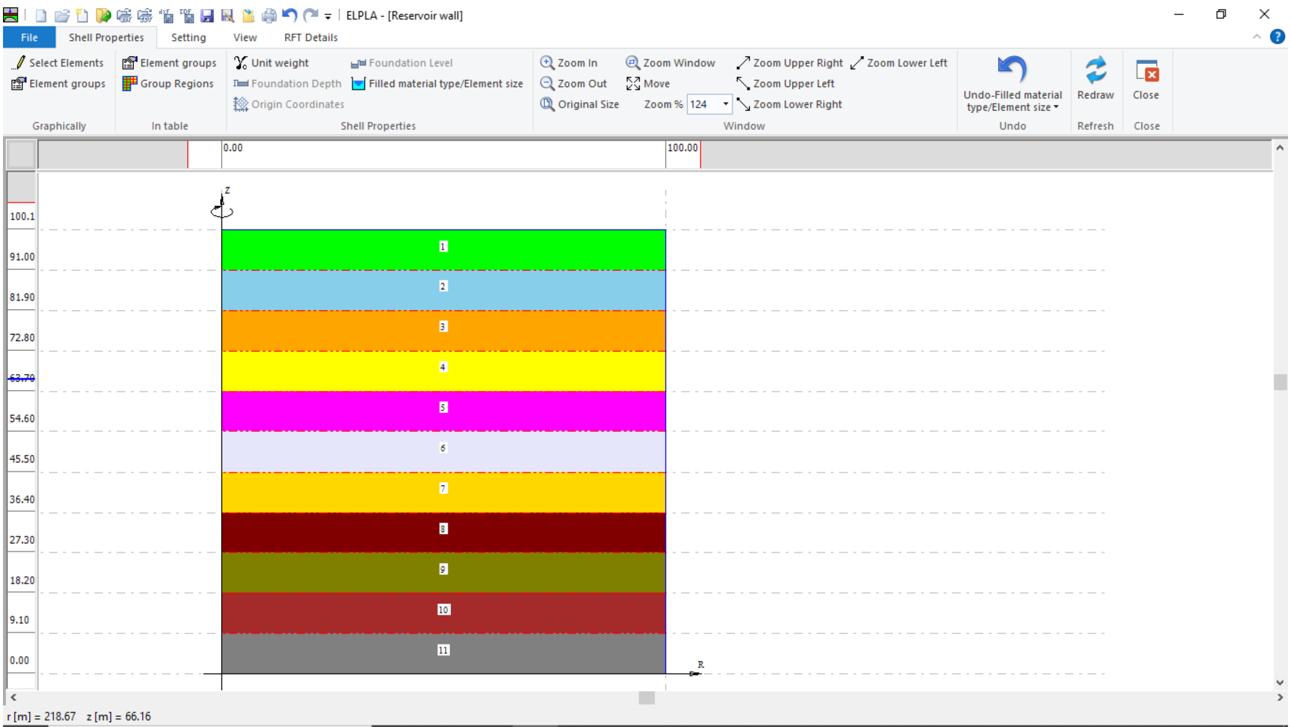


Figure 4.15 "Shell Properties" window after entering the data

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

- Choose "Save" command from "File" menu in Figure 4.15 to save the shell properties
- Choose "Close" command from "File" menu in Figure 4.15 to close the "Shell properties" window and return to *ELPLA* main window

4.5 Supports/ boundary conditions

To define the fixed support, choose "Supports/ Boundary Conditions" command from “Data” Tab. The following window in Figure 4.16 appears.

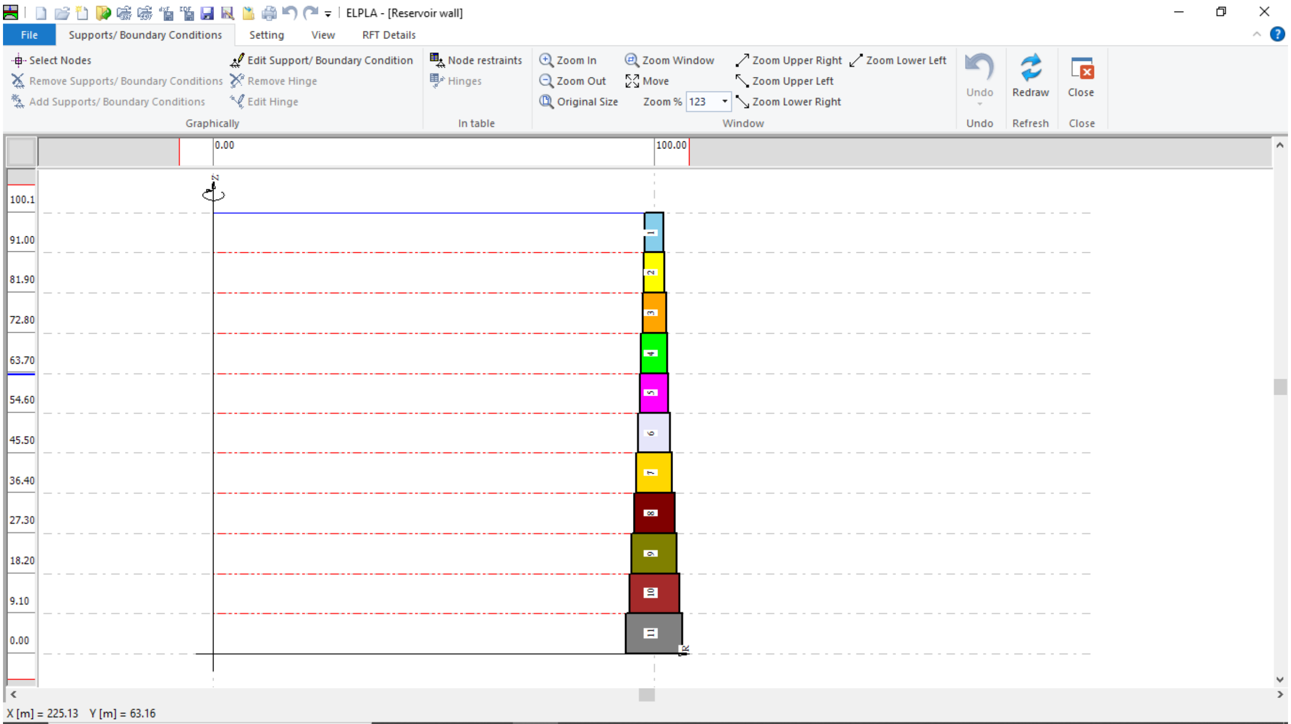


Figure 4.16 "Supports/ Boundary Conditions" Window

To define supports on the net:

- Choose "Select Nodes" command from "Graphically" menu in Figure 4.16. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the node that has the fixed support as shown in Figure 4.17
- After selecting the node, choose "Add Supports/ Boundary Conditions" command from "Graphically" menu Figure 4.16. The "Supports/ Boundary Conditions" dialog box in Figure 4.18 appears.

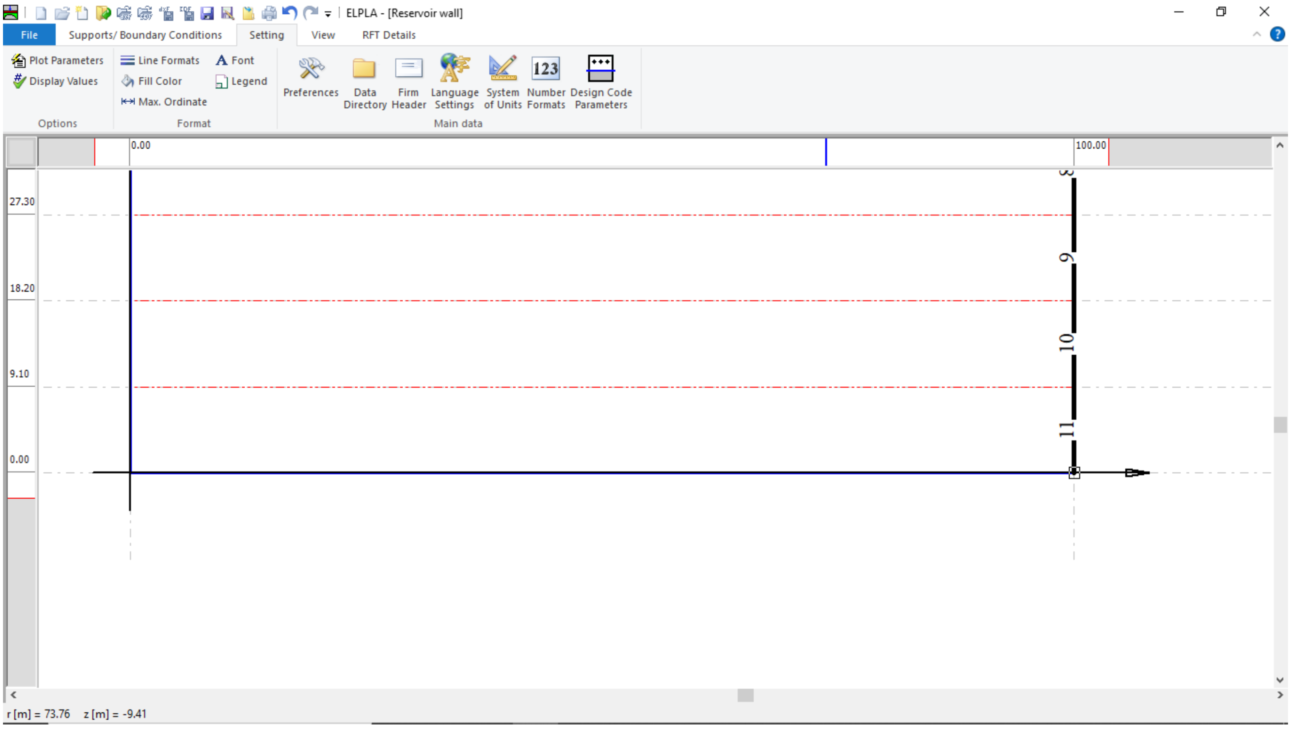


Figure 4.17 Selection of node that has a fixed support

In this dialog box

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

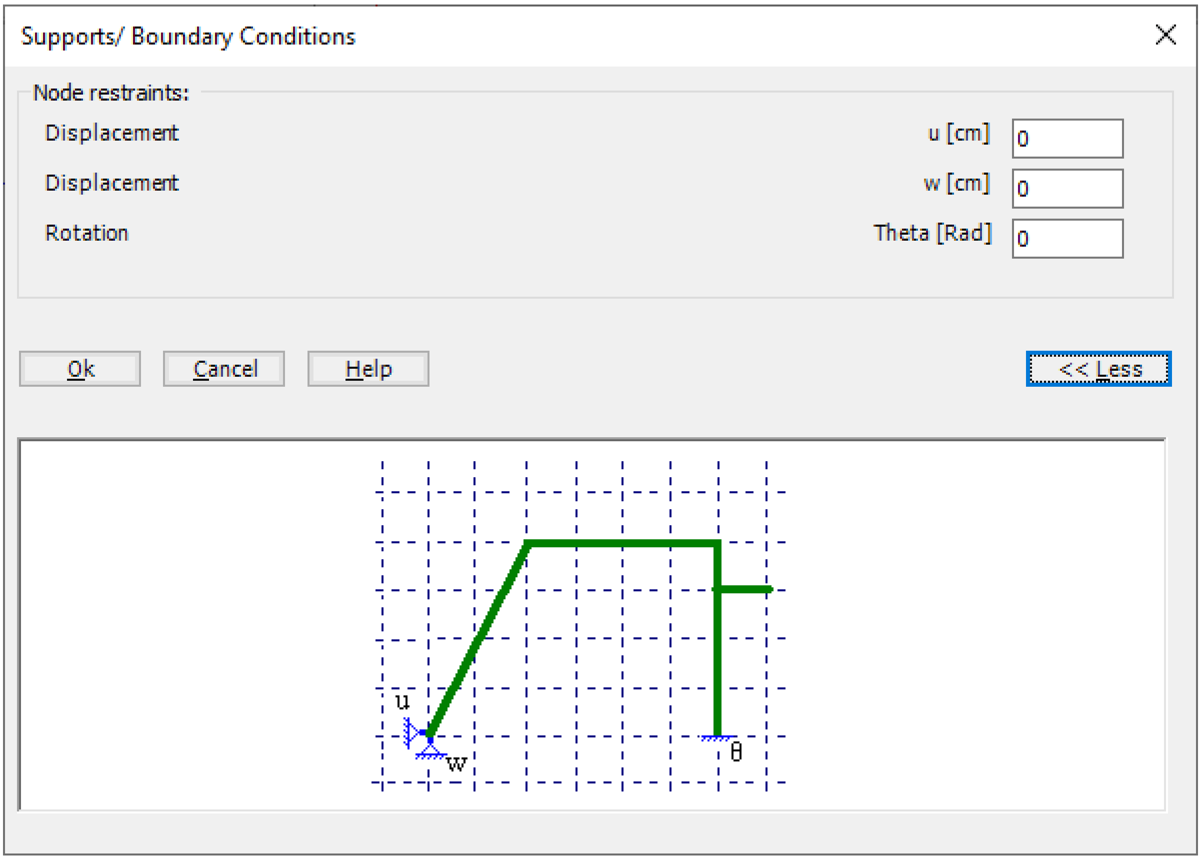


Figure 4.18 "Supports/ Boundary Conditions" dialog box

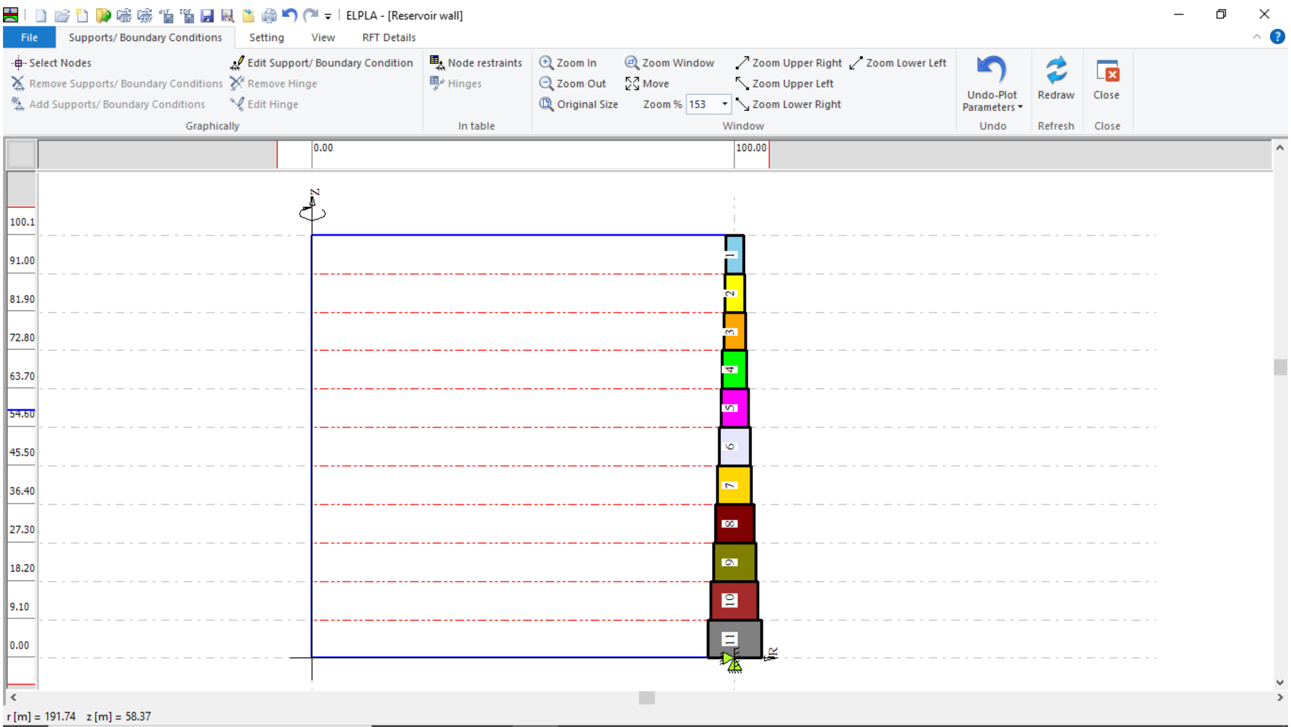


Figure 4.19 Supports on the screen

After entering supports, do the following two steps

- Choose "Save " command from "File" menu in Figure 4.19 to save the data of supports
- Choose "Close" command from "File" menu in Figure 4.19 to close the "Supports/ Boundary conditions" window and return to the main window

4.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 4.20 appears. In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 4.20. In this example, there is not applied load, as the vertical load has been already defined by the unit weight of the reservoir wall material, while the hydrostatic pressure on the reservoir wall is defined by the unit weight of water.

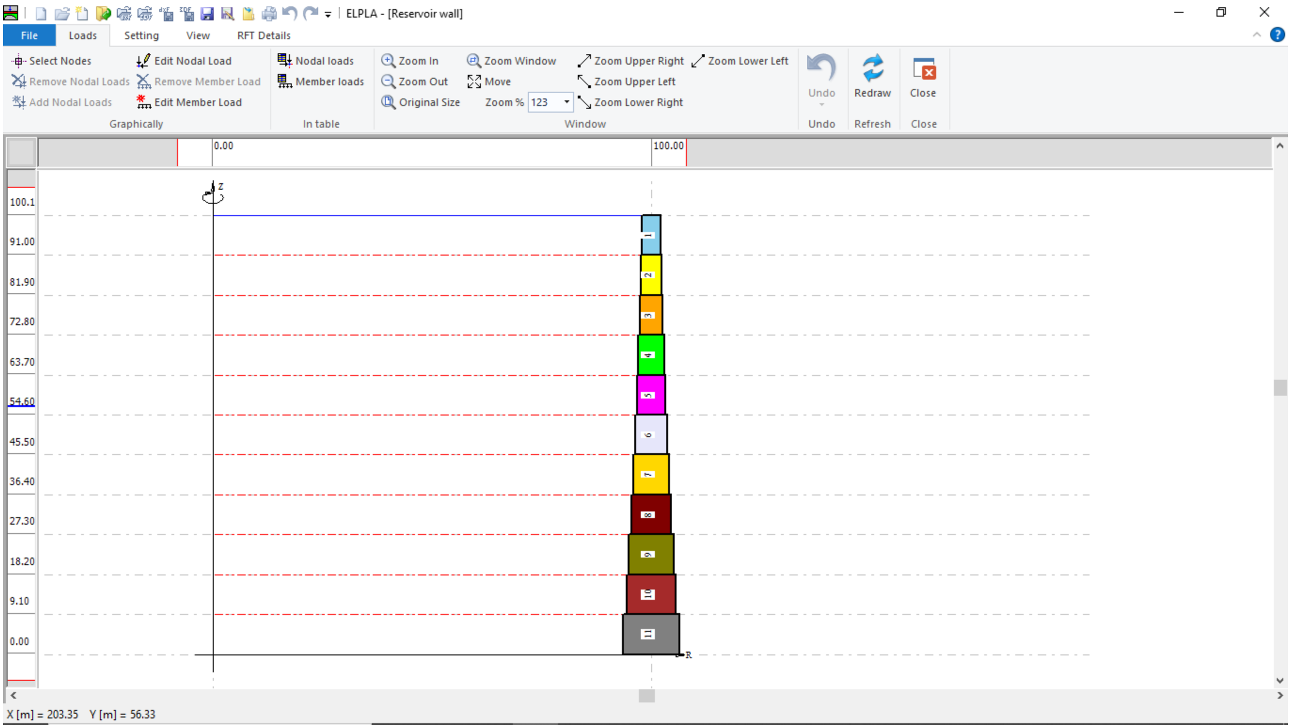


Figure 4.20 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 4.20 to save the load data
- Choose "Close" command from "File" menu in Figure 4.20 to close the "Loads" window and return to *ELPLA* main window

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

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 4.21.

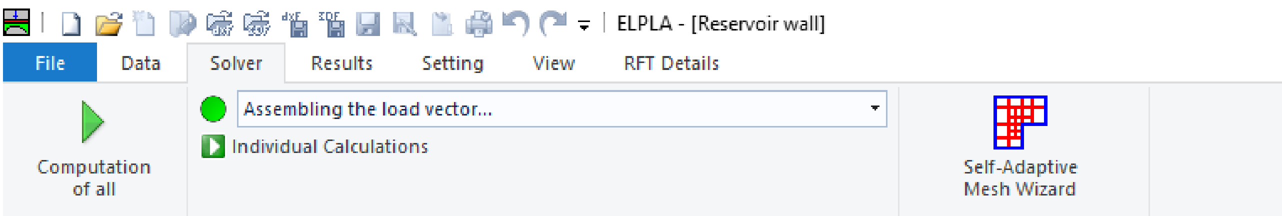


Figure 4.21 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

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

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

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab Window.

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

Analysis progress

Analysis progress menu in Figure 4.22 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

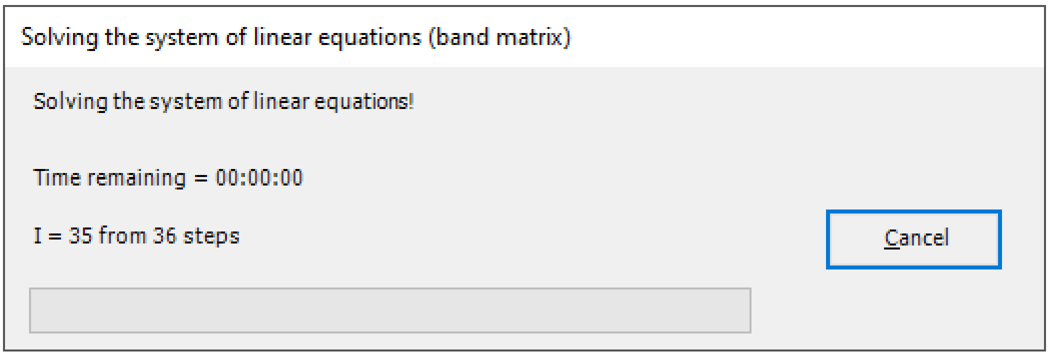


Figure 4.22 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 4.23. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

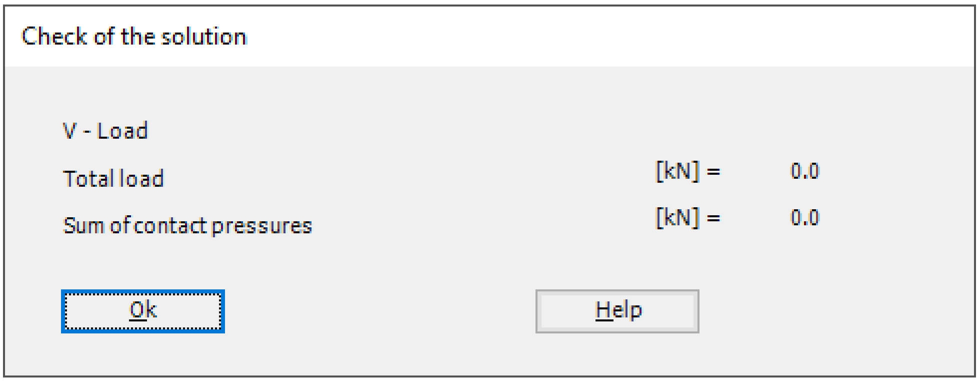


Figure 4.23 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab. To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 4.24).

Figure 4.24 "Results" Tab

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

- Data in the plan
- Supports Reactions
- Rotational shell Results
- Sections in shell wall
- Display tables of data
- Display tables of results

To view the meridional moments in the shell wall

- Choose "Sections in shell wall" command from "Section" menu. The following option box in Figure 4.25 appears
- In the "Sections in shell wall" option box, select "Meridional moments *My*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 4.26.

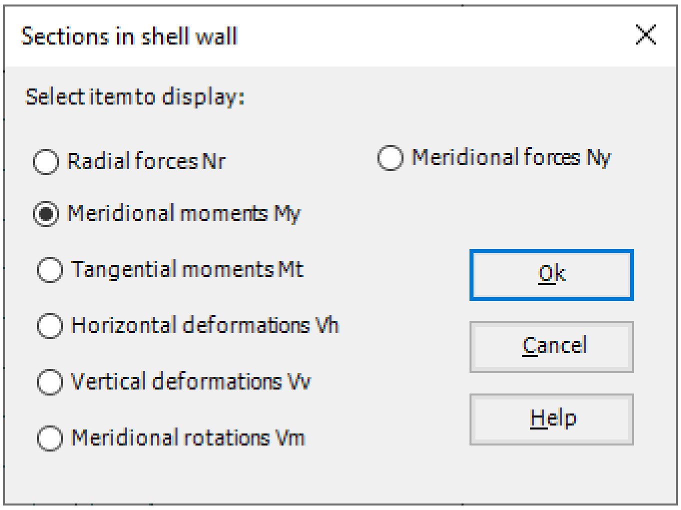


Figure 4.25 "Sections in shell base" option box

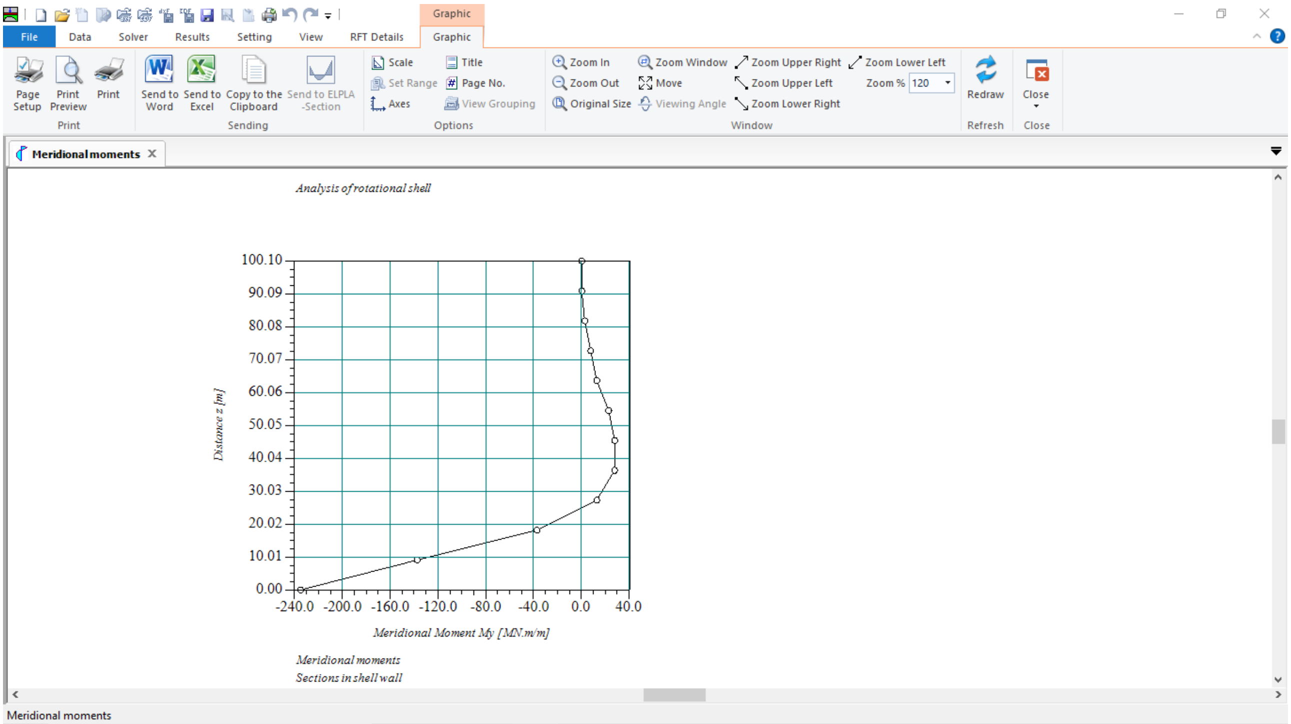


Figure 4.26 Meridional moments in shell wall

To view element groups of the reservoir

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 4.27 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

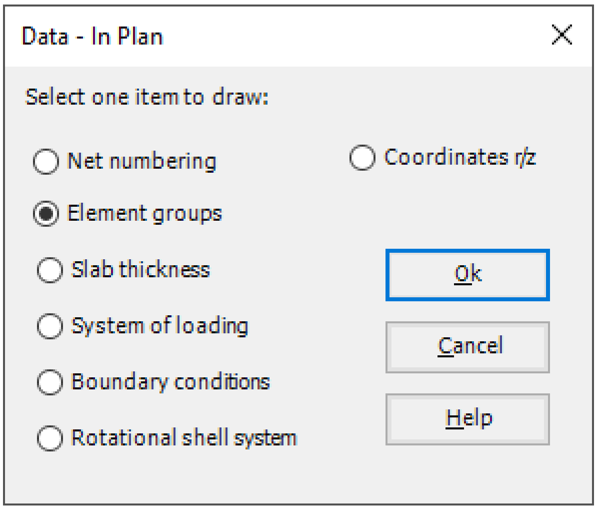


Figure 4.27 "Data – In Plan" option box

To view the supports / boundary conditions on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 4.28 appears
- In this check group box, check both "Supports Reactions *RV*", "Supports Reactions *M*" and "Supports /Boundary Conditions" check box
- The user can choose any other data to be viewed
- Click "OK" button

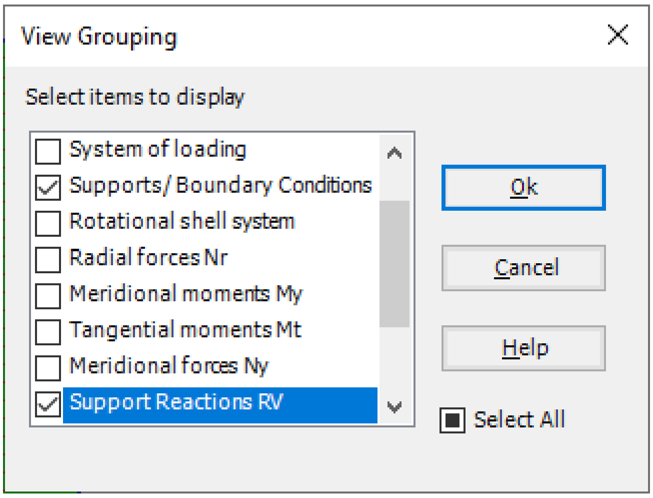


Figure 4.28 "View Grouping" check group box

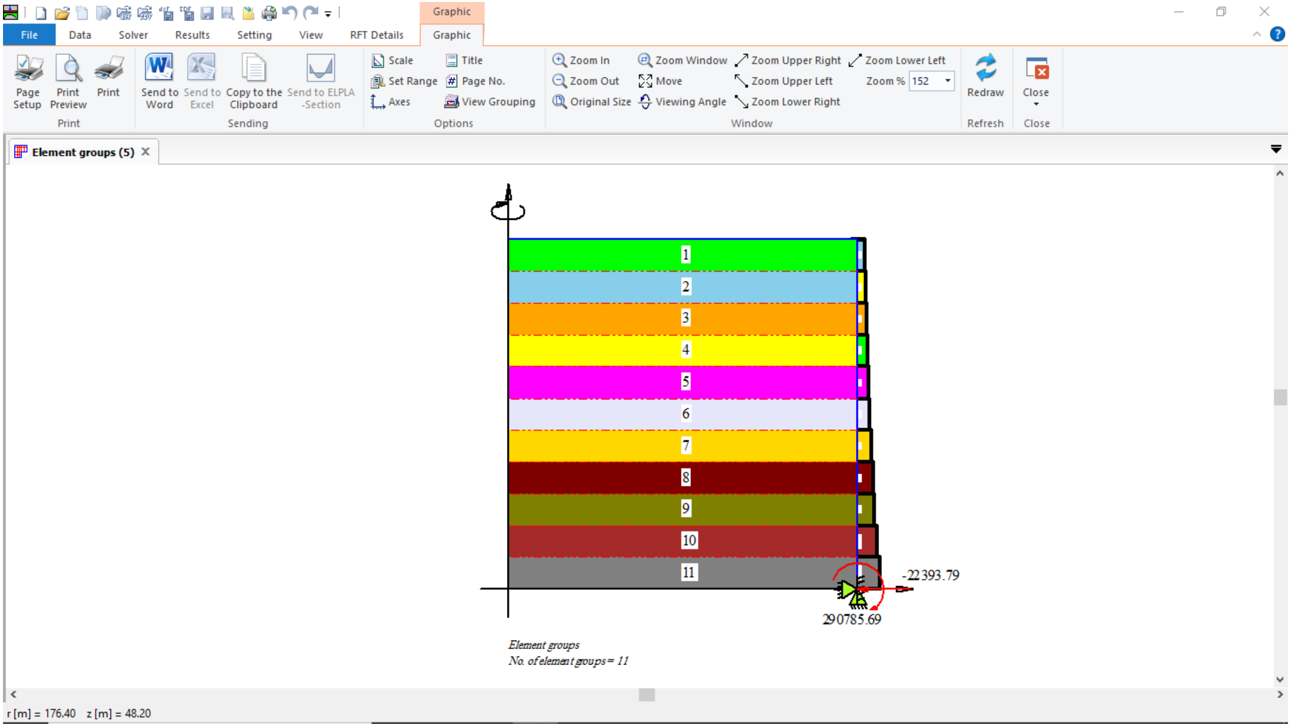


Figure 4.29 Element groups of the reservoir

Example 5

Analysis of a tank covered with a spherical dome

Contents Page

1 Description of the problem

An example of an axi-symmetrically circular cylindrical tank covered with a spherical dome roof is selected illustrate some features of *ELPLA* for analyzing shell elements.

2 Tank geometry and properties

Figure 5.1 shows half of an axial section of a large-diameter reinforced concrete circular cylindrical tank covered with a dome roof. The wall connection with the roof is monolithic, while the end of the wall is fixed at the base. Details concerning the geometry of the structure are shown in Figure 5.1. The elastic properties of the tank material are shown in Table 5.1. Only the self-weight is considered in this analysis.

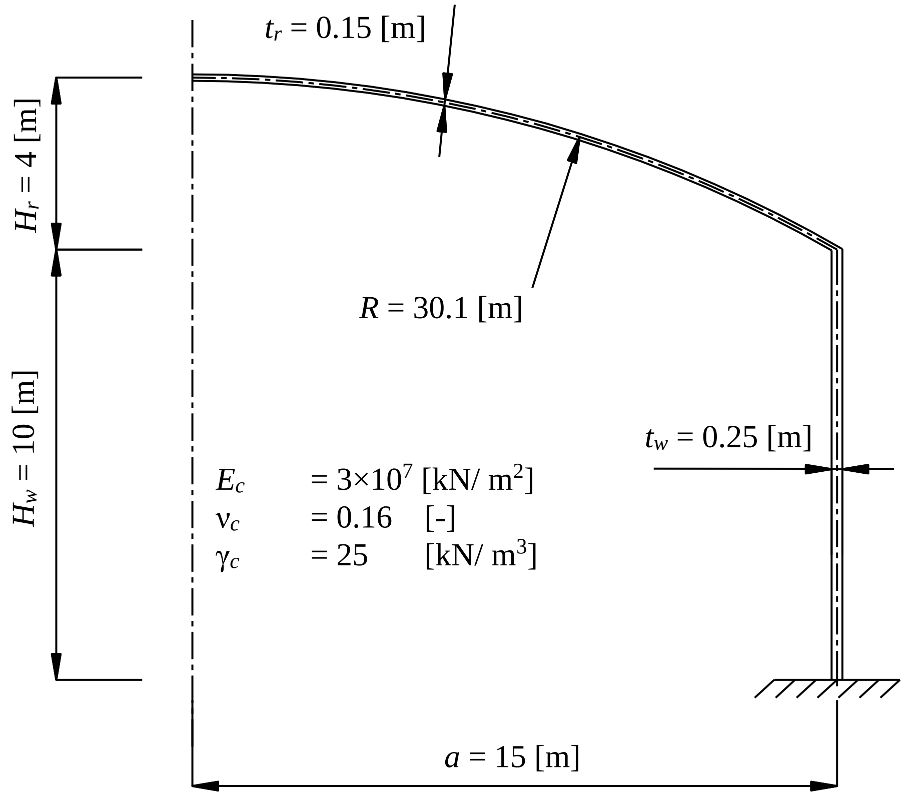


Figure 5.1 Radial section through the tank

Table 5.1 Tank material

Modulus of Elasticity of the tank material	E_c	= 3×10^7	[kN/ m^2]
Poisson's ratio of the tank material	ν_c	= 0.16	[-]
Unit weight of the tank material	γ_c	= 25	[kN/ m^3]

3 Numerical Analysis

In order to analyze the tank, the height of the wall is divided into 50 equal elements, each of 0.20 [m], while the roof shell (dome) is divided into 40 equal arcs each of 0.75 [°] as shown in Figure 5.2.

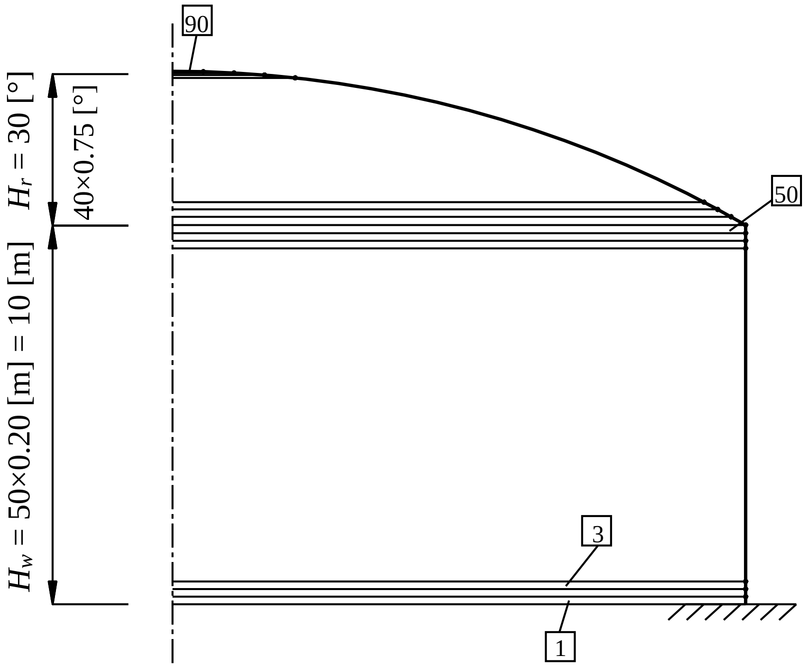


Figure 5.2 Finite element mesh of the tank

4 Creating the project

In this section, the user will learn how to create a project for analyzing an axi-symmetrically circular cylindrical tank covered with a spherical dome roof. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 5.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of Forms. The first Form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 5.3).

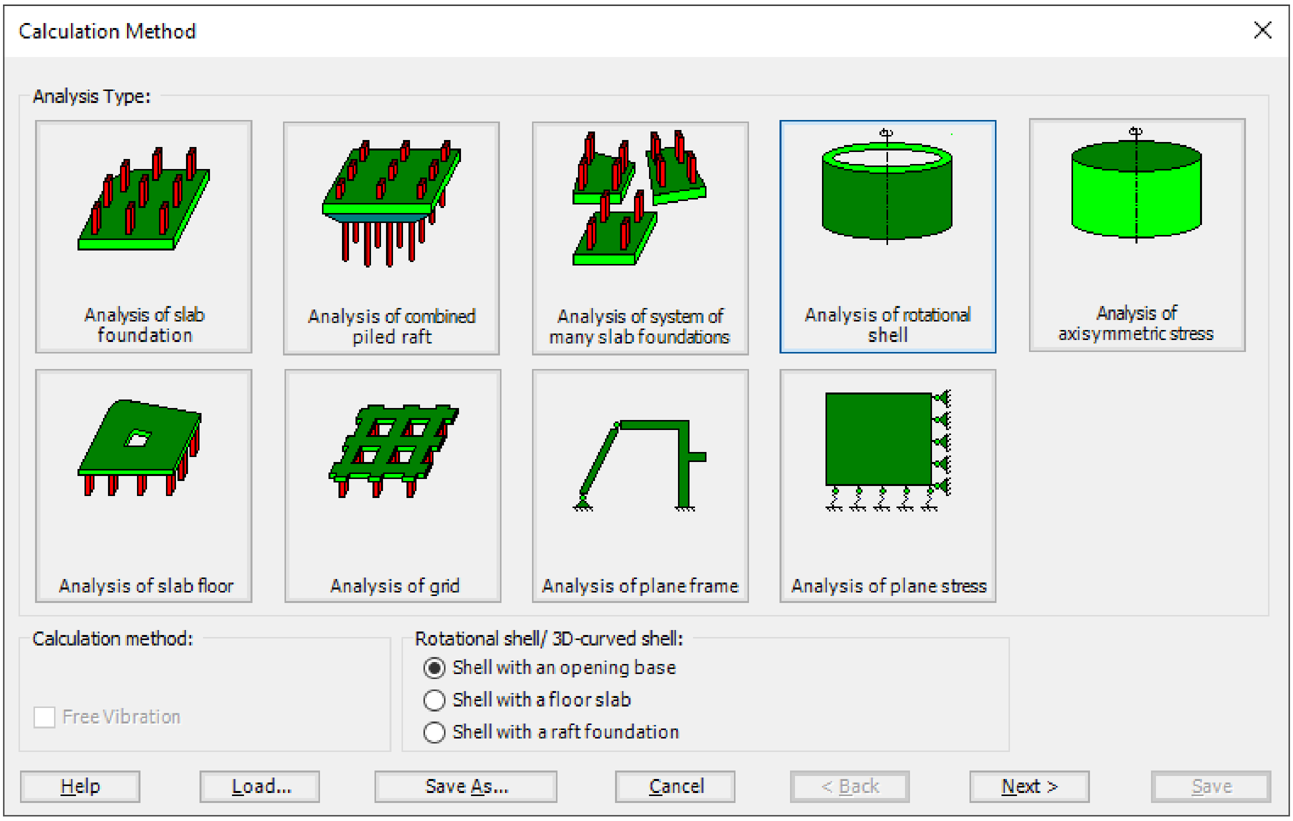


Figure 5.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 5.3, define the analysis type of the problem. As the analysis type is a circular cylindrical covered with a spherical dome roof problem, select "Analysis of rotational Shell" button, and check "Shell with an opening base" option, then click "Next" button to go to the next Form.

The last Form in the wizard is the "Options" Form, Figure 5.4. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Select "Supports/ Boundary Conditions", then click the "Save" button.

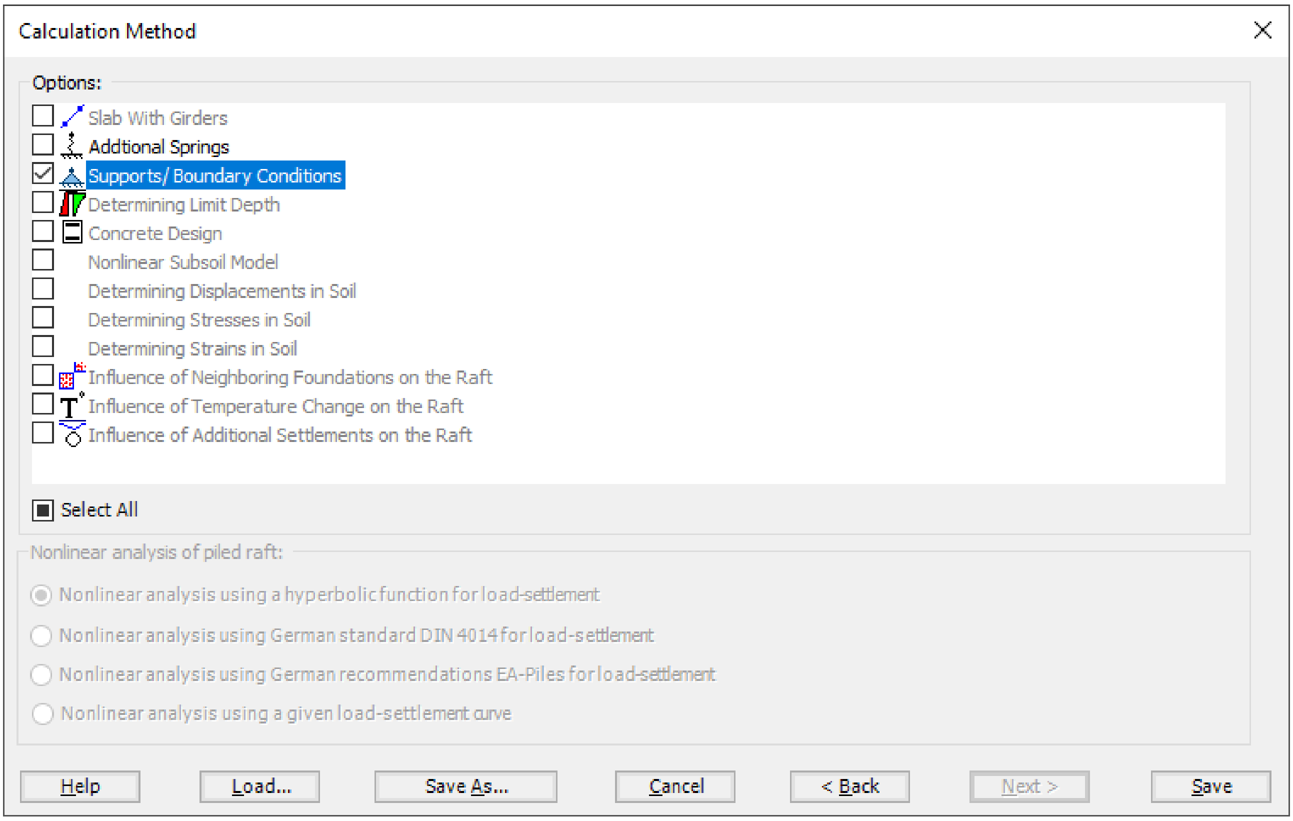


Figure 5.4 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 5.5. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Tank with covered roof". *ELPLA* will use automatically this file name in all reading and writing processes.

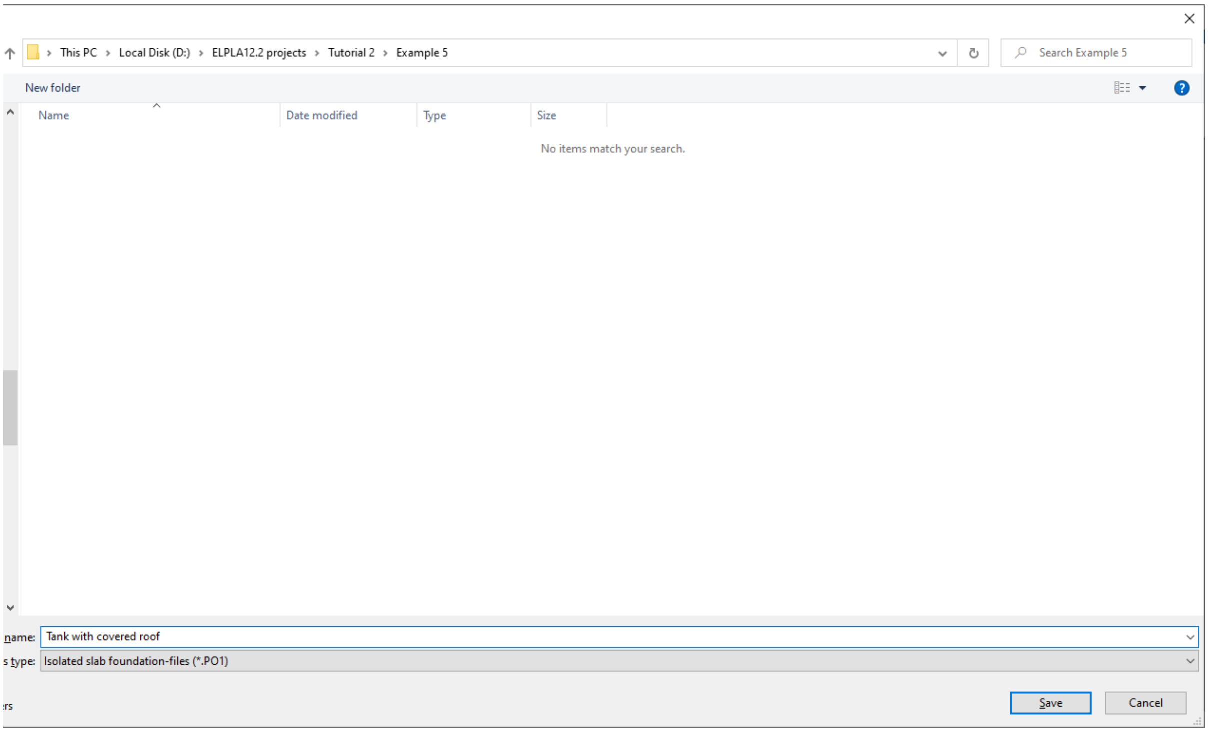


Figure 5.5 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Tank with covered roof] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 5.6 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a tank covered with a spherical dome"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

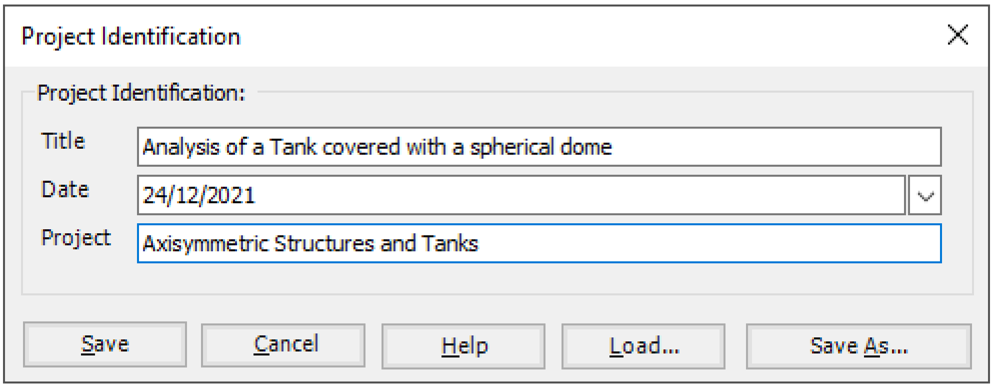


Figure 5.6 "Project Identification" dialog box

4.3 FE-Net data

For the given problem, a tank covered with a dome roof has a lower radius of a = 15 [m], a clear height of H_w = 10 [m], and the radius of the spherical roof is R = 30.1 [m]. the height of the wall is divided into 50 equal elements, each of 0.20 [m], while the roof shell (dome) of 30[°] is divided into 40 equal arcs each of 0.75 [°] (gives also 40 elements). To define the FE-Net for this tank, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 5.7. This wizard will guide you through the steps required to generate a FE-Net.

The first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

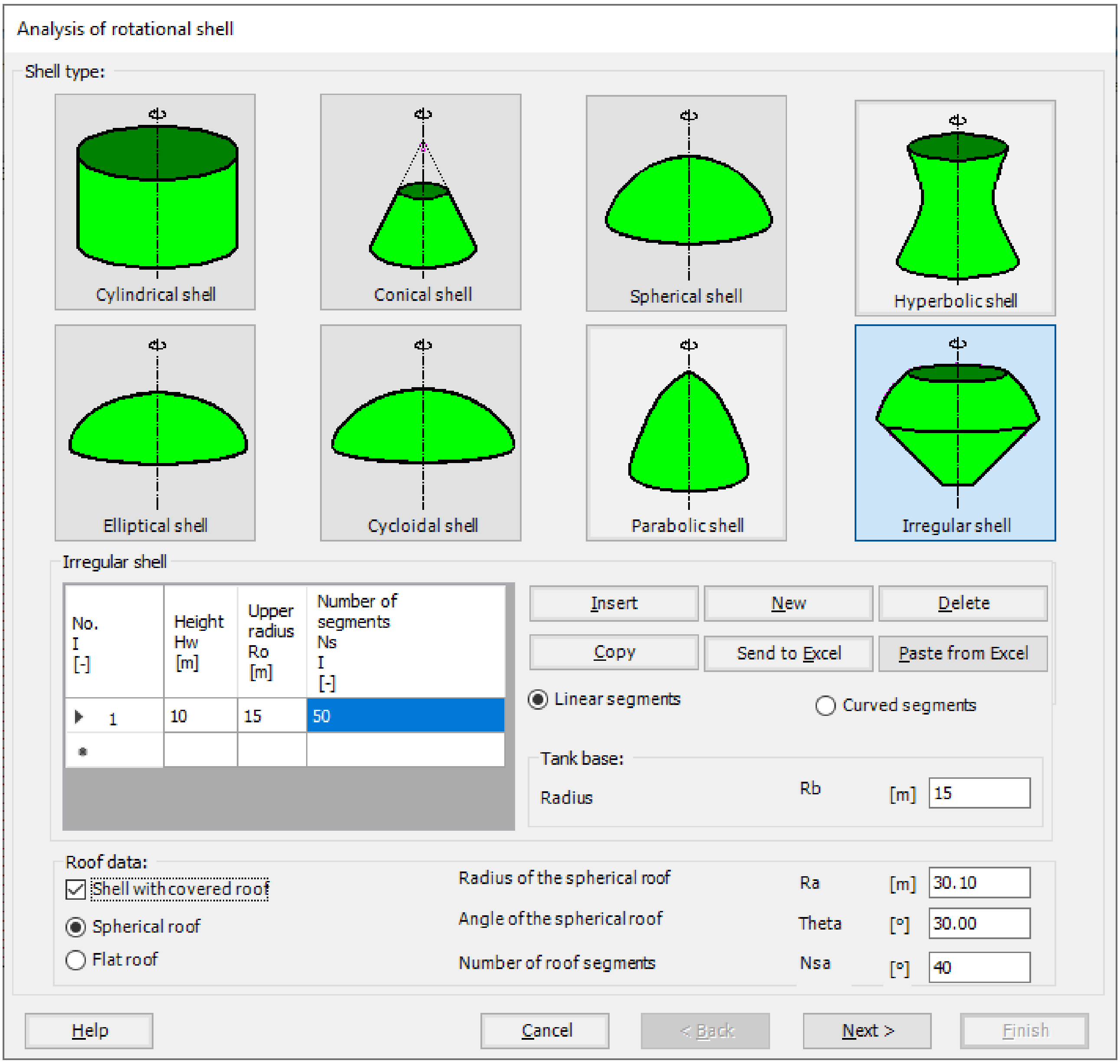


Figure 5.7 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net of the tank:

- In the "Shell type" options choose "Irregular shell" button
  To define the base of the tank:
  - Type 15 in the "Tank base radius *Rb*" edit box

  To define the height of the tank:
  - Choose "Linear segments" option
  - Type 10 in the "height *Hw*" edit box
  - Type 15 in the "Upper radius *Ro*" edit box, as the upper radius is the same as the base radius
  - Type 50 in the "Number of segments *Ns*" edit box

  To define the roof of the tank:
  - Check the "Shell with covered roof" check box
  - Choose "Spherical roof" option
  - Type 30.10 in the "Radius of the spherical roof" edit box
  - Type 30 in the "Angle of the spherical roof" edit box
  - Type 40 in the "Number of roof segments" edit box
- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Irregular shell" Form appears Figure 5.8, *ELPLA* divides the height of the wall into 50 equal elements, each of 0.20 [m], while the roof shell (dome) is divided into 40 equal arcs each of 0.75 [°], the user can edit the data of the segments individually by using "Modify" button, or all of them by using "In Table" button, if it is necessary.

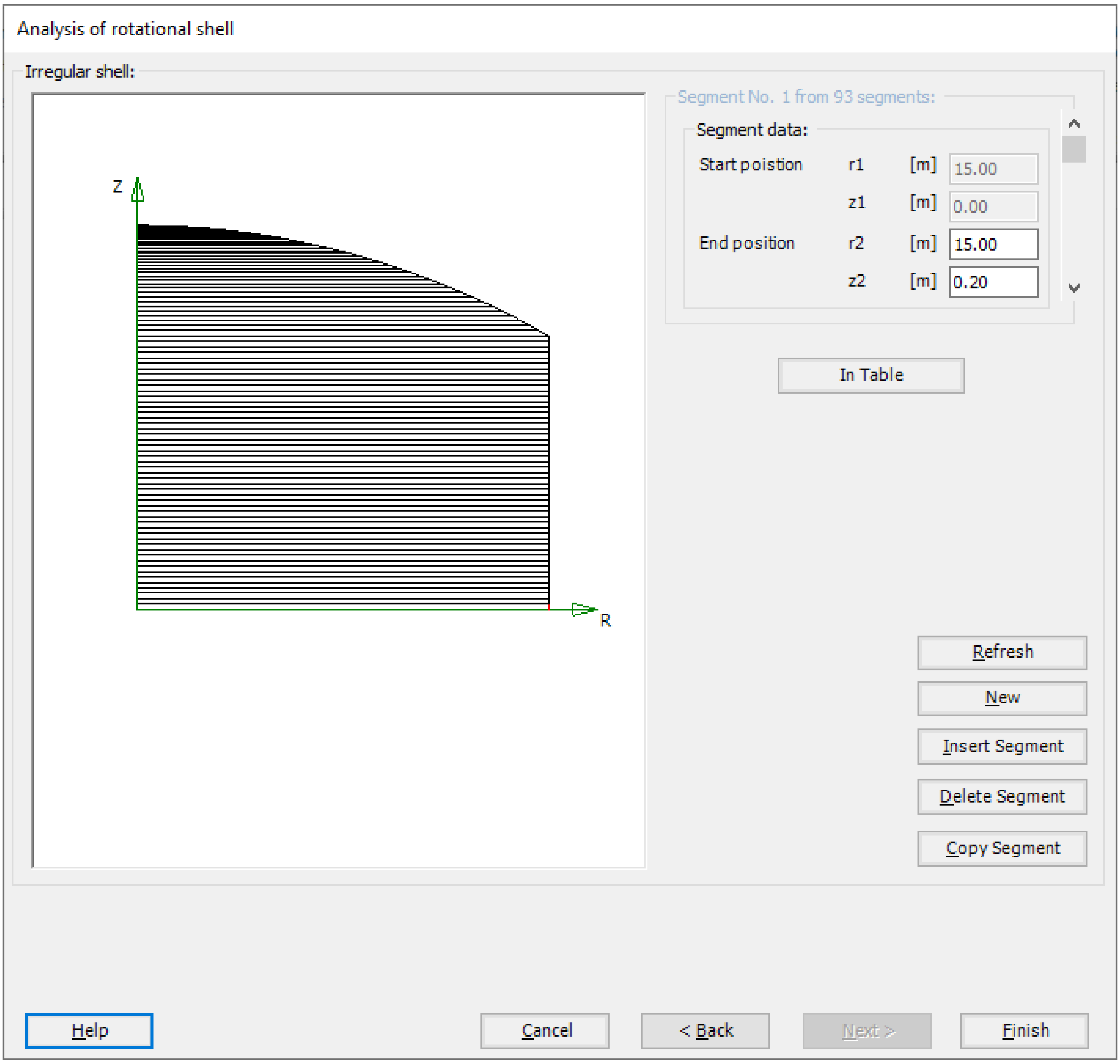


Figure 5.8 "Irregular shell" Form

Click "Finish" in "Analysis of rotational shell" wizard, to generate the FE-Net. The generated FE-Net appears Figure 5.9.

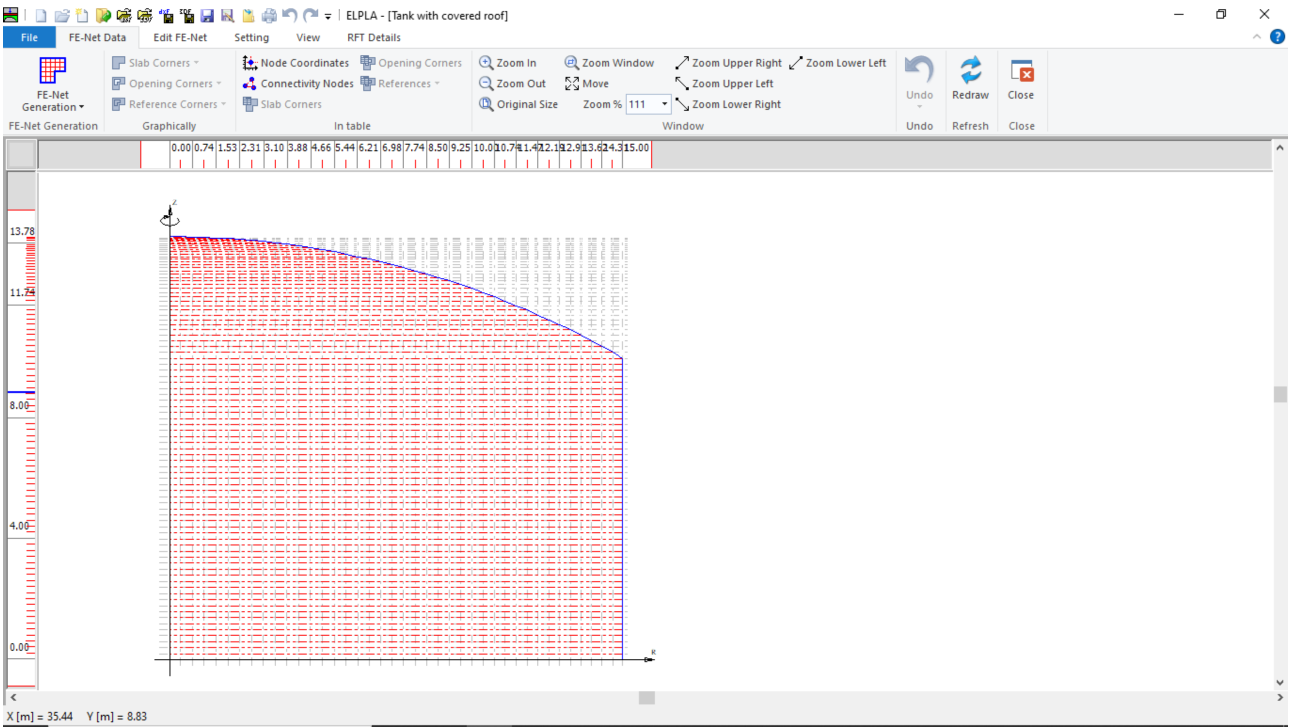


Figure 5.9 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 5.9 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 5.9 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Shell properties

To define the tank properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 5.10 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, group regions, unit weight of the tank, and filled material properties.

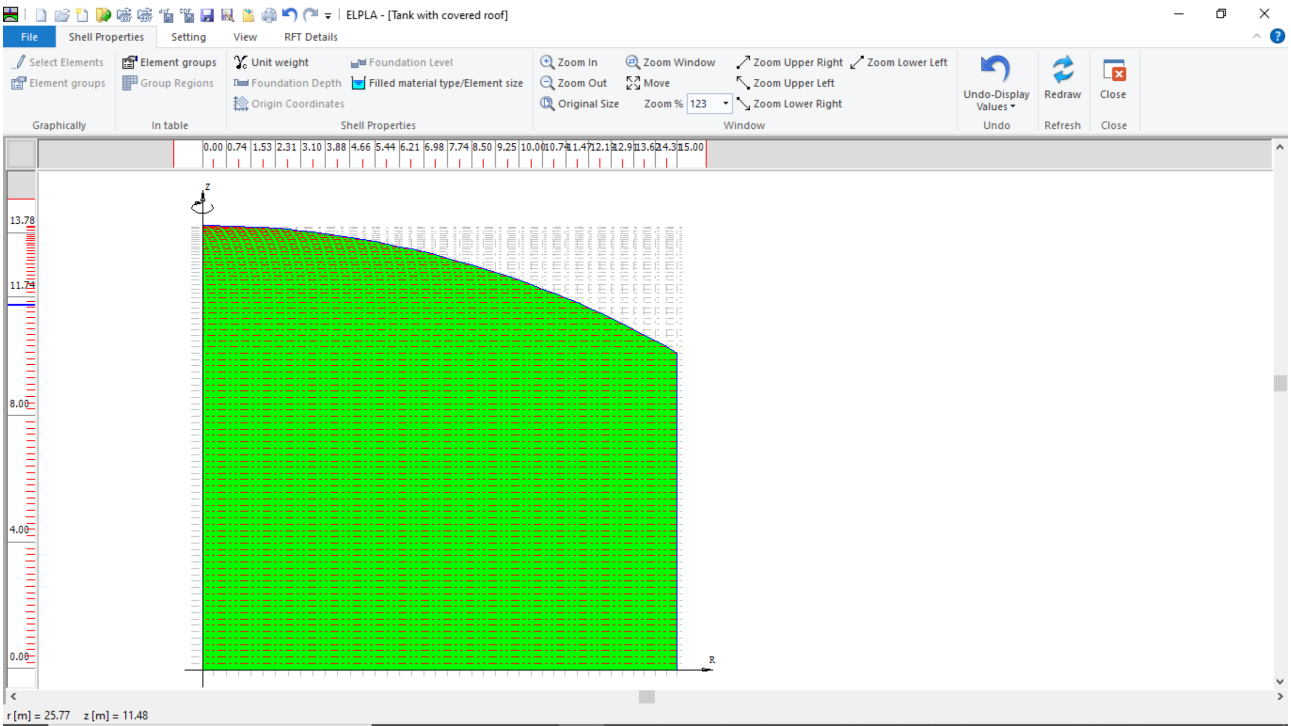


Figure 5.10 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 5.11 appears. In this list box, define E-Modulus, *Poisson's* ratio and slab thickness for both the tank wall and the tank roof as they differ in thickness. Then click "OK" button.

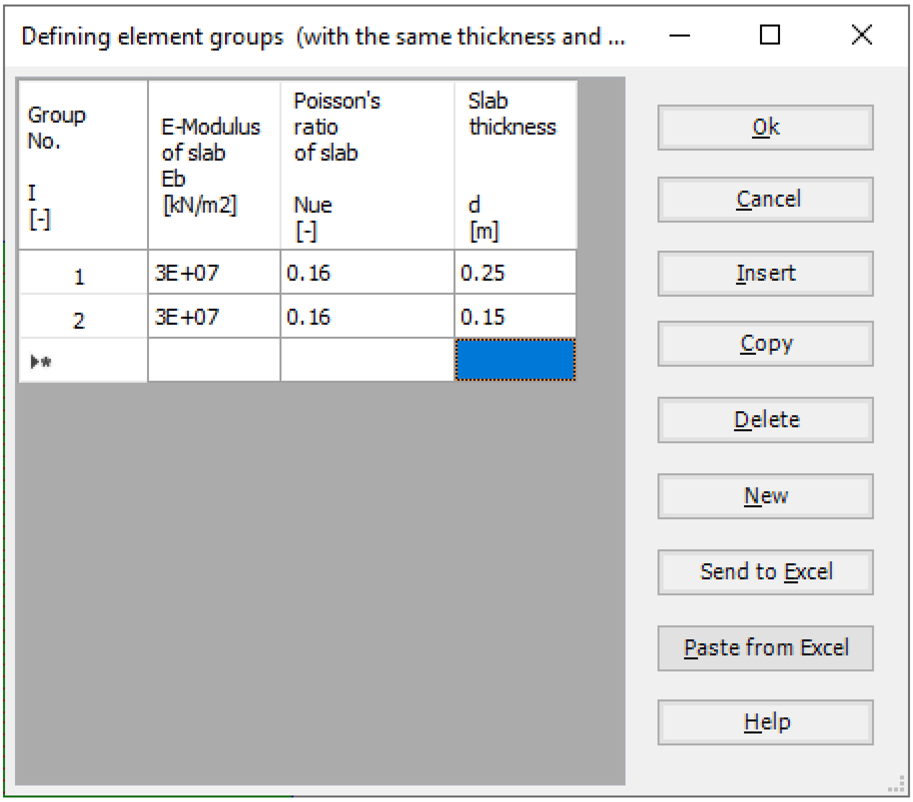


Figure 5.11 "Defining element groups" list box

Defining the slab thickness for materials on the net may be carried out either graphically or numerically (in a table). In the current example, the user will define the slab thickness on the net graphically.

To define the slab thickness for the tank roof

- Choose "Select Elements" command from "Graphically" menu in the window of Figure 5.10.
- When "Select Elements" command is chosen, the cursor will change from an arrow to a cross hair. A group of elements can be selected by holding the left mouse button down at the corner of the region. Then, drag the mouse until a rectangle encompasses the required group of elements. When the left mouse button is released, all elements in the rectangle are selected
- Select the elements that include the tank roof as Figure 5.12
- Choose "Elements Groups" command from "Graphically" menu in the window of Figure 5.10, "Group Regions" dialog box Figure 5.13 appears
- Define the "Group No." as type "2", while "Group No." of the wall elements will be as type "1", where type "1" is the default "Group No." then click "OK" button

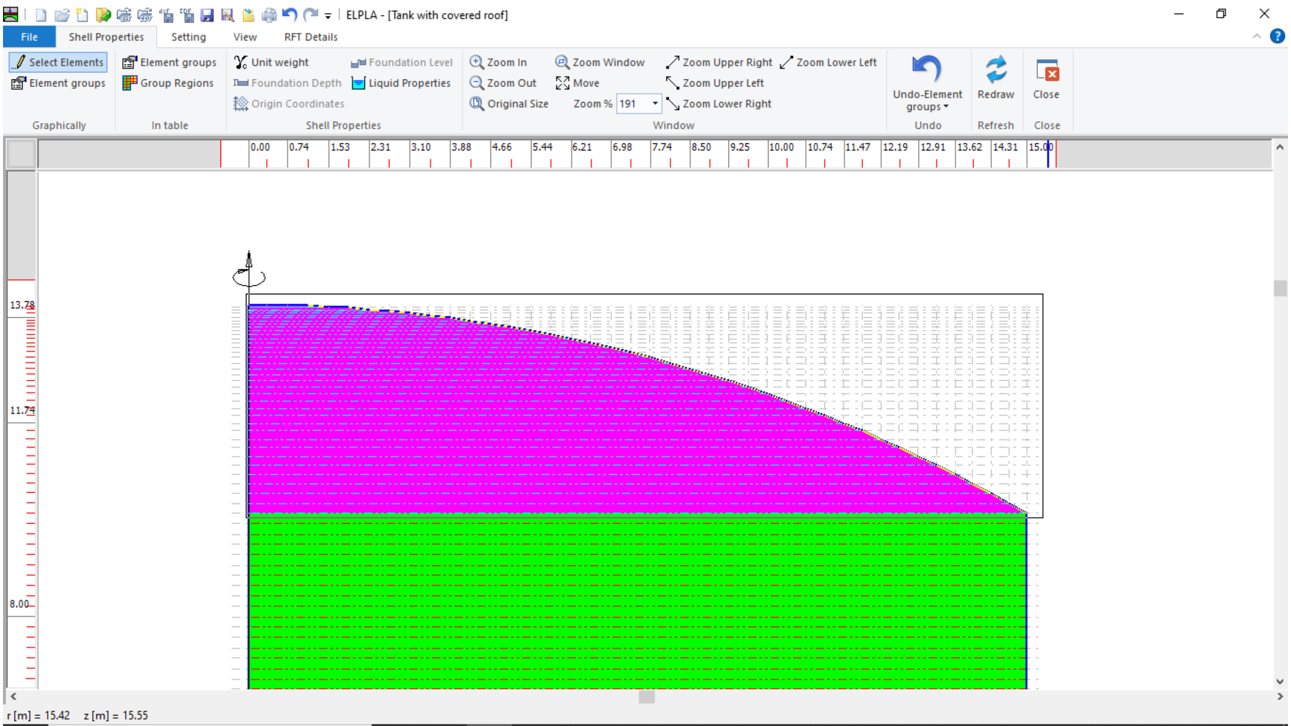


Figure 5.12 Selecting the nodes that include the tank roof

Figure 5.13 "Group Regions" dialog box

To enter the unit weight of the tank, choose "Unit weight" command from "Shell Properties" menu in Figure 5.10. The following dialog box in Figure 5.14 with a default unit weight of 25 [kN/m^3] appears, click "OK" button.

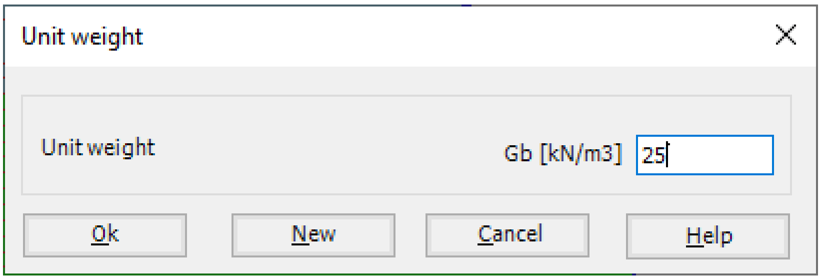


Figure 5.14 "Unit weight" dialog box

To define the element size of the shell, choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 5.10. The following form in Figure 5.15 appears.

To define the element size of the ring wall:

- Select "Empty container" option
- Check the "Constant element sizes in z-direction" check box
- Type 1 in the "Element size in each shell segment" edit box. The element size is chosen to be 1 [m] larger than the segment size in order to ignore further subdivision of the segments into smaller elements. In some cases, it is necessary to divide the segment into smaller elements in order to make the analysis more precise. Nevertheless, the final results of the internal forces appear only at nodes of segments
- Click "OK" button

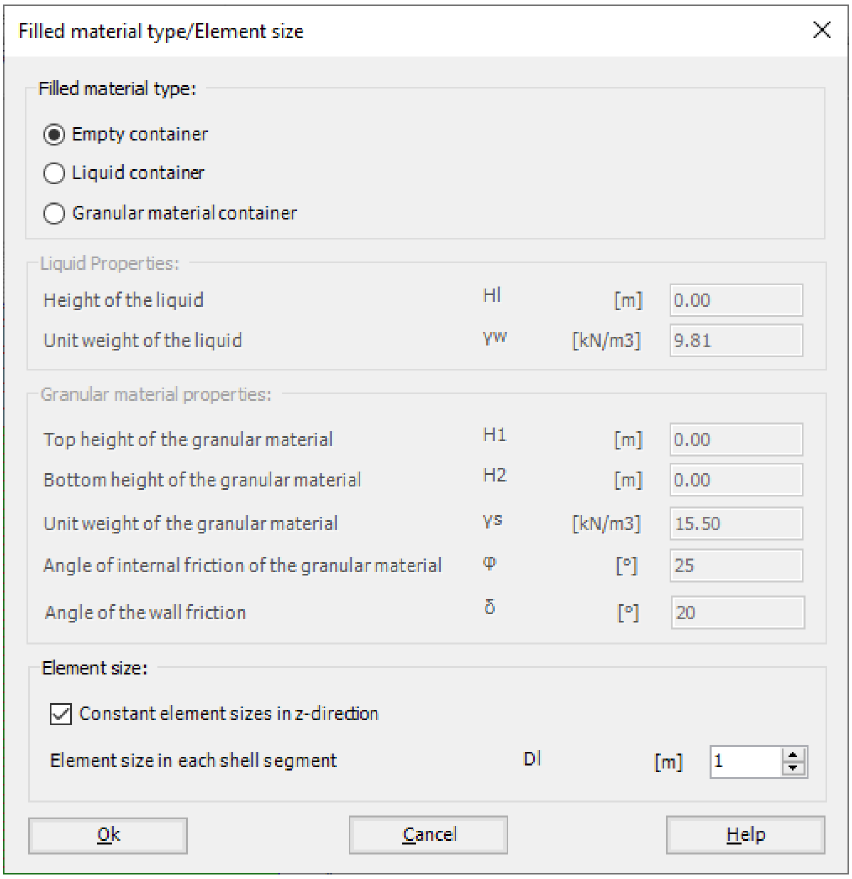


Figure 5.15 "Liquid Properties/Element size" dialog box

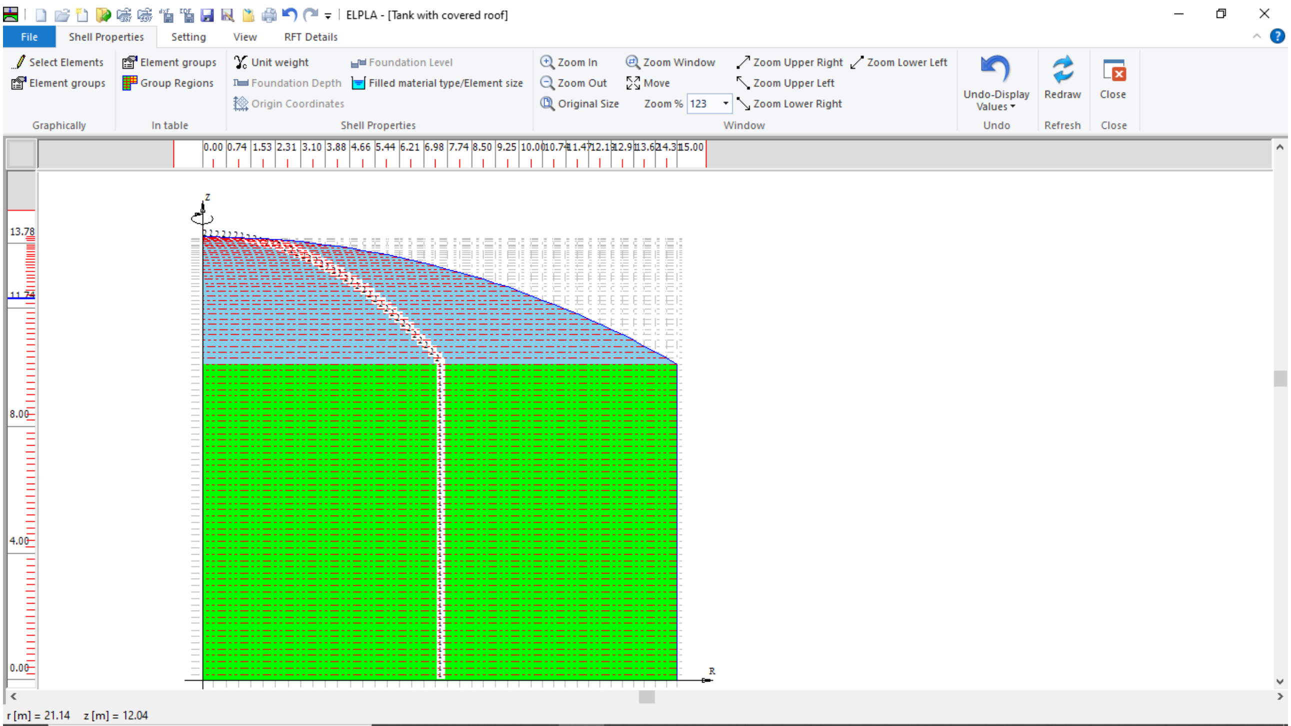


Figure 5.16 "Shell Properties" window after defining the shell data

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

- Choose "Save" command from "File" menu in Figure 5.16 to save the shell properties
- Choose "Close" command from "File" menu in Figure 5.16 to close the "Shell properties" window and return to *ELPLA* main window

4.5 Supports/ boundary conditions

To define the support, choose "Supports/ Boundary Conditions" command from “Data” Tab. The following window in Figure 5.17 appears.

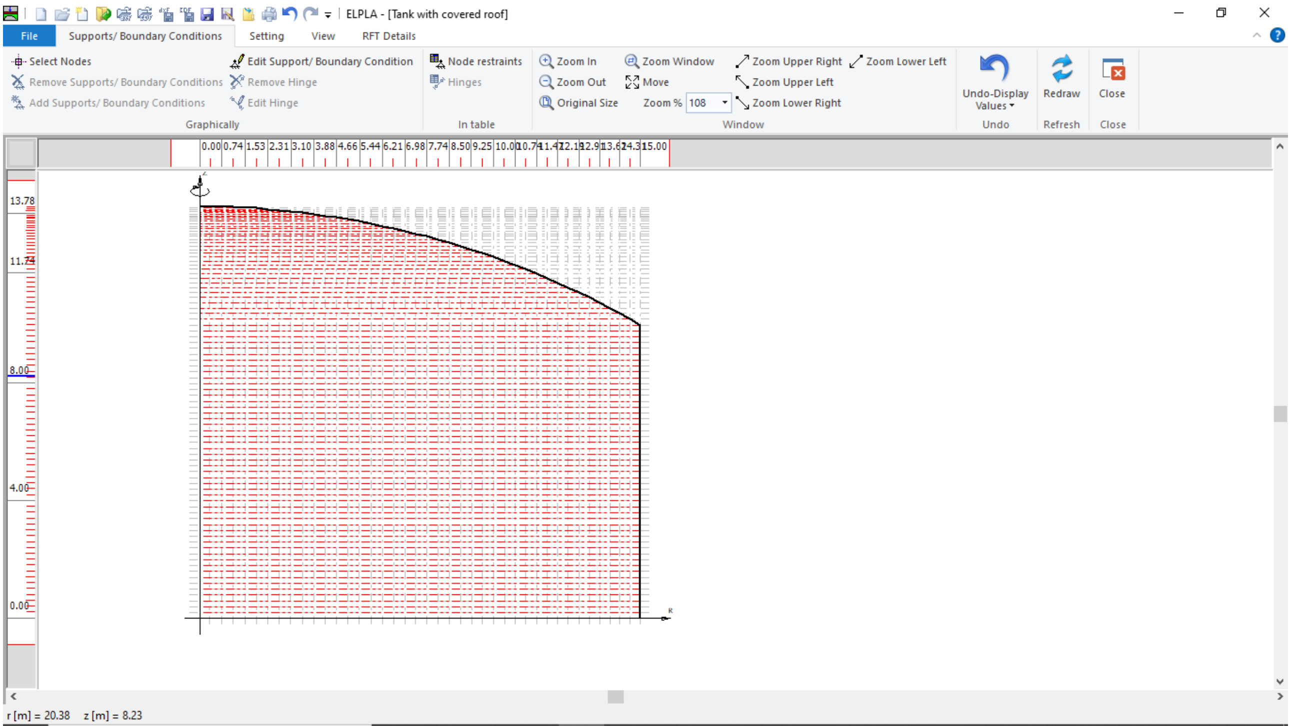


Figure 5.17 "Supports/ Boundary Conditions" Window

To define the support on the net:

- Choose "Select Nodes" command from "Graphically" menu in Figure 5.17. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the node that has the fixed support as shown in Figure 5.18
- After selecting the node, choose "Add Supports/ Boundary Conditions" command from "Graphically" menu (Figure 5.17). The "Supports/ Boundary Conditions" dialog box in Figure 5.19 appears.

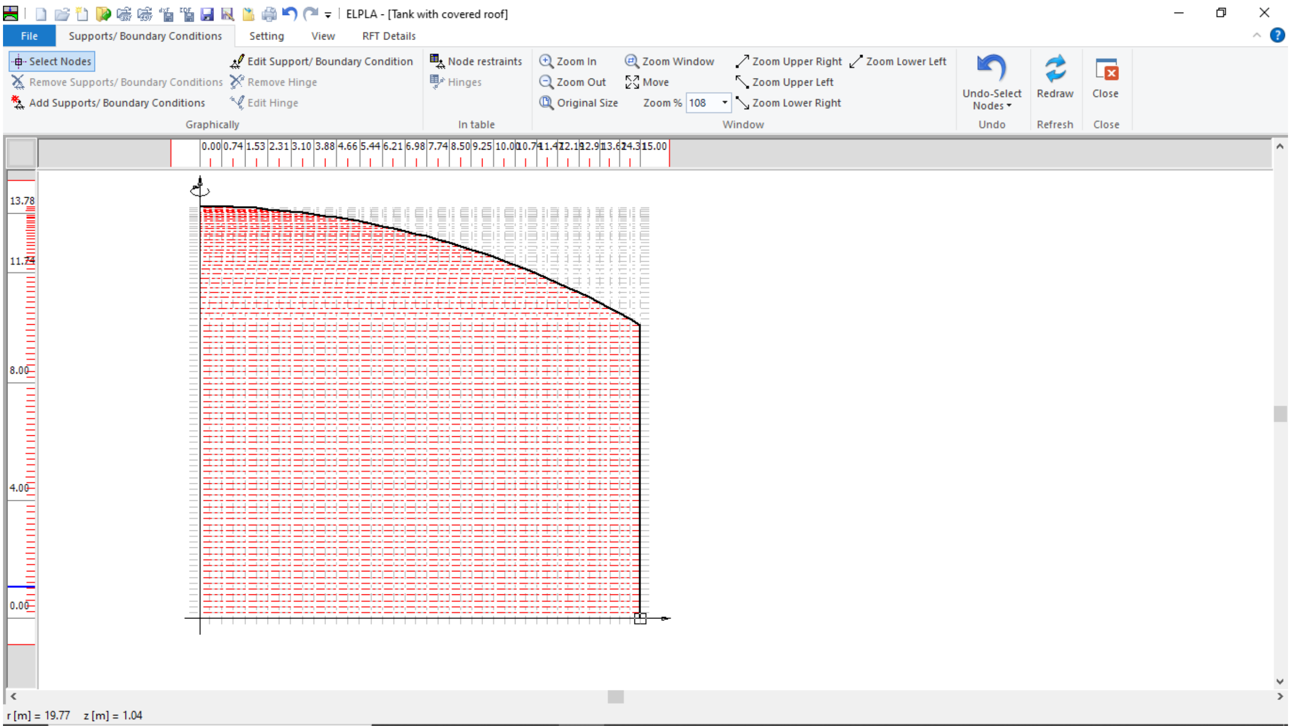


Figure 5.18 Selection of the node that has the fixed support

In this dialog box

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

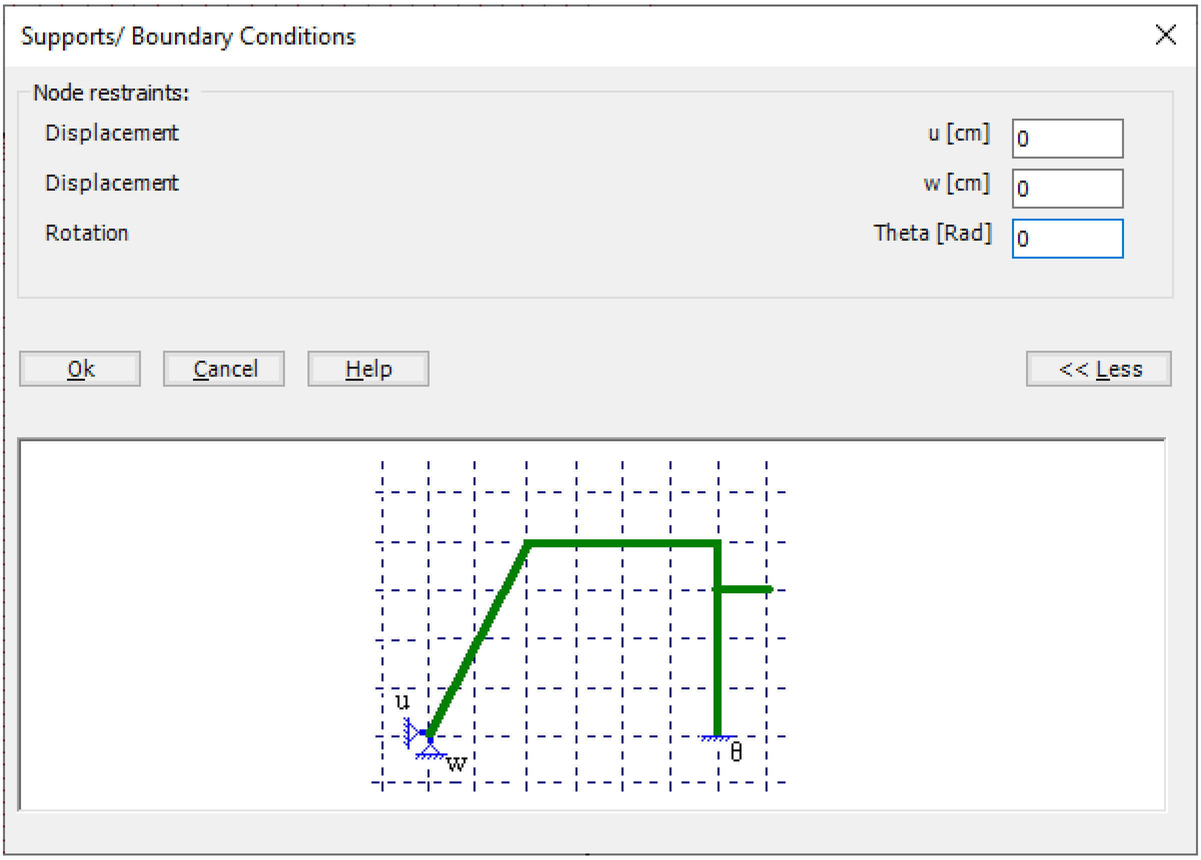


Figure 5.19 "Supports/ Boundary Conditions" dialog box

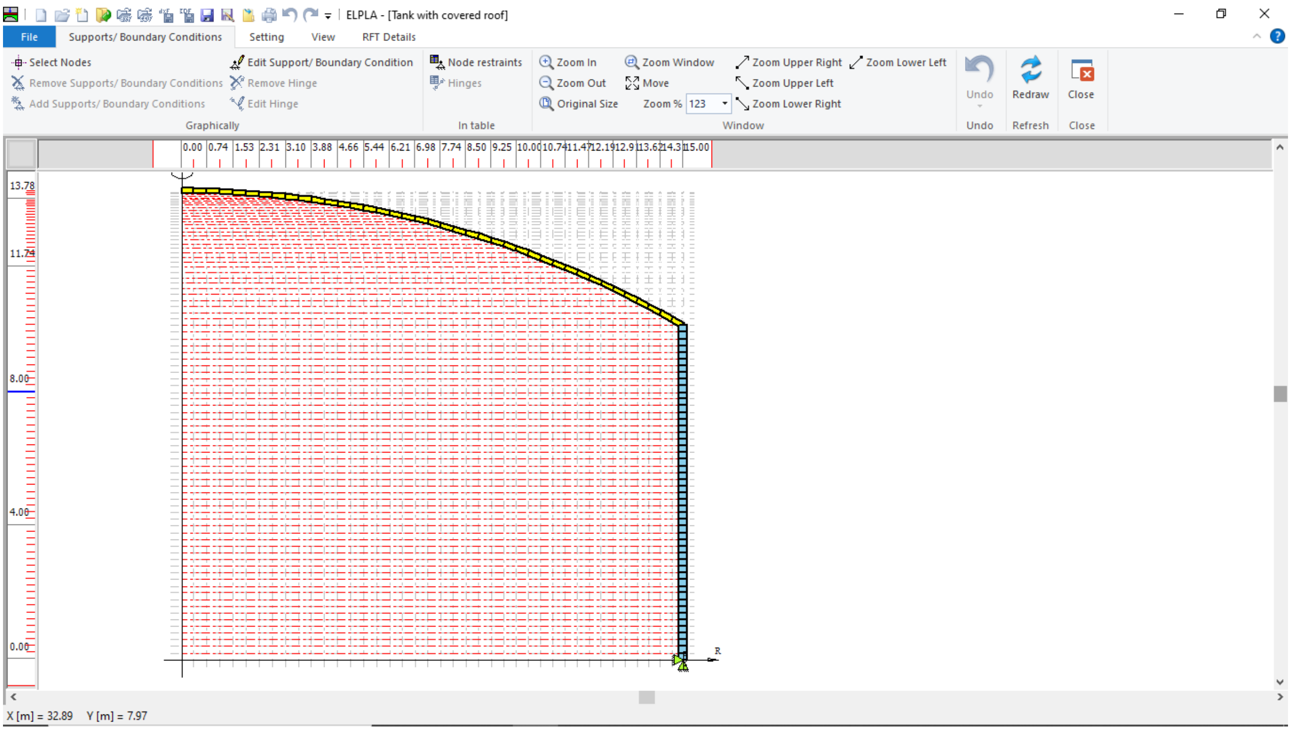


Figure 5.20 "Supports/ Boundary Conditions" window after defining the support

After defining the supports, do the following two steps

- Choose "Save " command from "File" menu in Figure 5.20 to save the data of supports
- Choose "Close" command from "File" menu in Figure 5.20 to close the "Supports/ Boundary conditions" window and return to the main window

4.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 5.21 appears. In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 5.21. In this example, there is not applied load, as the vertical load has been already defined by the unit weight of the tank.

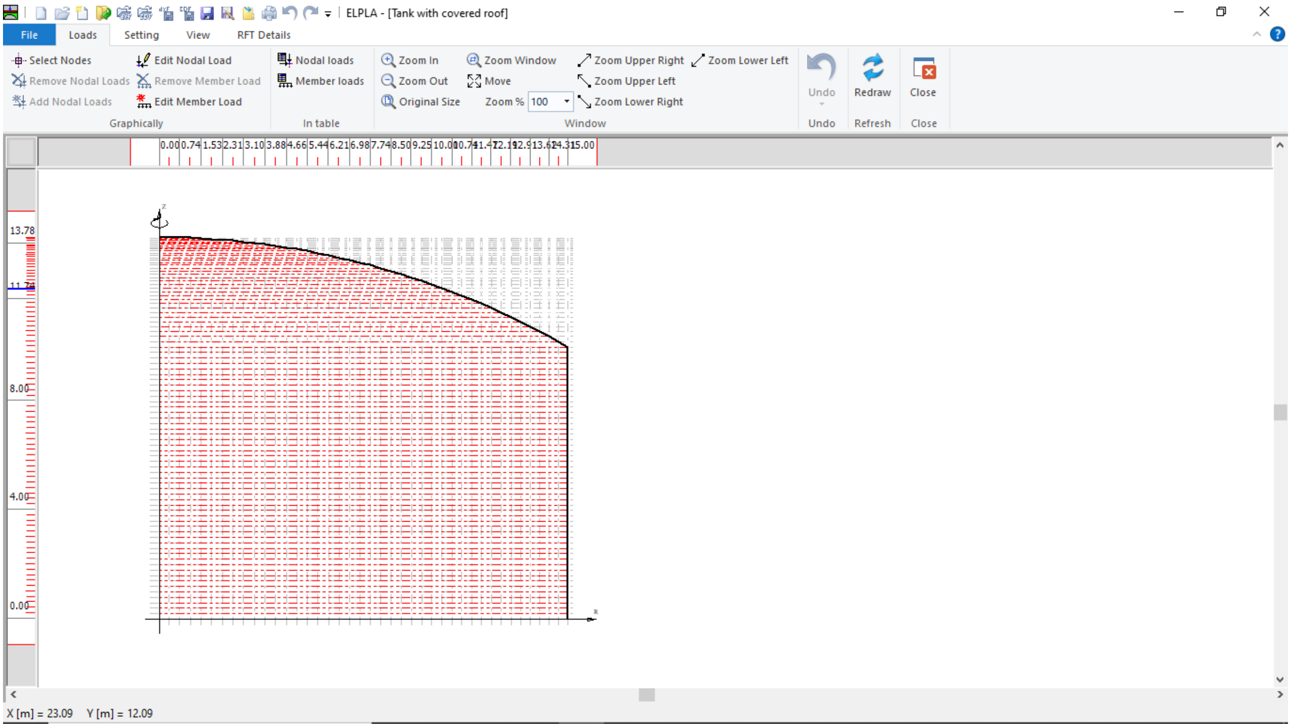


Figure 5.21 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 5.21 to save the load data
- Choose "Close" command from "File" menu in Figure 5.21 to close the "Loads" window and return to *ELPLA* main window

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

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 5.22.

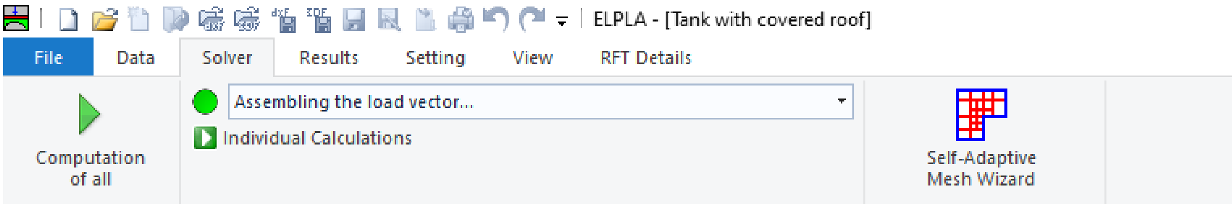


Figure 5.22 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

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

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

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab window.

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

Analysis progress

Analysis progress menu in Figure 5.23 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

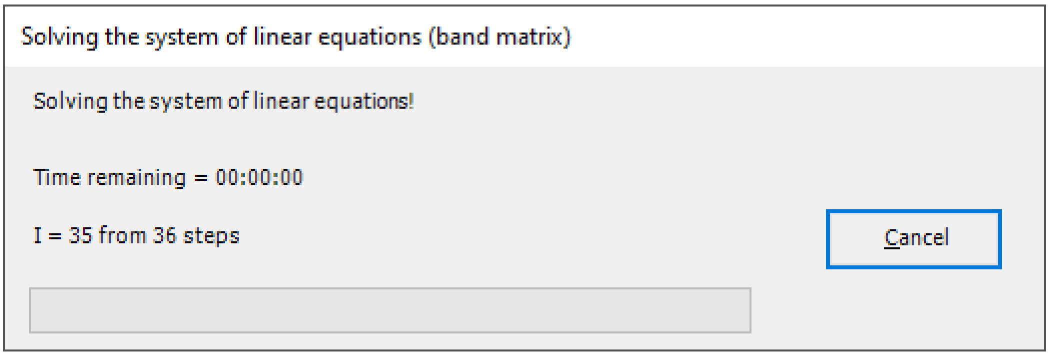


Figure 5.23 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 5.24. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

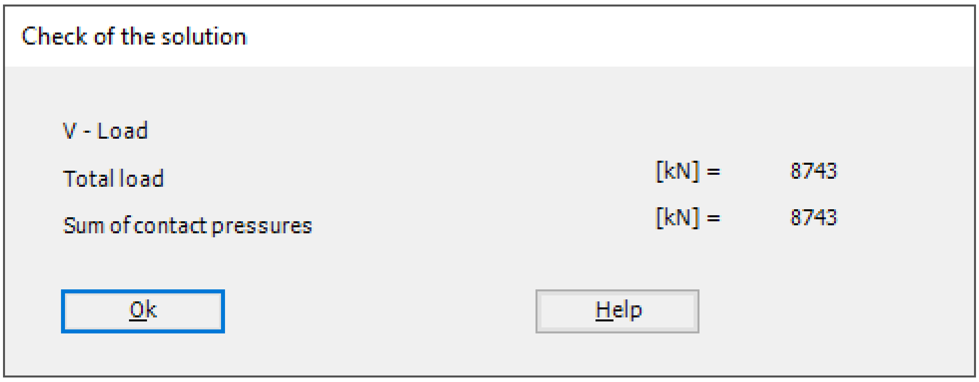


Figure 5.24 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab. To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 5.25).

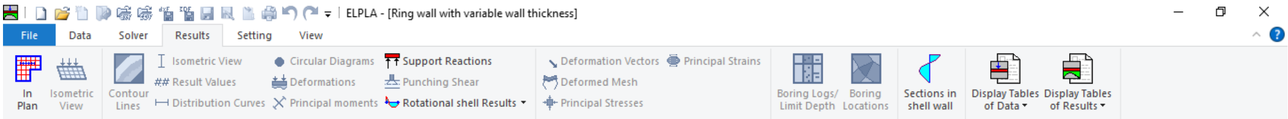


Figure 5.25 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Support Reactions
- Sections in shell wall
- Display tables of data
- Display tables of results

To view element groups of the tank

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 5.25 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

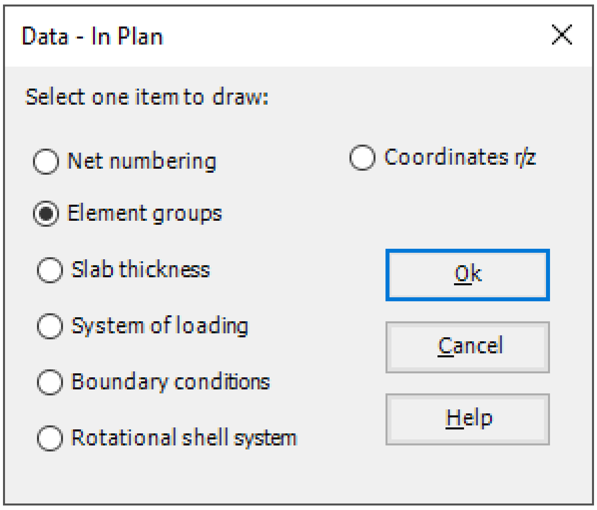


Figure 5.26 "Data – In Plan" option box

To view the supports / boundary conditions on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 5.27 appears
- In this check group box, check "Supports Reactions *Rv*", "Supports Reactions *Rh*", "Supports Reactions *Rm*", "Supports /Boundary Conditions", "Meridional moments *My*" check boxes
- The user can choose any other data to be viewed
- Click "OK" button

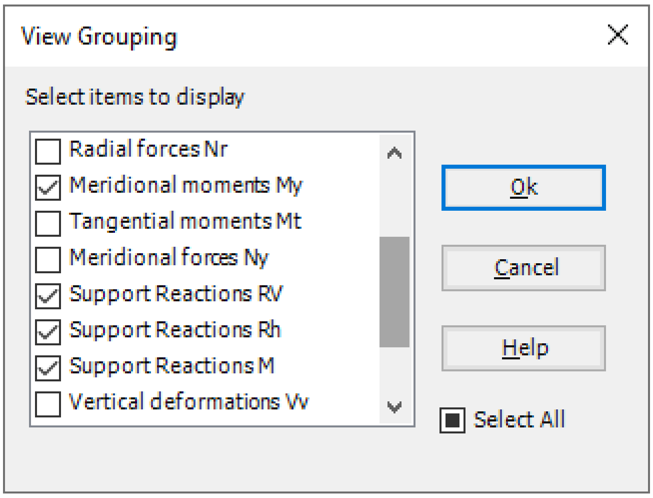


Figure 5.27 "View Grouping" check group box

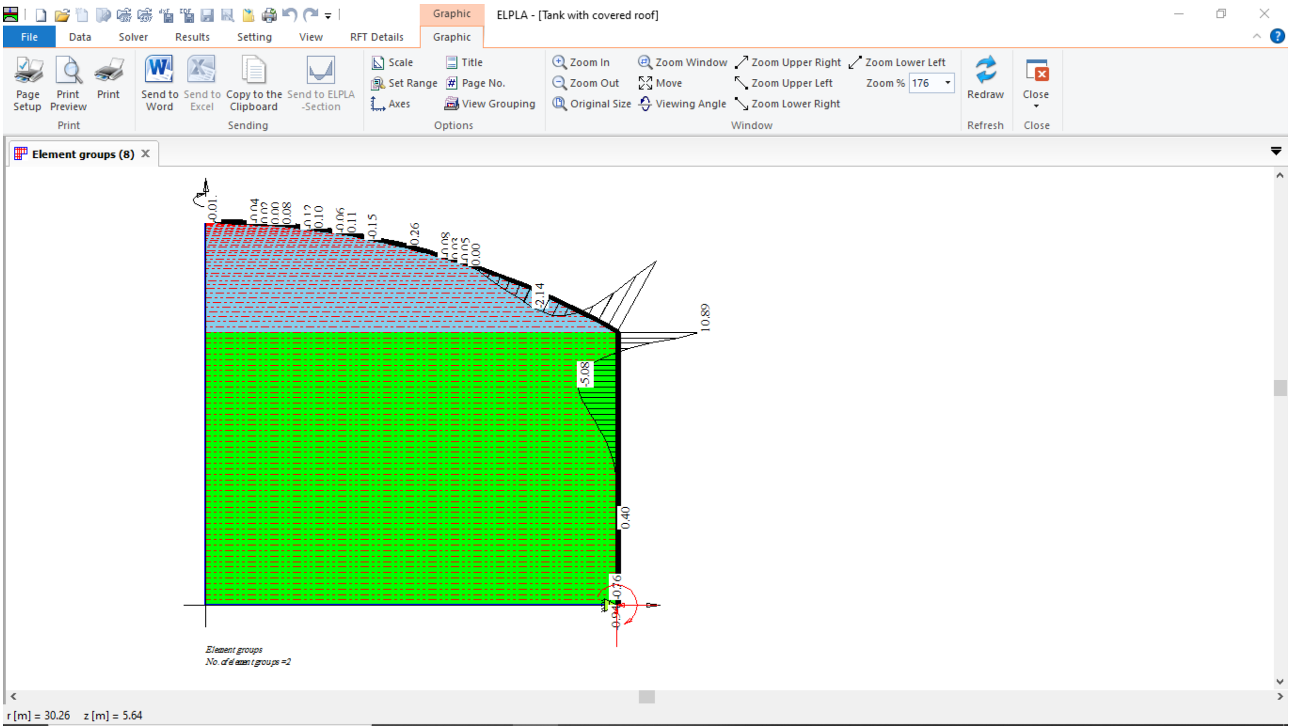


Figure 5.28 Element groups of the tank

Example 6

Analysis of a tank resting on *Winkler's* medium

Contents Page

1 Description of the problem

An example of an axi-symmetrically circular cylindrical tank resting on elastic foundation using *Winkler's* model is selected to illustrate some features of *ELPLA* for analyzing shell elements.

2 Tank geometry and properties

A circular cylindrical tank of an inner diameter of d = 13 [m] and a height of H = 3.5 [m] is considered as shown in Figure 6.1. Thickness of the tank wall is t = 0.175 [m]. The tank is filled with water. The soil under the base of the tank is represented by isolated springs of stiffness k_s, which represent modulus of subgrade reaction. The tank material, unit weight of the water and the modulus of subgrade reaction are listed in Table 6.1.

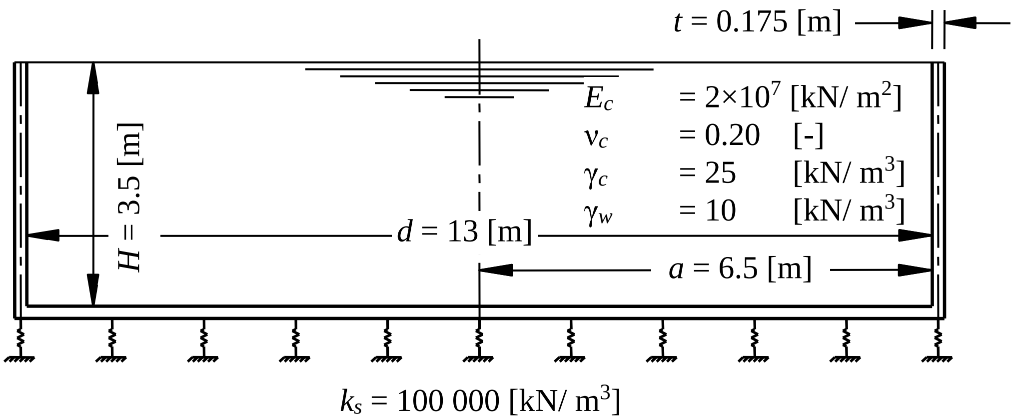


Figure 6.1 Circular cylindrical tank resting on isolated springs with dimensions

Table 6.1 Tank material, water unit weight and modulus of subgrade reaction

Modulus of Elasticity of the tank material	E_c	= 2×10^7	[kN/ m^2]
Poisson's ratio of the tank material	ν_c	= 0.2	[-]
Unit weight of the tank material	γ_c	= 25	[kN/ m^3]
Unit weight of the water	γ_w	= 10	[kN/ m^3]
Modulus of subgrade reaction	k_s	= 100 000	[kN/ m^3]

3 Numerical Analysis

In order to analyze the tank, the height of the tank is divided into 35 equal segments, each of 0.10 [m], as shown in Figure 6.2. The base of the tank is divided into 50 equal segments, each of 0.13 [m].

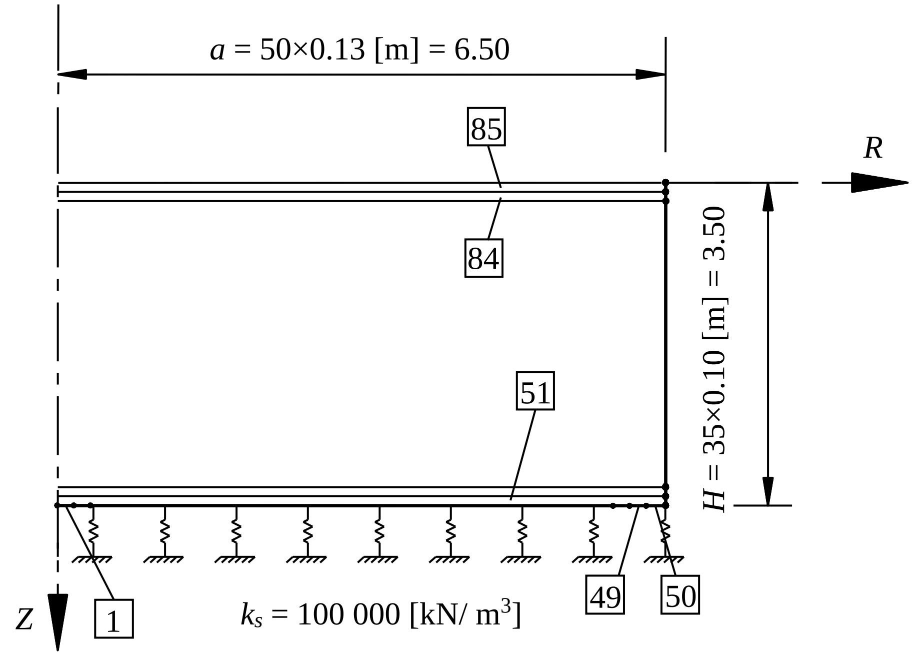


Figure 6.2 Finite element mesh of the tank

4 Creating the project

In this section, the user will learn how to create a project for analyzing an axi-symmetrically circular cylindrical tank resting on elastic foundation using *Winkler's* model. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 6.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of Forms. The first Form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 6.3).

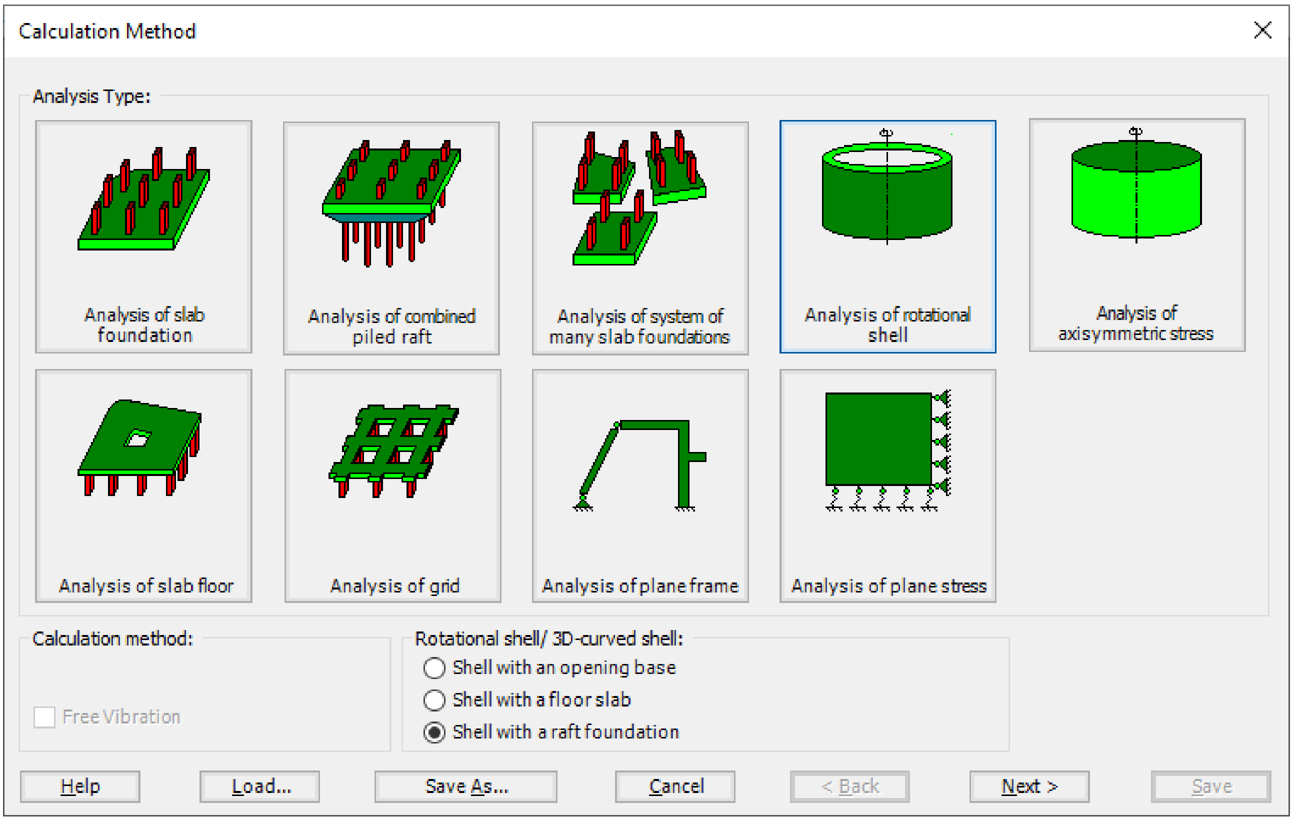


Figure 6.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 6.3, define the analysis type of the problem. As the analysis type is a circular cylindrical tank problem, select "Analysis of rotational Shell" button, and check "Shell with a raft foundation" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Calculation Methods" Form appears, Figure 6.4.

To define the calculation method:

- Select the calculation method "2/3 Constant/ Variable Modulus of Subgrade Reaction"
- To determine the modulus of subgrade reaction, select "Modulus is defined by the user"
- Click "Next" button to go to the next Form

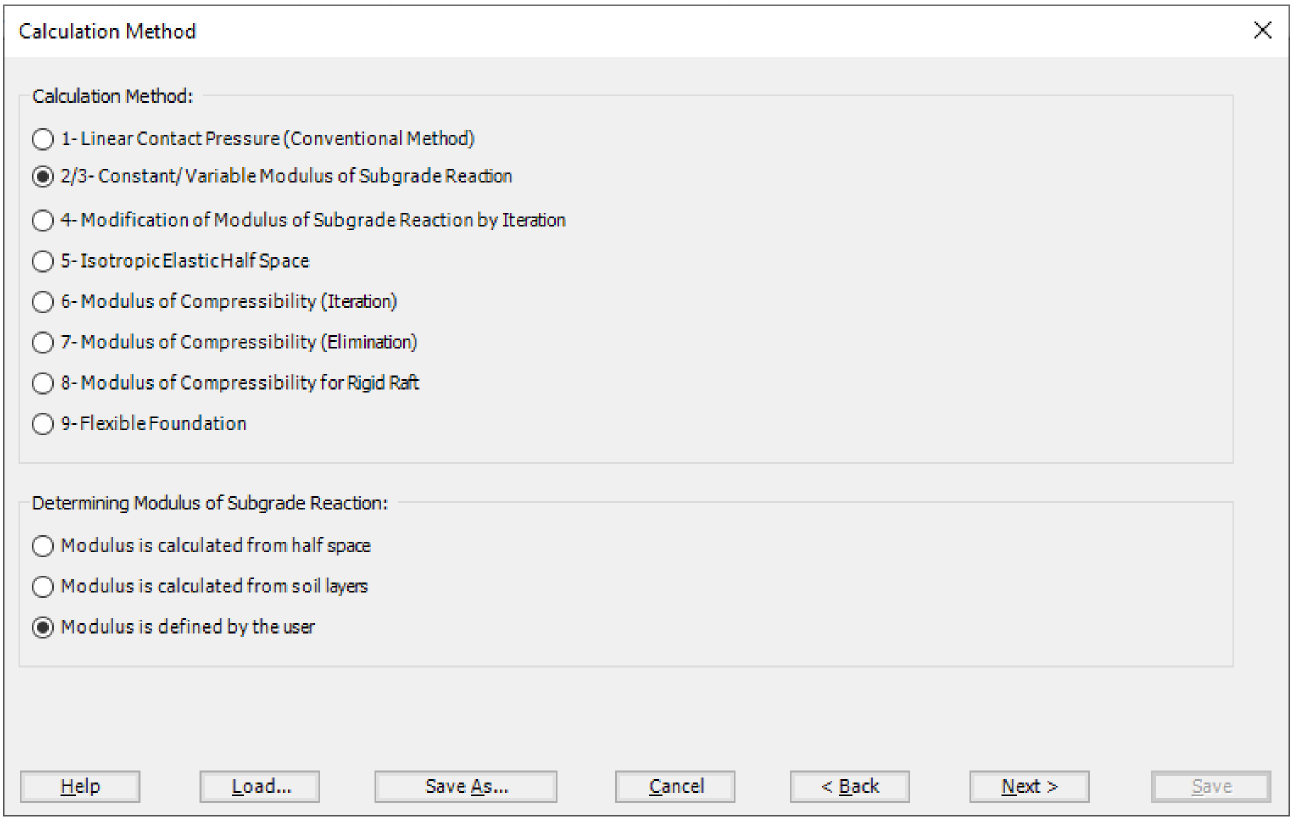


Figure 6.4 "Calculation Method" Form

The last Form in the wizard is the "Options" Form, Figure 6.5. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Since no option will be considered in the analysis, click the "Save" button.

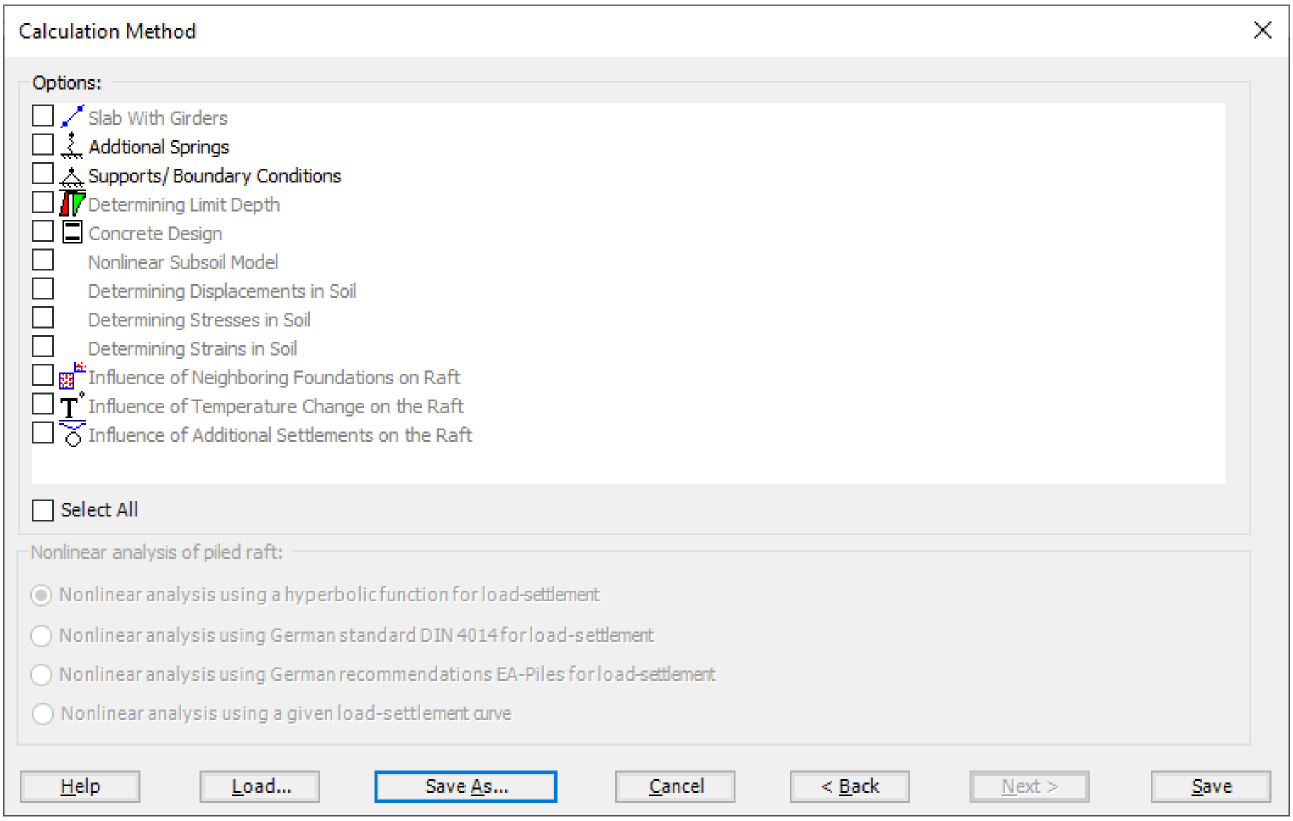


Figure 6.5 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 6.6. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Tank on Winkler's medium". *ELPLA* will use automatically this file name in all reading and writing processes.

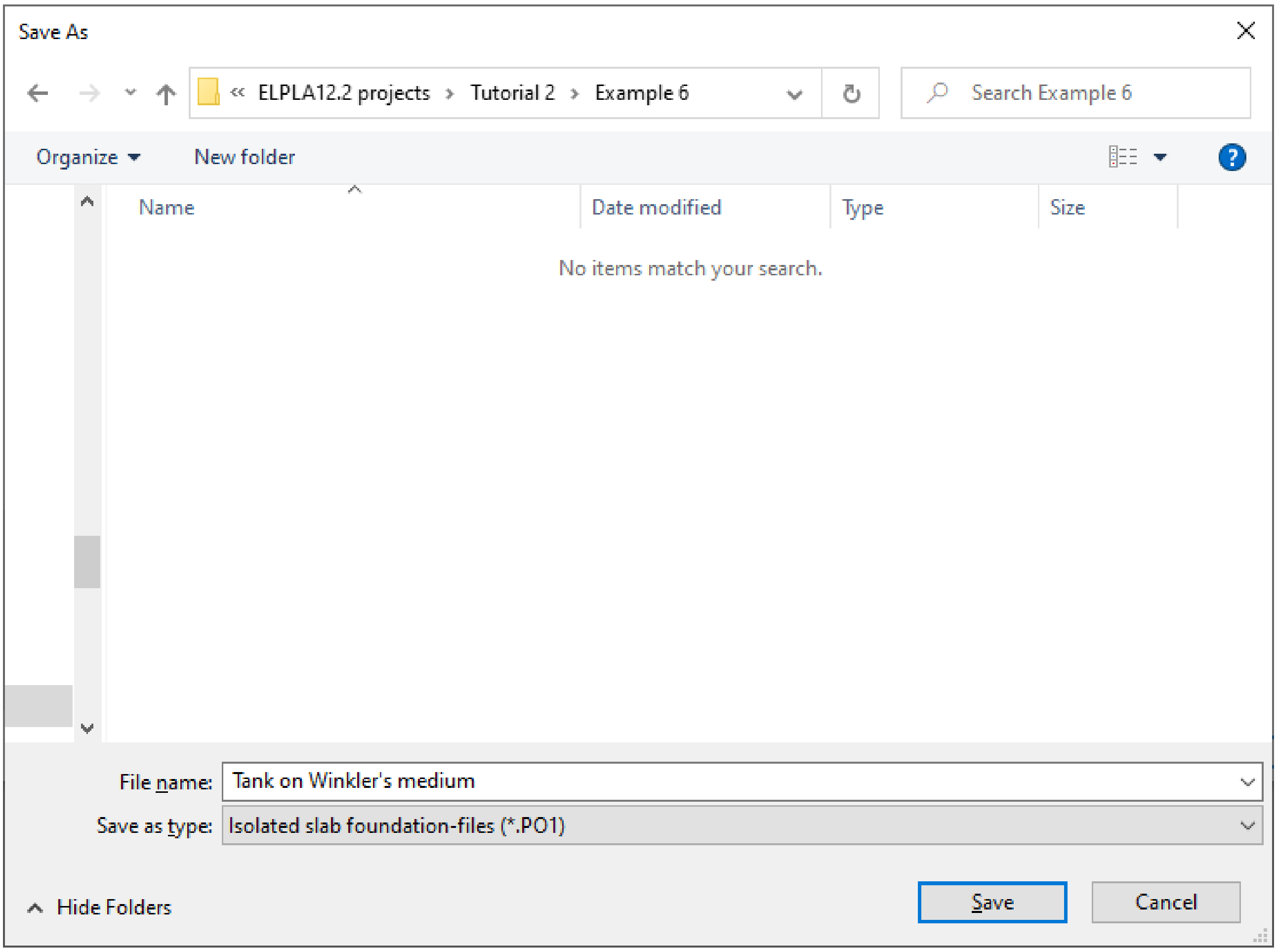


Figure 6.6 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Tank on *Winkler's* medium] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 6.7 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box:
  "Analysis of a tank resting on *Winkler's* medium"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

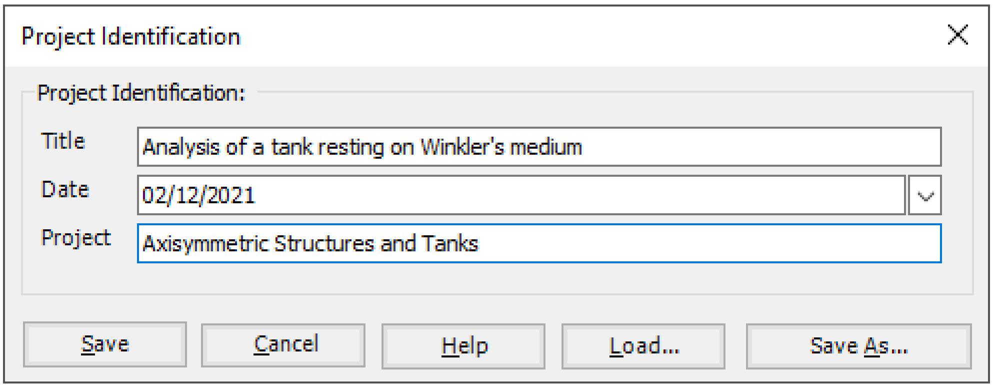


Figure 6.7 "Project Identification" dialog box

4.3 FE-Net data

For the given problem, the tank has an inner diameter of d = 13 [m] and a height of H = 3.5 [m]. the tank height is divided into 35 equal segments, each of 0.10 [m], where the base of the tank is divided into 50 equal segments, each of 0.13 [m]. To define the FE-Net for this tank, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 6.8. This wizard will guide you through the steps required to generate a FE-Net.

The first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

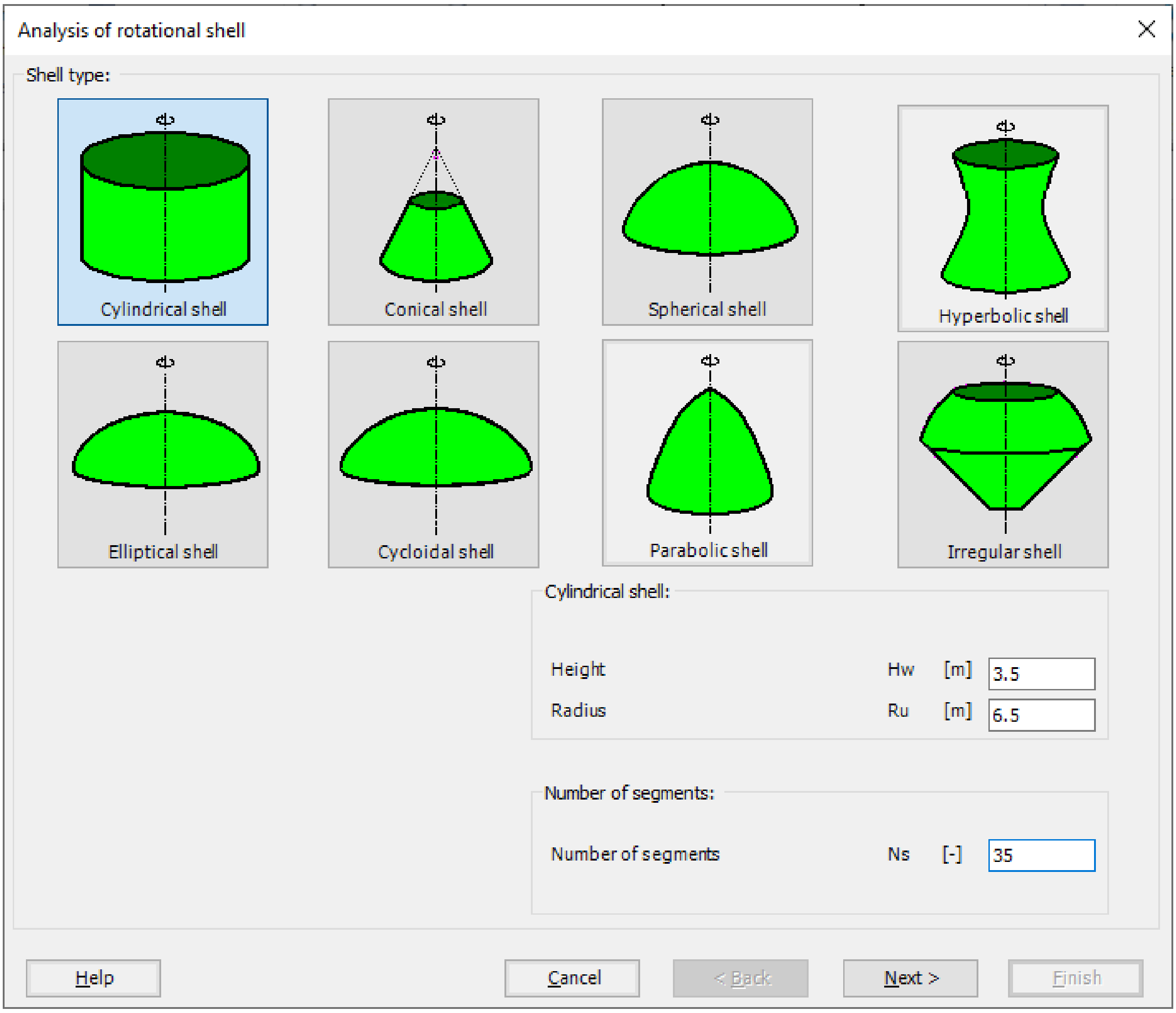


Figure 6.8 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Cylindrical shell" button
- Type 3.5 in the "Height" edit box
- Type 6.5 in the "Radius" edit box
- Type 35 in the "Number of segments" edit box
- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Cylindrical shell" Form appears Figure 6.9, *ELPLA* divides the height of the tank into 35 equal segments, the user can edit the data of the segments individually by using "Modify" button, or all of them by using "In Table" button, if it is necessary.

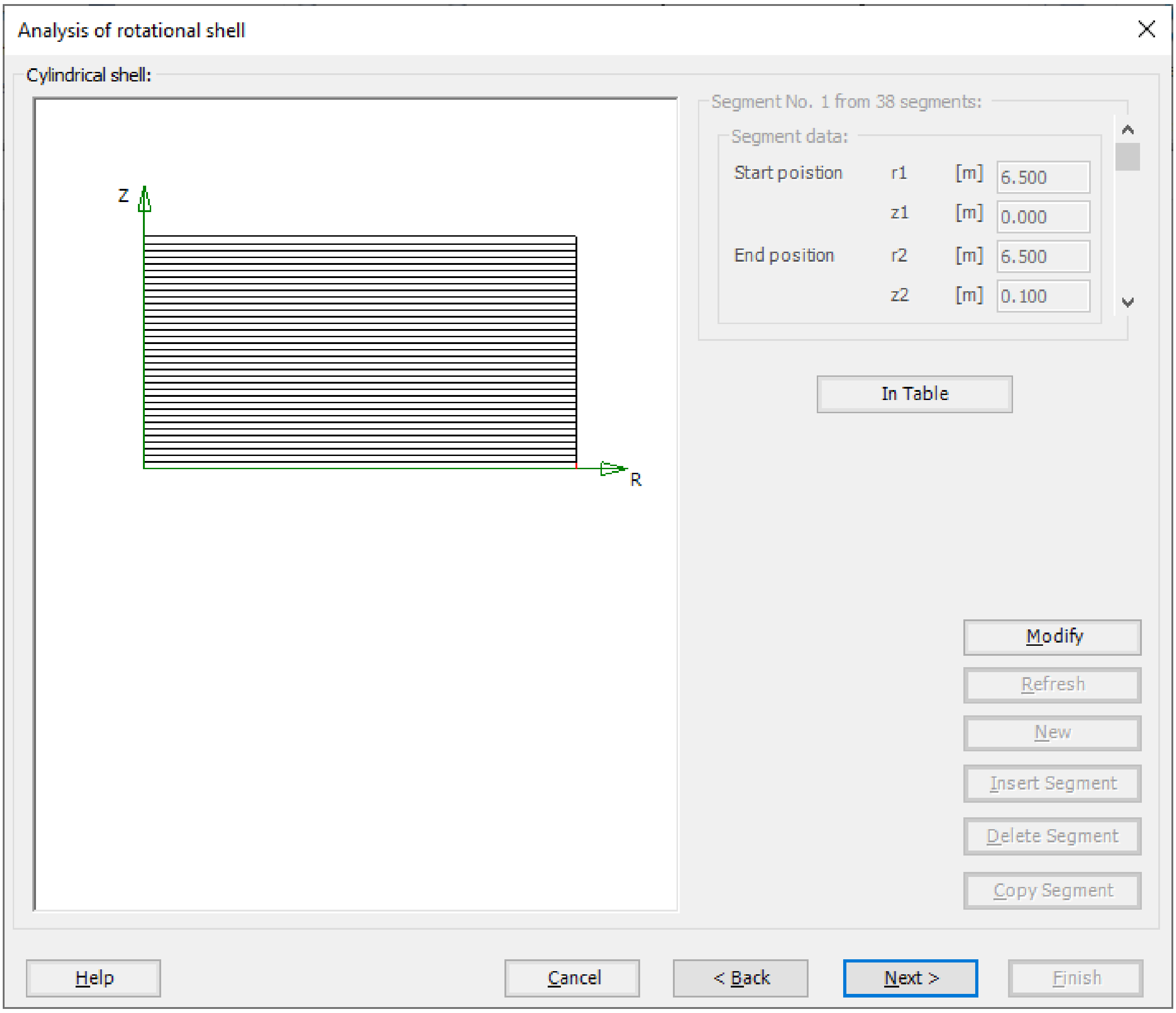


Figure 6.9 "Cylindrical shell" Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Net of Base" Form appears Figure 6.10.

To edit the grid spacing in *x*-direction, do the following steps in "Grid in *x*-direction" frame:

- Choose "Constant grid interval" check box
- Type 50 in the "No. of grid intervals" edit box

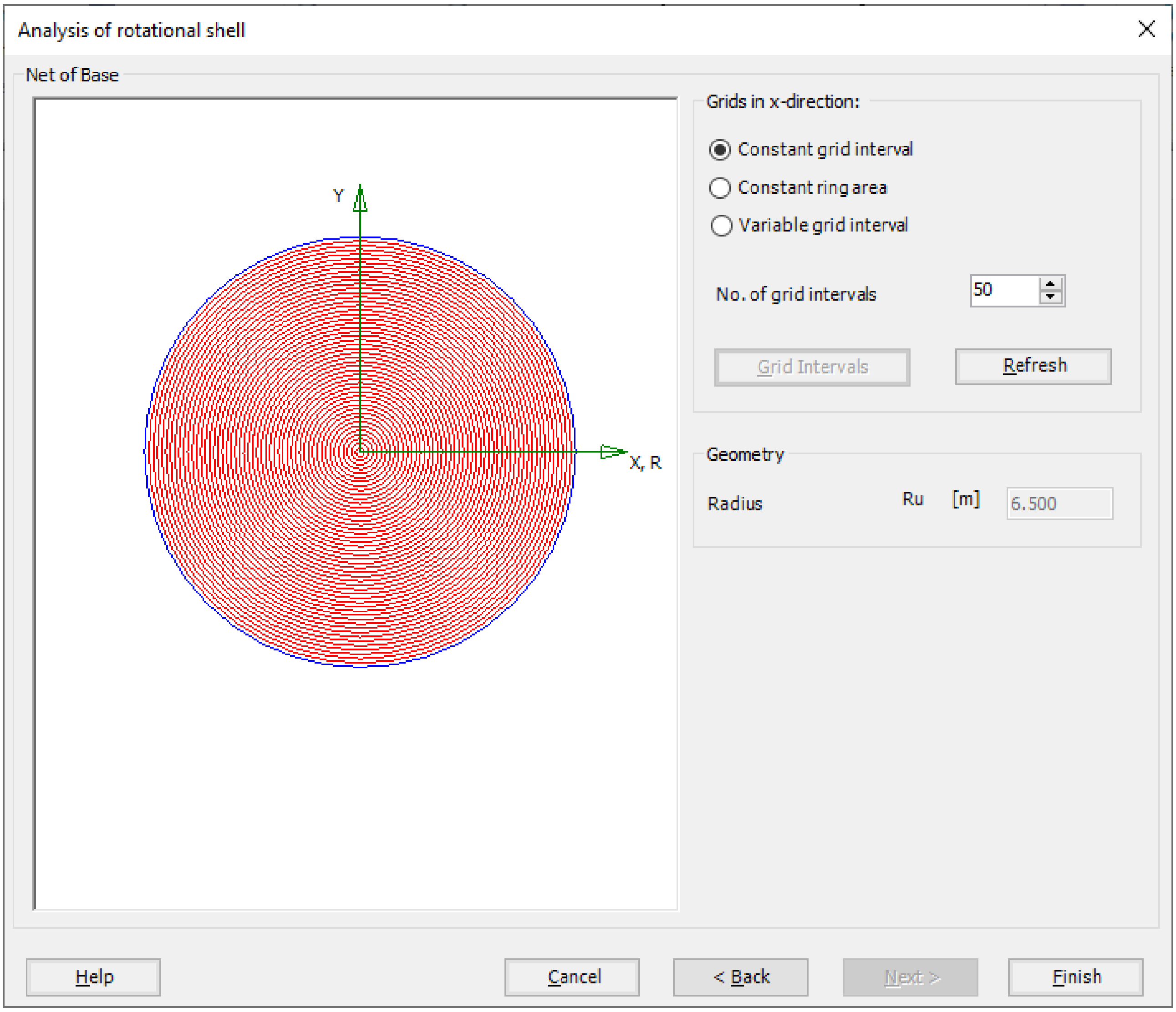


Figure 6.10 "Net of Base" Form

Click "Finish" button, the FE-Net of the tank wall and a sector from the base appear in Figure 6.11

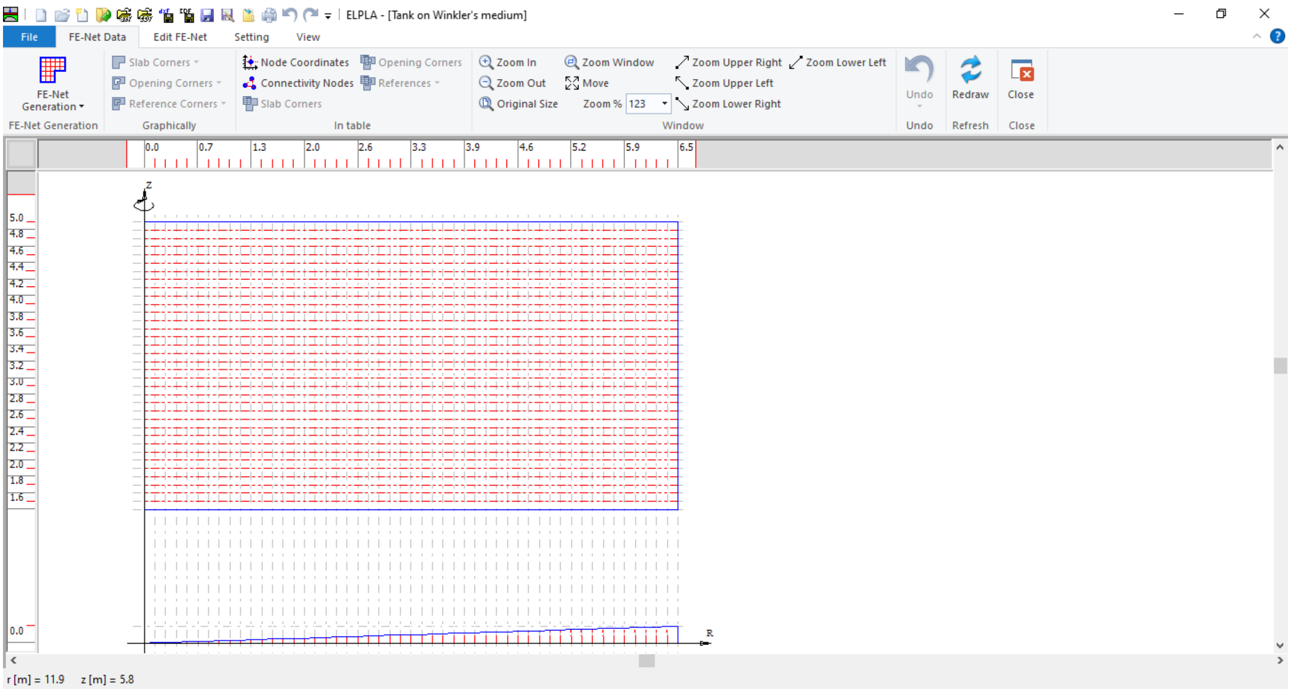


Figure 6.11 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 6.11 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 6.11 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Shell properties

To define the tank properties, choose "Shell Properties" command from "Data" Tab. The following window in Figure 6.12 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, unit weight of the tank, and filled material properties.

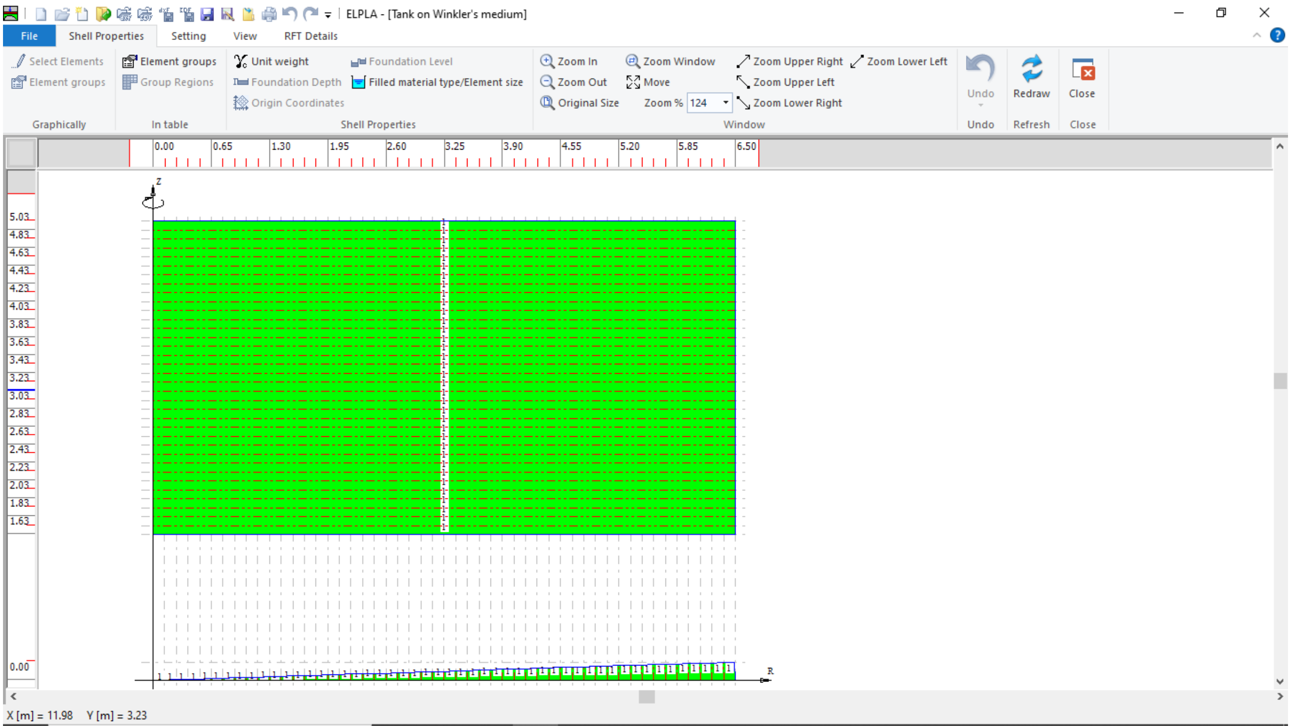


Figure 6.12 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 6.13 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness. Then click "OK" button.

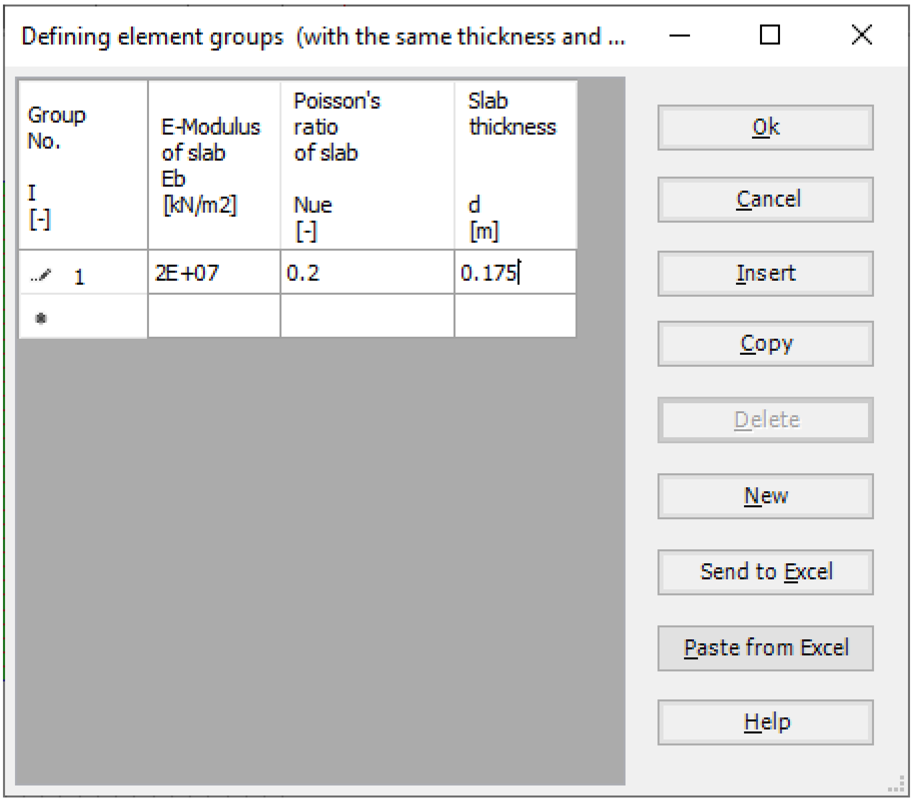


Figure 6.13 "Defining element groups" list box

To enter the unit weight of the tank, choose "Unit weight" command from "Shell Properties" menu in Figure 6.12. The following dialog box in Figure 6.14 with a default unit weight of 25 [kN/m^3] appears, click "OK" button.

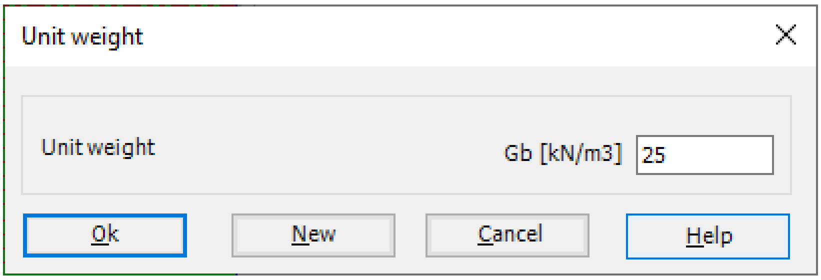


Figure 6.14 "Unit weight" dialog box

To define the liquid properties of the shell, choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 6.12. The following form in Figure 6.15 appears.

To define the liquid properties of the tank:

- Select the "Liquid container" option
- Type 3.5 in the "Height of the liquid" edit box
- Type 10 in the "Unit weight of the liquid" edit box

To define the element size of the tank:

- Check the "Constant element sizes in z-direction" check box
- Type 1 in the "Element size in each shell segment" edit box. The element size is chosen to be 1 [m] larger than the segment size in order to ignore further subdivision of the segments into smaller elements. In some cases, it is necessary to divide the segment into smaller elements in order to make the analysis more precise. Nevertheless, the final results of the internal forces appear only at nodes of segments
- Click "OK" button

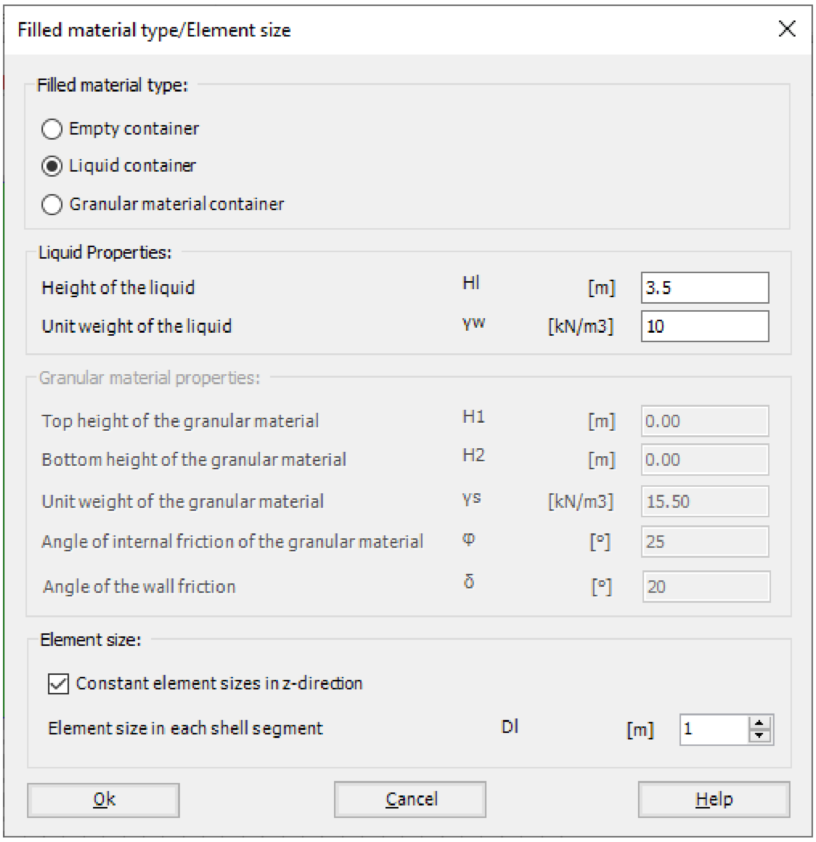


Figure 6.15 "Liquid properties/Element size" Form

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

- Choose "Save" command from "File" menu in Figure 6.12 to save the shell properties
- Choose "Close" command from "File" menu in Figure 6.12 to close the "Shell properties" window and return to *ELPLA* main window

4.5 Soil Properties

To define the soil properties, choose "Soil Properties" command from “Data” Tab. The following "Soil Properties" form in Figure 6.16 appears, enter the modulus of subgrade reaction of the soil and the ground water depth under the ground surface. Other data for this example is not required.

Soil data

Modulus of subgrade reaction	k_s	= 100 000	[kN/ m^3]
Ground water depth under the surface	Gw	= 1	[m]

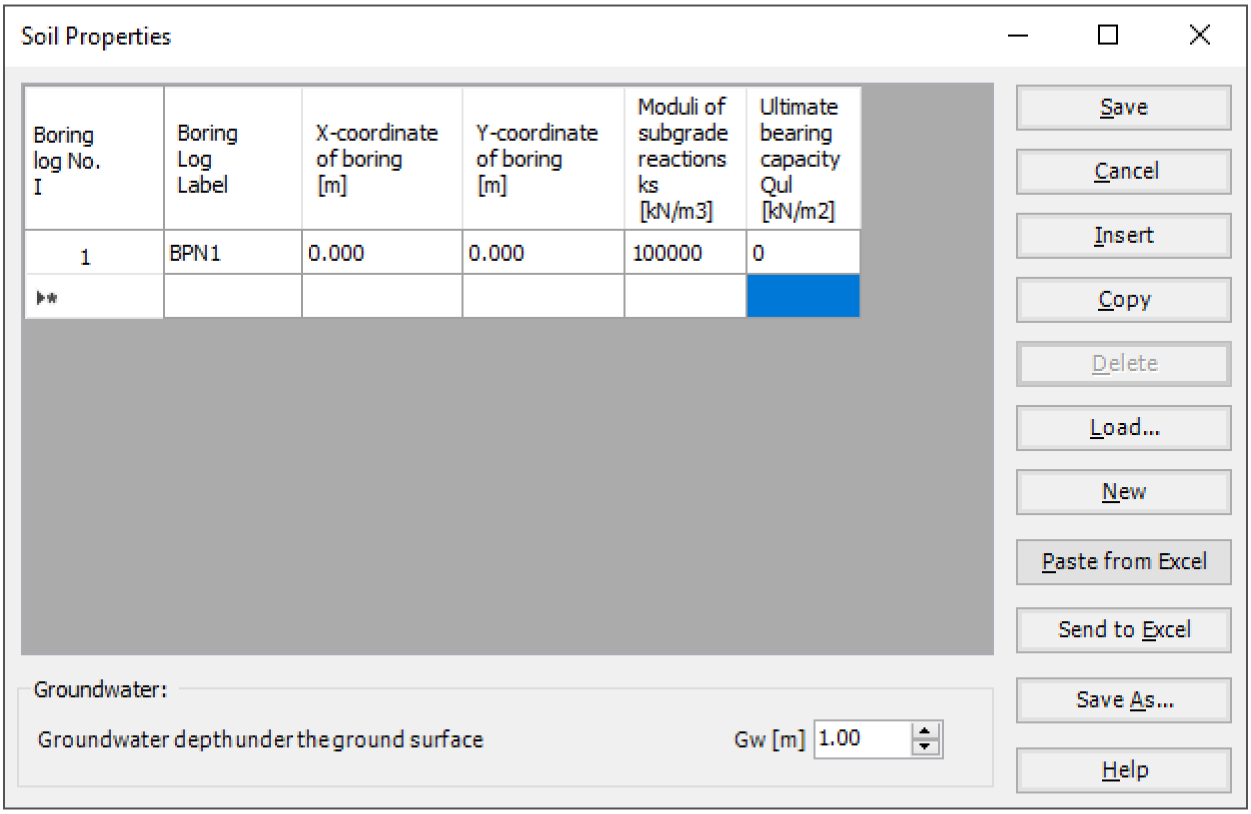


Figure 6.16 "Soil Properties" Form

4.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 6.17 appears. In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" tab in Figure 6.17. In this example, there is not applied load, as the vertical load has been already defined by the unit weight of the tank, while the hydrostatic pressure on the tank wall is defined by the unit weight of water.

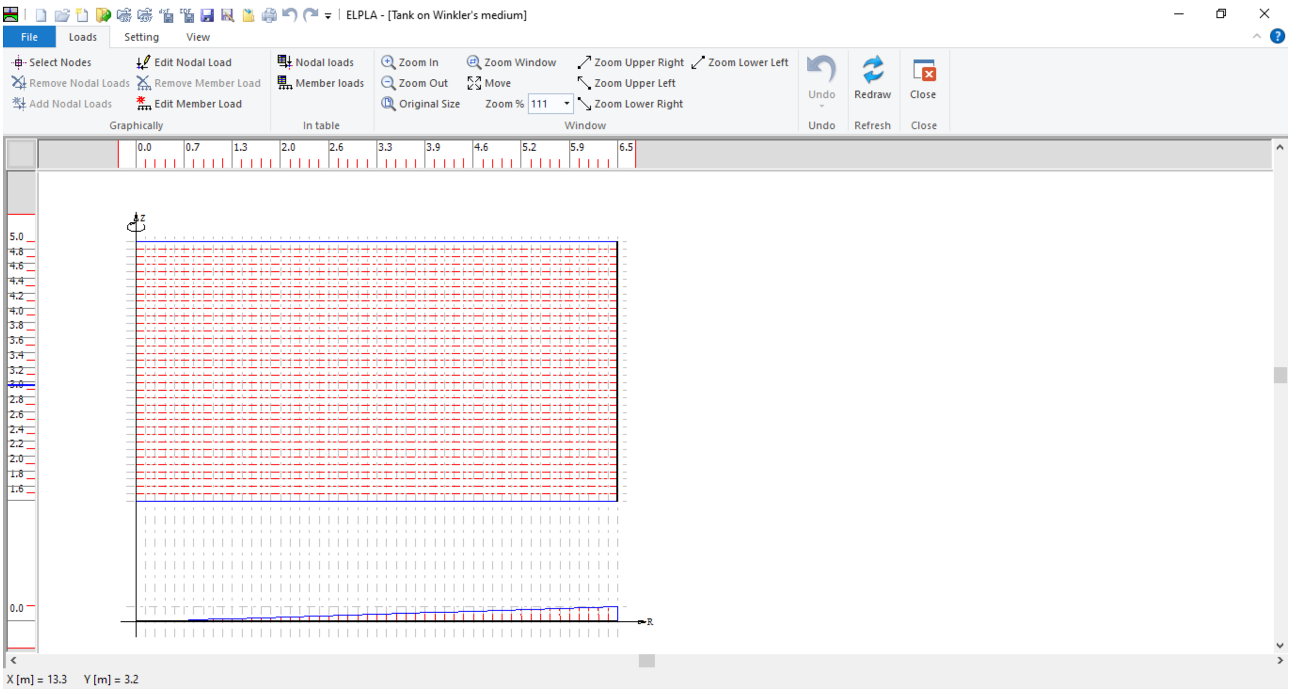


Figure 6.17 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 6.17 to save the load data
- Choose "Close" command from "File" menu in Figure 6.17 to close the "Loads" window and return to *ELPLA* main window

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

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 6.18.

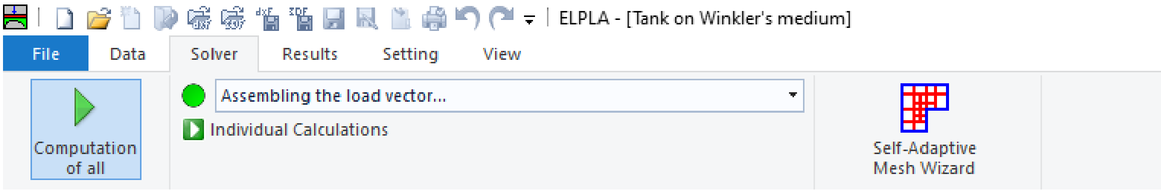


Figure 6.18 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

- Assembling the load vector
- Determining the modulus of subgrade reaction
- Assembling the slab stiffness matrix
- Solving the system of linear equations (band matrix)
- Determining deformation, internal forces, contact pressures

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

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab Window.

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

Analysis progress

Analysis progress menu in Figure 6.19 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

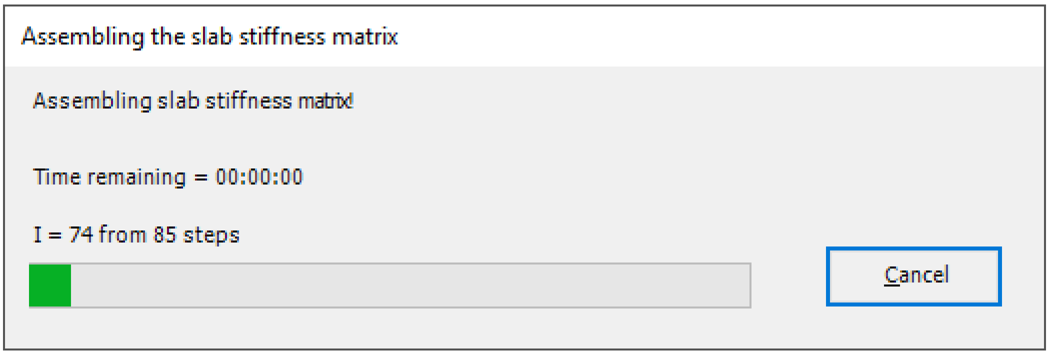


Figure 6.19 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 6.20. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

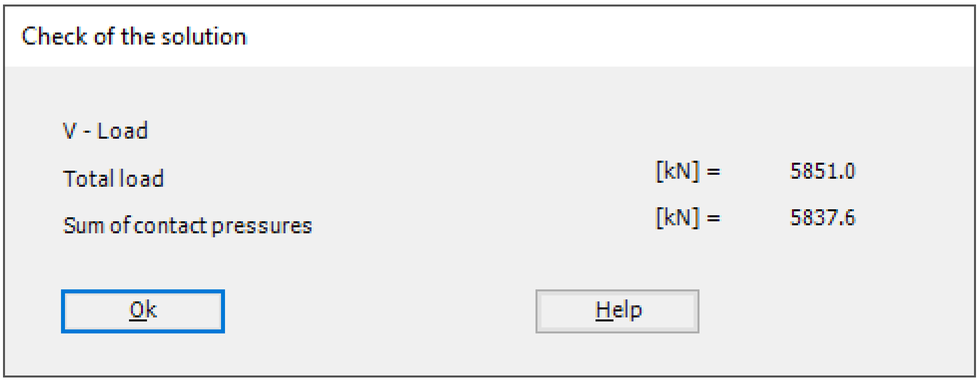


Figure 6.20 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab.

To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 6.21).

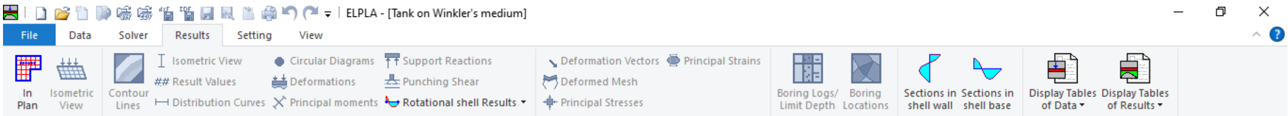


Figure 6.21 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Sections in shell wall
- Sections in shell base
- Display tables of data
- Display tables of results

To view the radial forces in the shell wall

- Choose "Sections in shell wall" command from "Section" menu. The following option box in Figure 6.22 appears
- In the "Sections in shell wall" option box, select "Radial forces *Nr*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 6.23.

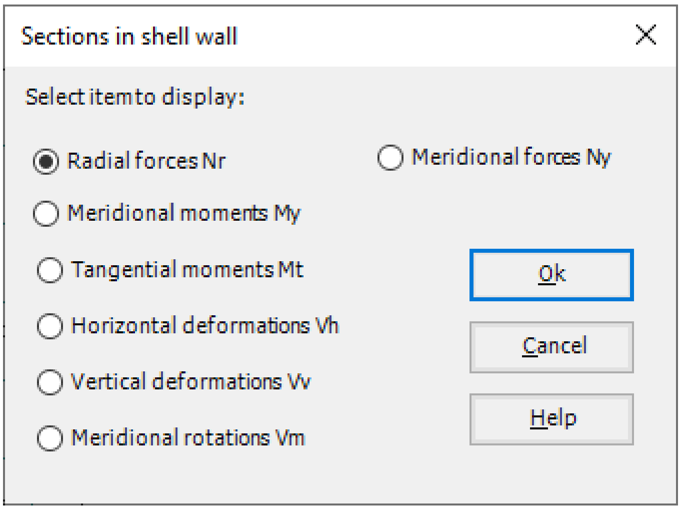


Figure 6.22 "Sections in shell wall" option box

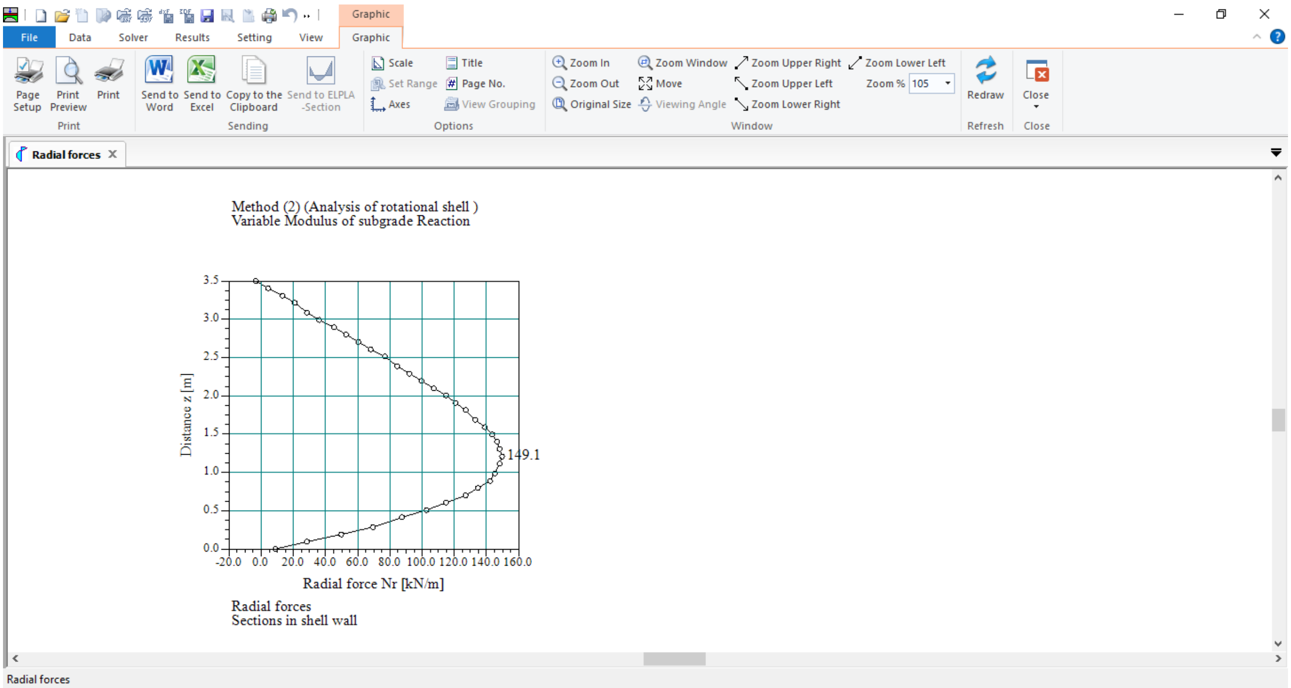


Figure 6.23 Radial forces in shell wall

To view the meridional moments on the shell wall

- From "Rotational shell results" command in the "Results" menu, choose "In Plan" command, the following option box in Figure 6.24 appears
- In the "Distribution of Internal Forces" option box, select "Meridional moments *My*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 6.23.

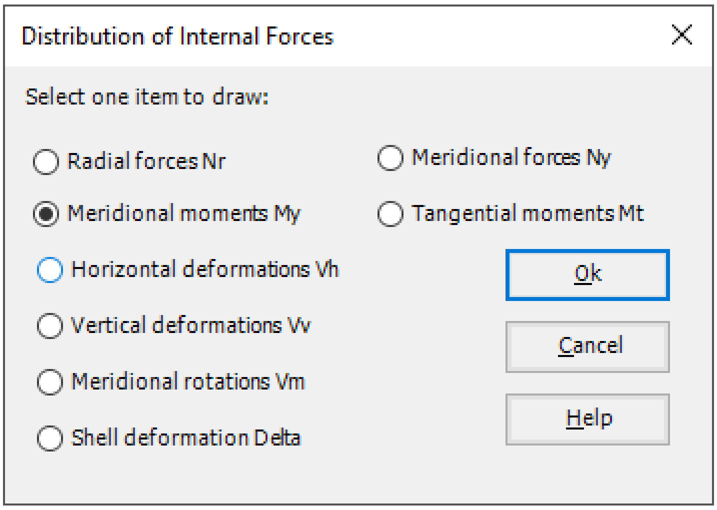


Figure 6.24 "Distribution of Internal Forces" option box

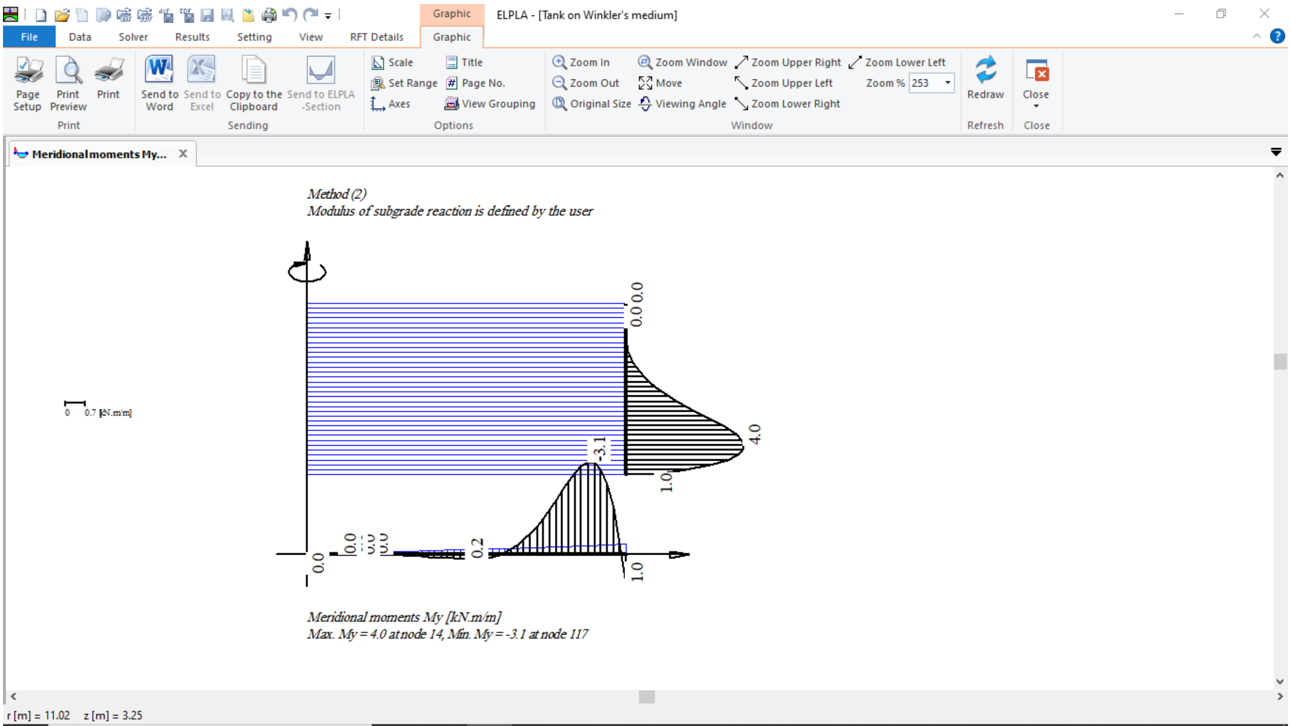


Figure 6.25 Meridional moments on the shell wall

Example 7

Analysis of a tank with a conical base resting on *Winkler's* medium

Contents Page

1 Description of the problem

An example of a circular cylindrical tank with a conical base resting on elastic foundation using *Winkler's* model is selected to illustrate some features of *ELPLA* for analyzing circular cylindrical shell elements.

2 Loads

A circular cylindrical tank of an inner diameter of d = 15 [m] and a height of H = 6 [m] is considered as shown in Figure 7.1. Thickness of the tank wall is t = 0.5 [m]. The tank is filled with water. The soil under the base of the tank is represented by isolated springs of stiffness k_s, which represent modulus of subgrade reaction. Figure 7.1 shows the tank with dimensions, while the tank material, unit weight of the water and the modulus of subgrade reaction are listed in Table 7.1.

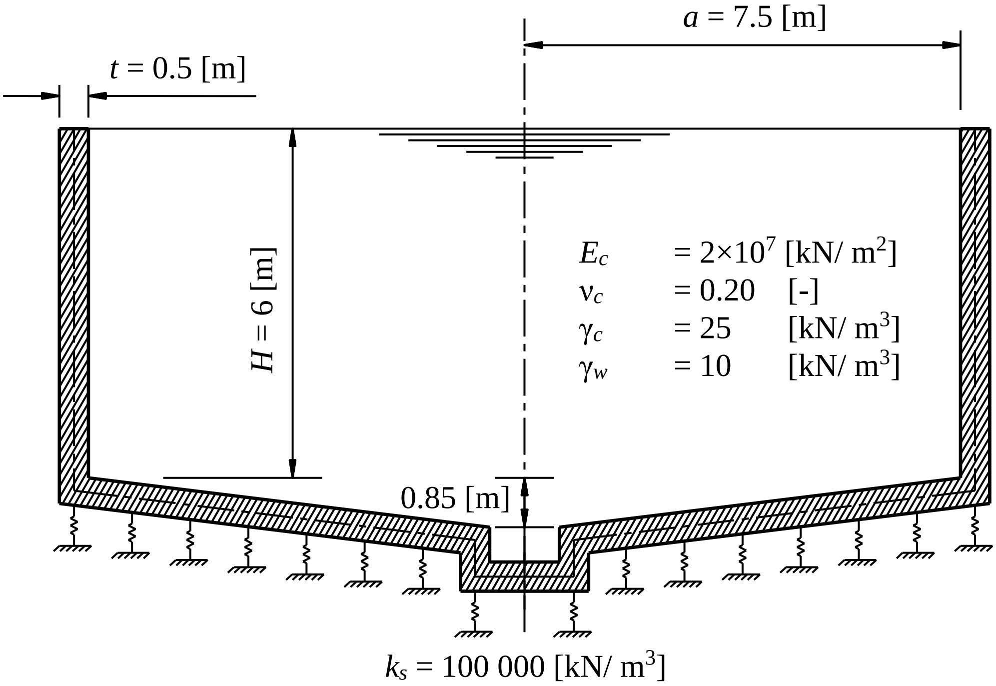


Figure 7.1 Circular cylindrical tank on isolated springs with dimensions

Table 7.1 Tank material, water unit weight and modulus of subgrade reaction

Modulus of Elasticity of the tank material	E_c	= 2×10^7	[kN/ m^2]
Poisson's ratio of the tank material	ν_c	= 0.2	[-]
Unit weight of the tank material	γ_c	= 25	[kN/ m^3]
Unit weight of the water	γ_w	= 10	[kN/ m^3]
Modulus of subgrade reaction	k_s	= 100 000	[kN/ m^3]

3 Numerical Analysis

In the analysis, the height of the tank is divided into 20 equal elements, each of 0.30 [m], as shown in Figure 7.2 . The conical base of the tank is divided into 17 equal elements.

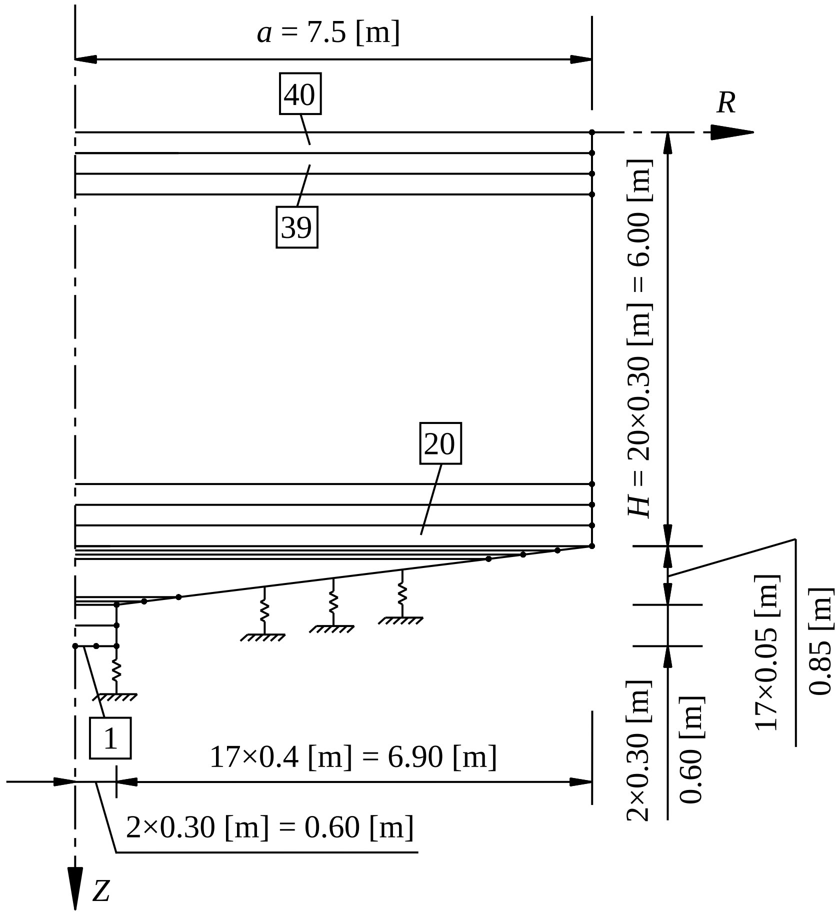


Figure 7.2 Finite element mesh of the tank

4 Creating the project

In this section, the user will learn how to create a project for analyzing an axi-symmetrically circular cylindrical tank resting on elastic foundation using *Winkler's* model. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 7.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of Forms. The first Form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 7.3).

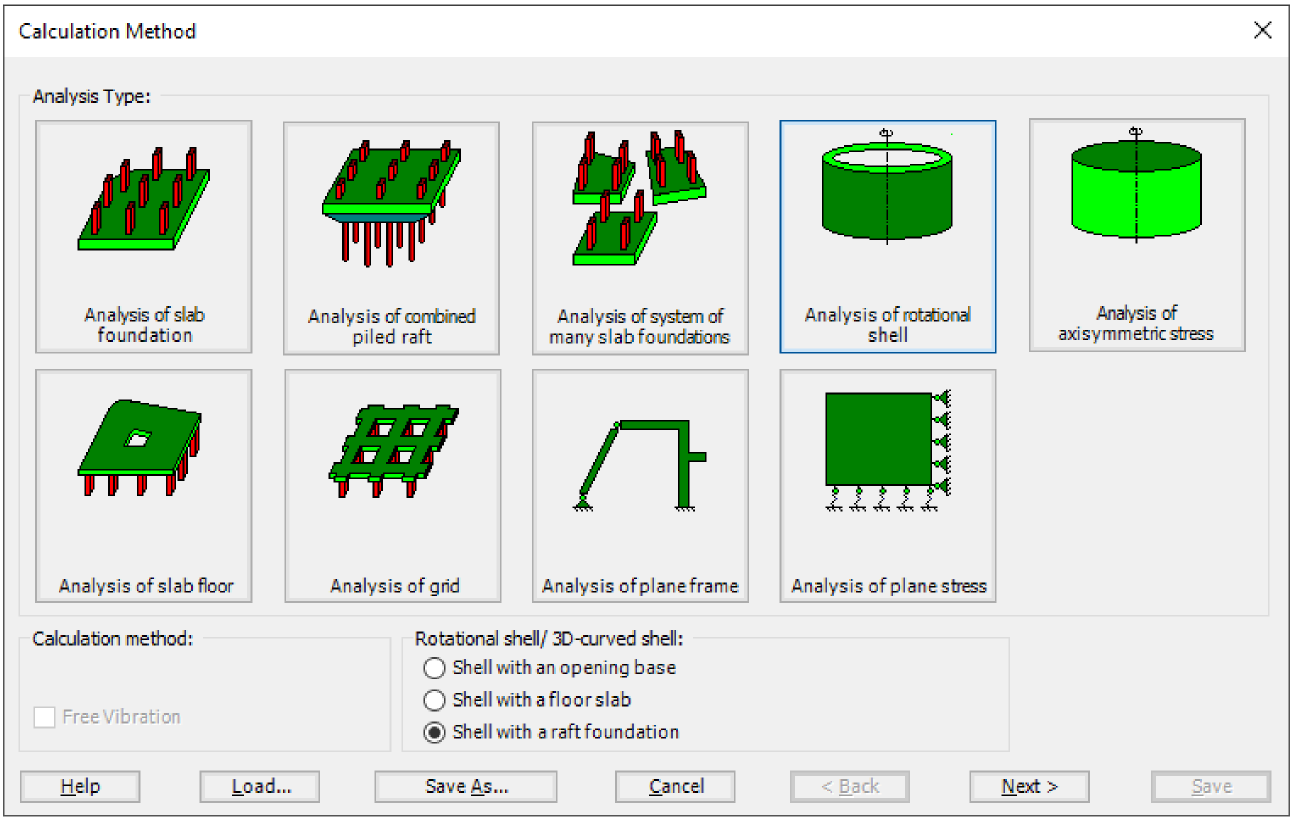


Figure 7.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 7.3, as the analysis type is a circular cylindrical tank problem, select "Analysis of rotational Shell" button and check "Shell with a raft foundation" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Calculation Methods" Form appears, Figure 7.4.

To define the calculation method:

- Select the calculation method "2/3 Constant/ Variable Modulus of Subgrade Reaction"
- To determine the modulus of subgrade reaction, select "Modulus is defined by the user"
- Click "Next" button to go to the next Form.

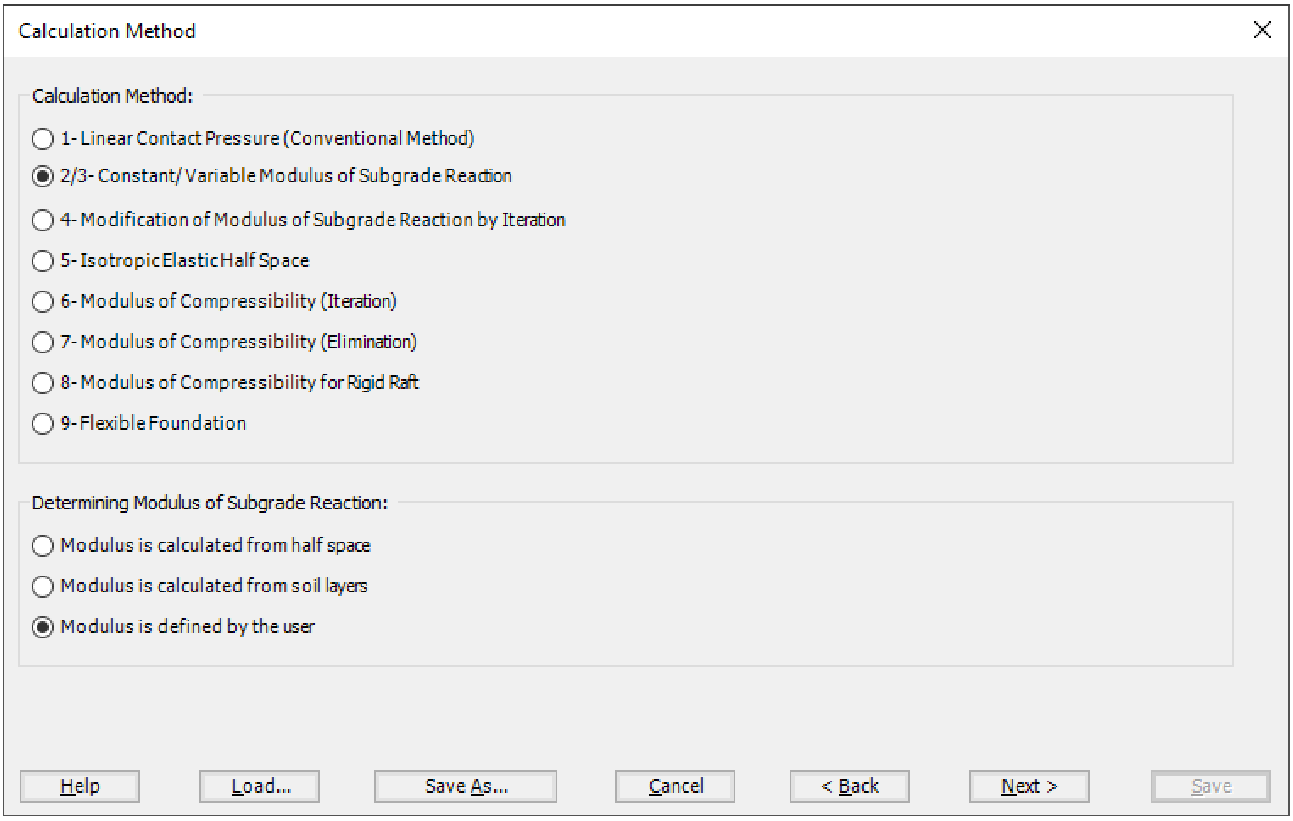


Figure 7.4 "Calculation Method" Form

The last Form in the wizard is the "Options" Form, Figure 7.5. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Select "Additional Springs", then click the "Save" button.

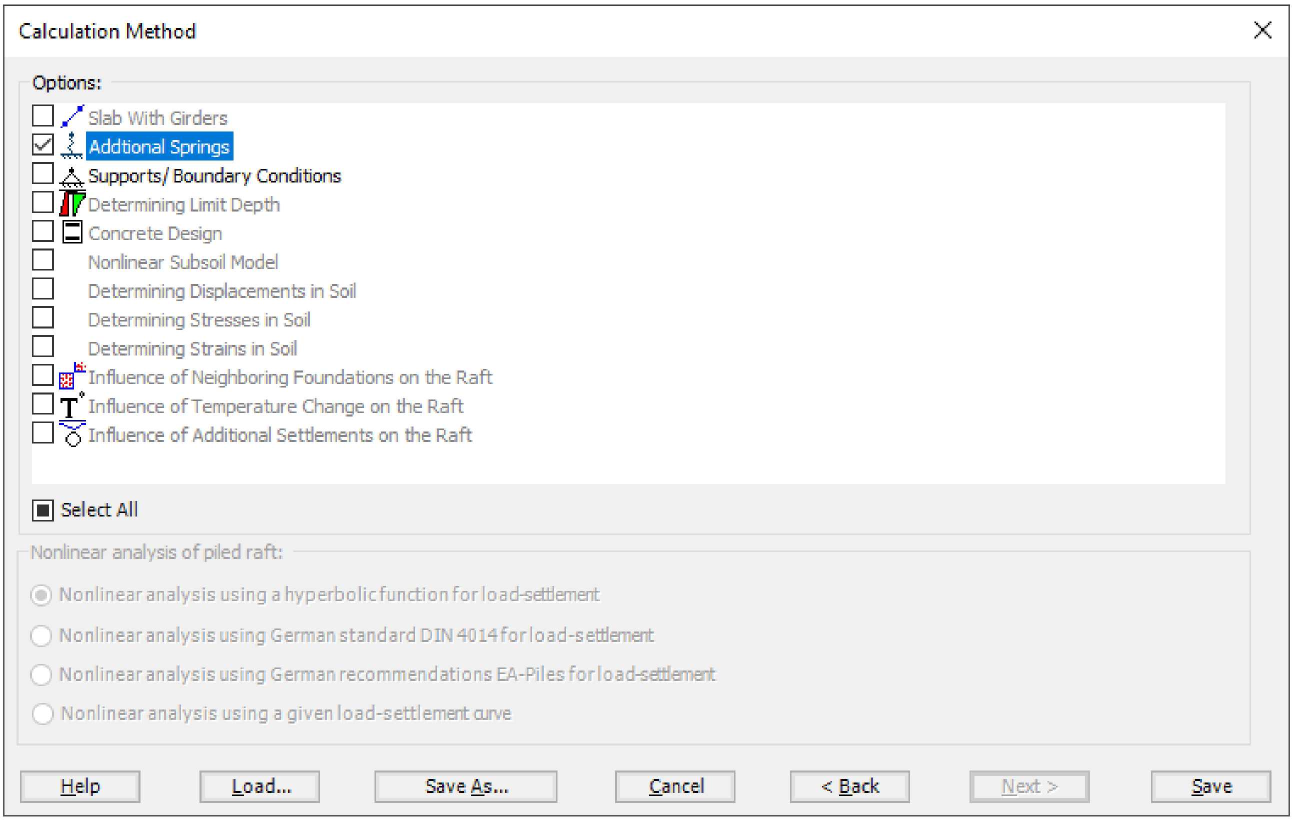


Figure 7.5 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 7.6. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Conical Tank". *ELPLA* will use automatically this file name in all reading and writing processes.

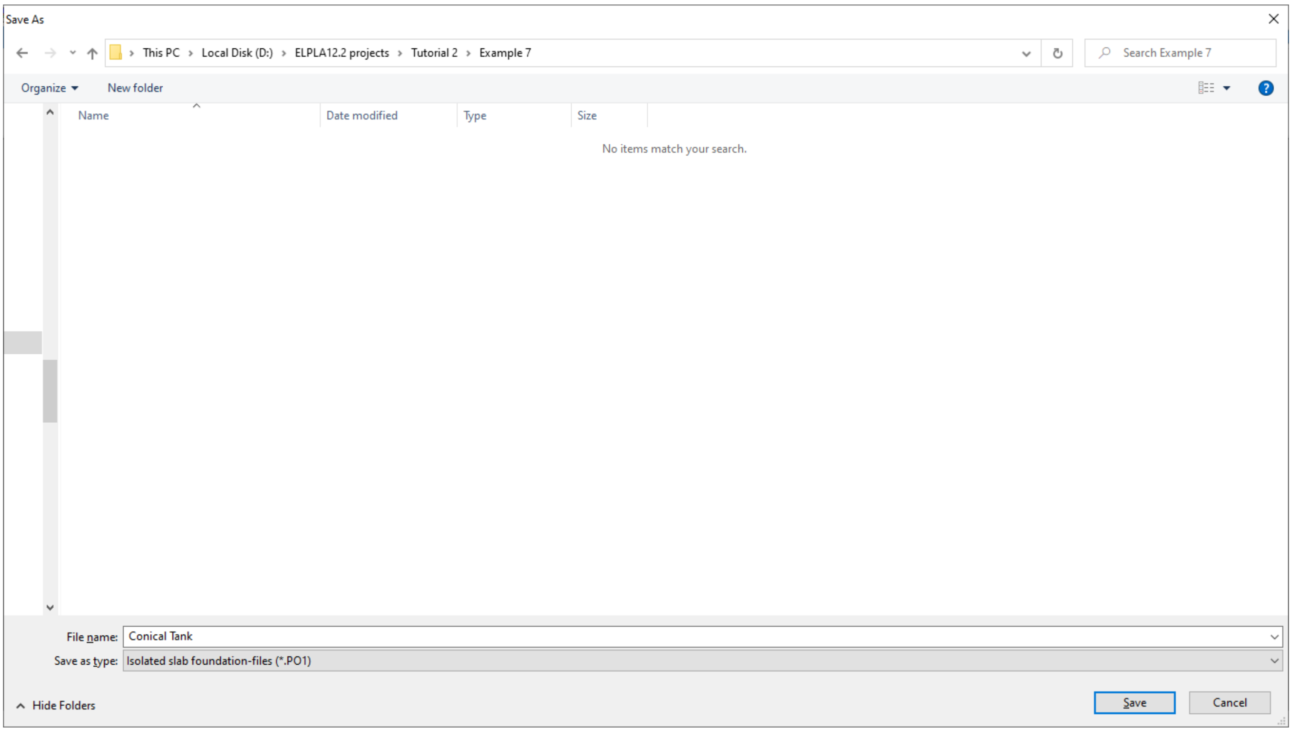


Figure 7.6 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Conical Tank] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 7.7 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a tank with conical base resting on *Winkler's* medium"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

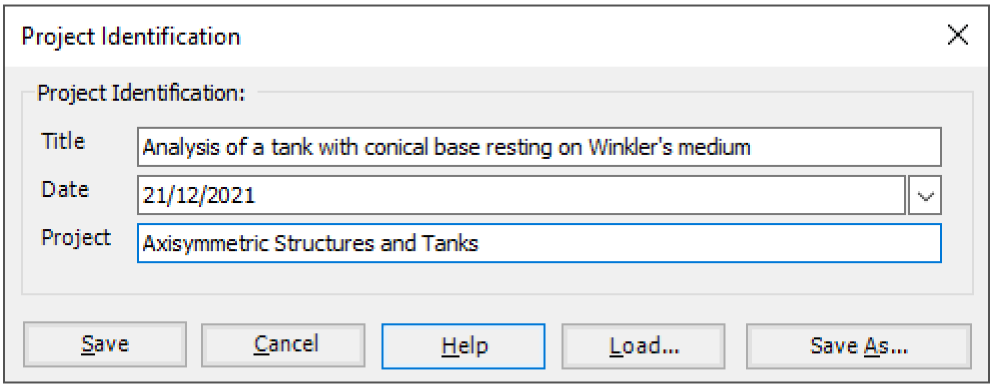


Figure 7.7 "Project Identification" dialog box

4.3 FE-Net data

For the given problem, a cylindrical water container with a conical base of a radius of a = 7.5 [m] and a height of H = 6 [m] is considered, the height of the tank is divided into two main segments, the main segment of the wall is divided into 20 elements (20×0.3 [m]), while the conical part is divided into 17 elements.

To define the FE-Net for this tank, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 7.8. This wizard will guide you through the steps required to generate a FE-Net, the first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

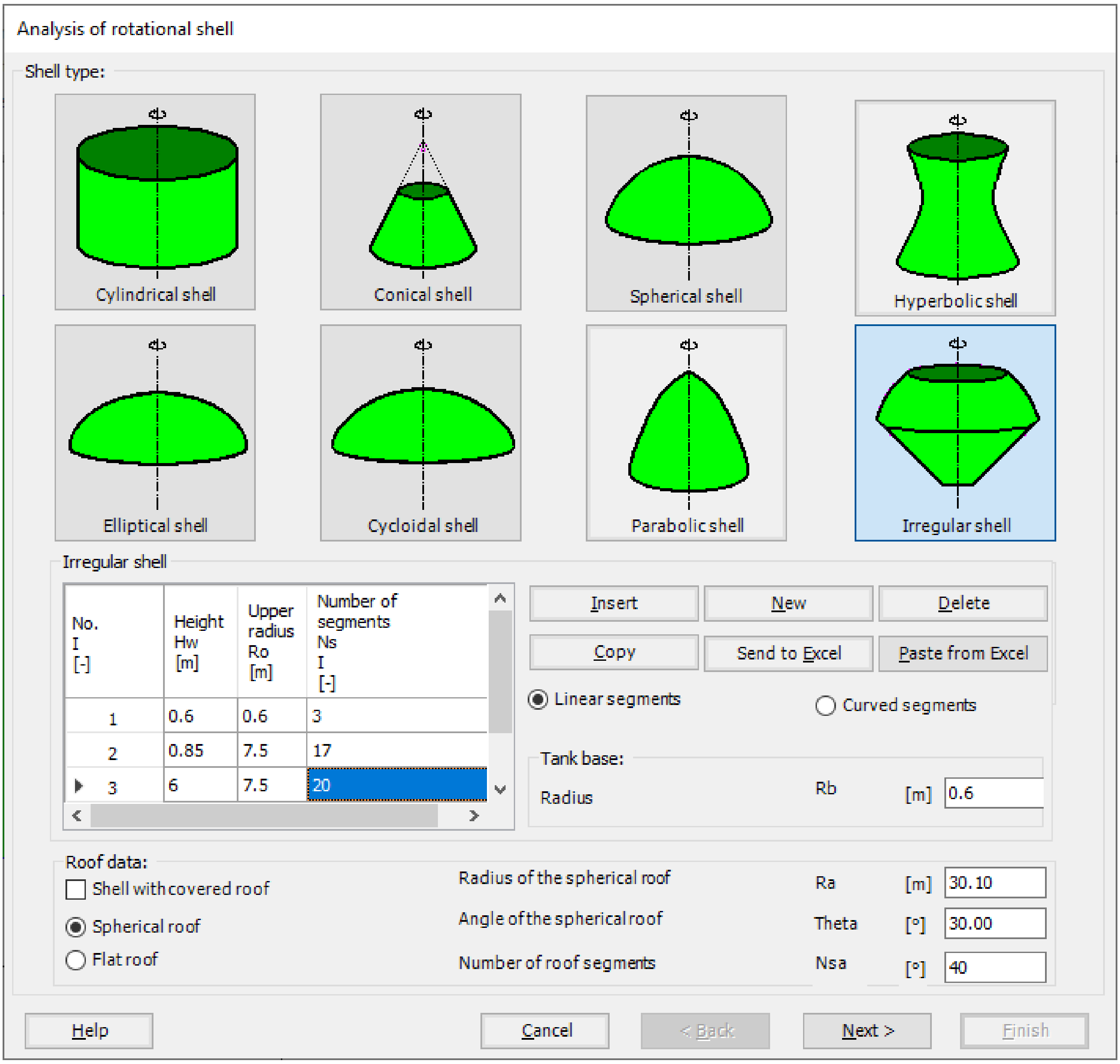


Figure 7.8 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Irregular shell" button
- Type 0.6 in the "Radius of the tank base" edit box,

To identify the foundation segment

- Type 0.6 in the "Height" edit box
- Type 0.6 in the "Upper radius" edit box
- Type 3 in the "Number of segments" edit box

To identify the segment of the conical part

- Type 0.85 in the "Height" edit box
- Type 7.5 in the "Upper radius" edit box
- Type 17 in the "Number of segments" edit box

To identify the segment of the tank height

- Type 6 in the "Height" edit box
- Type 7.5 in the "Upper radius" edit box
- Type 20 in the "Number of segments" edit box

- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Irregular shell" Form containing the data of the segments appears in Figure 7.9, The user can edit the data of each segment individually or all of them by using "In Table" button.

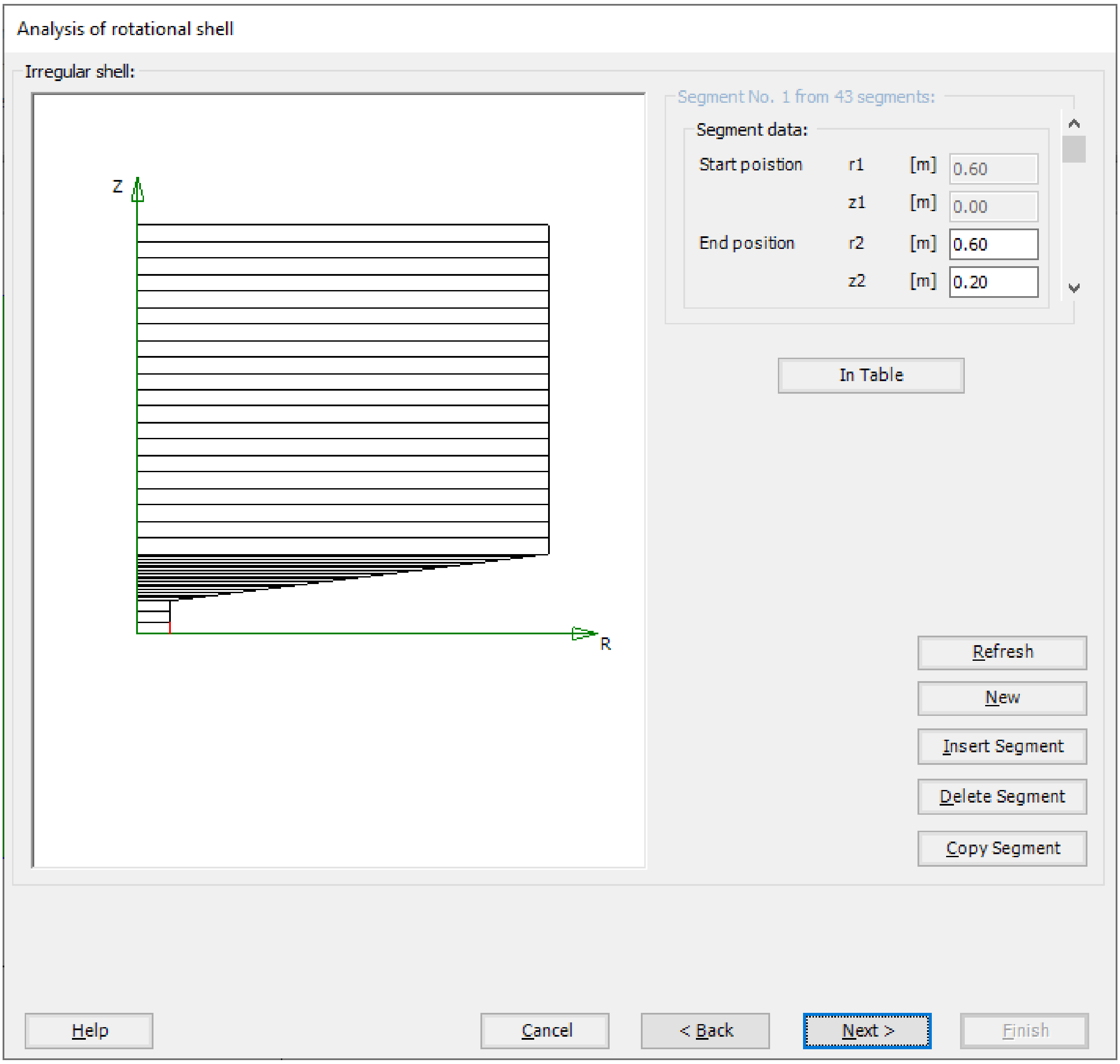


Figure 7.9 " Irregular shell" Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Net of Base" Form appears Figure 7.10.

To edit the grid spacing in *x*-direction of the tank foundation, do the following steps in "Grid in *x*-direction" frame:

- Choose "Constant grid interval" check box
- Type 10 in the "No. of grid intervals" edit box

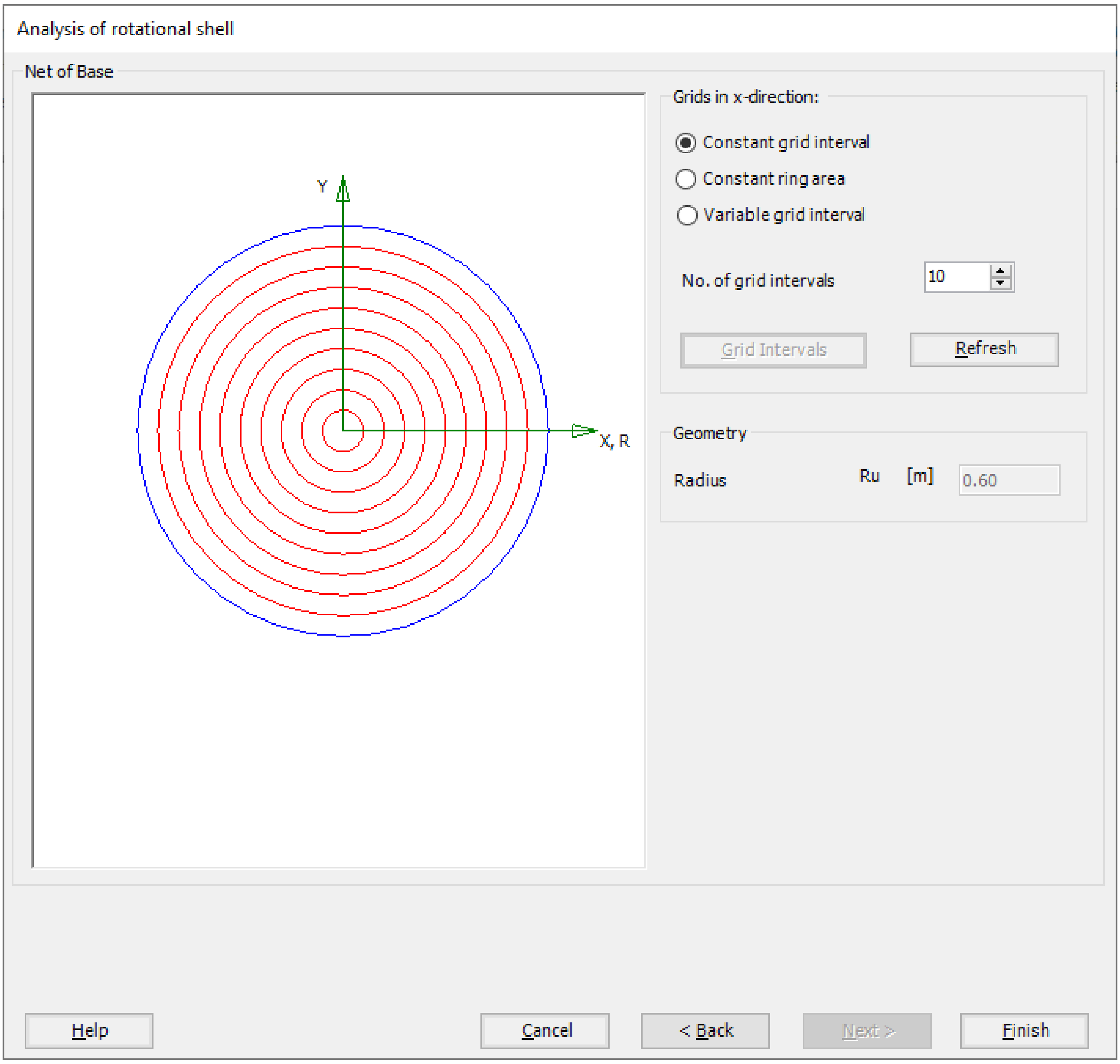


Figure 7.10 "Net of Base" Form

Click "Finish" button, the FE-Net of the wall and a sector from the base appears in Figure 7.11

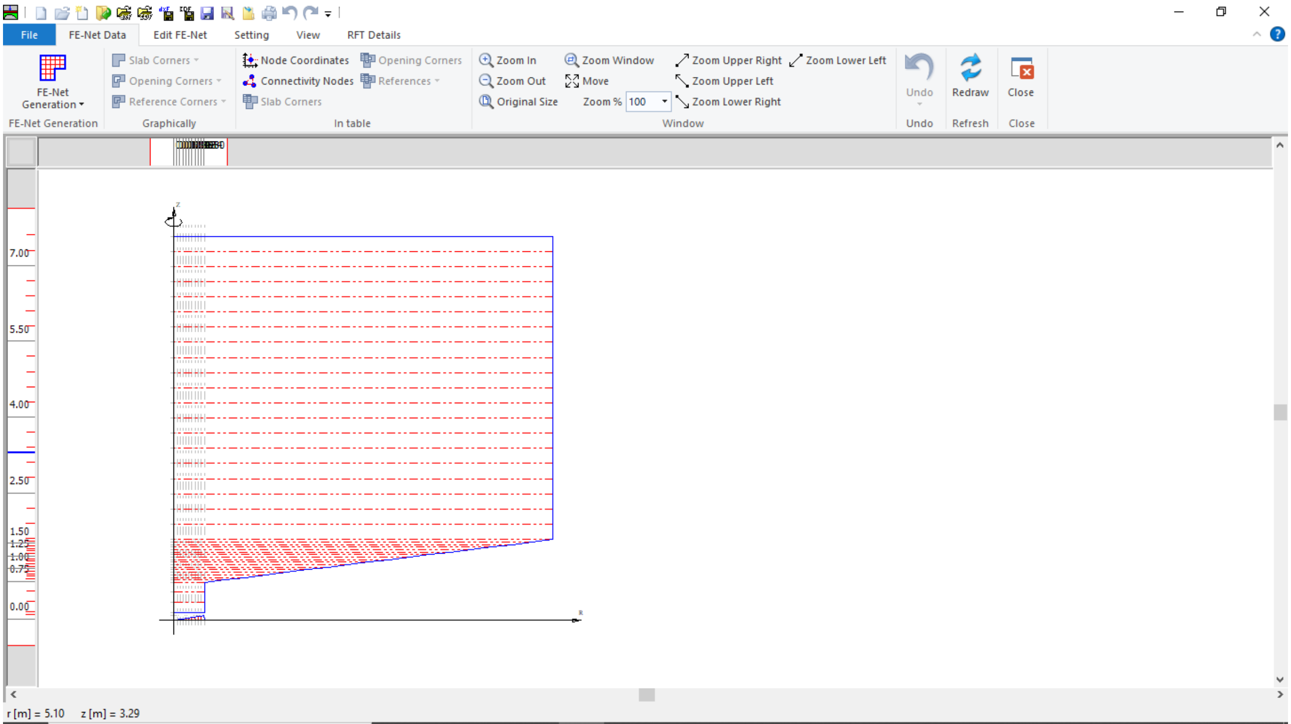


Figure 7.11 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 7.11 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 7.11 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Shell properties

To define the tank properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 7.12 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, unit weight of the tank, and the filled material properties.

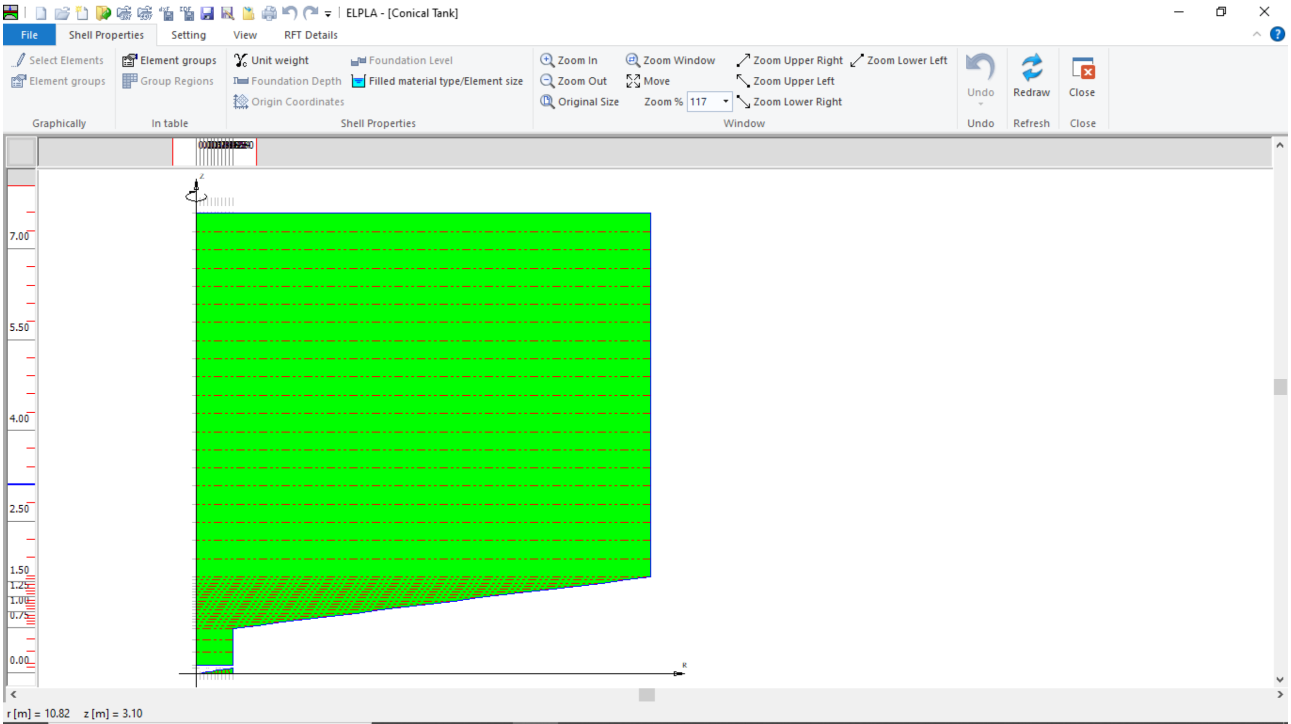


Figure 7.12 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 7.13 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness. Then click "OK" button.

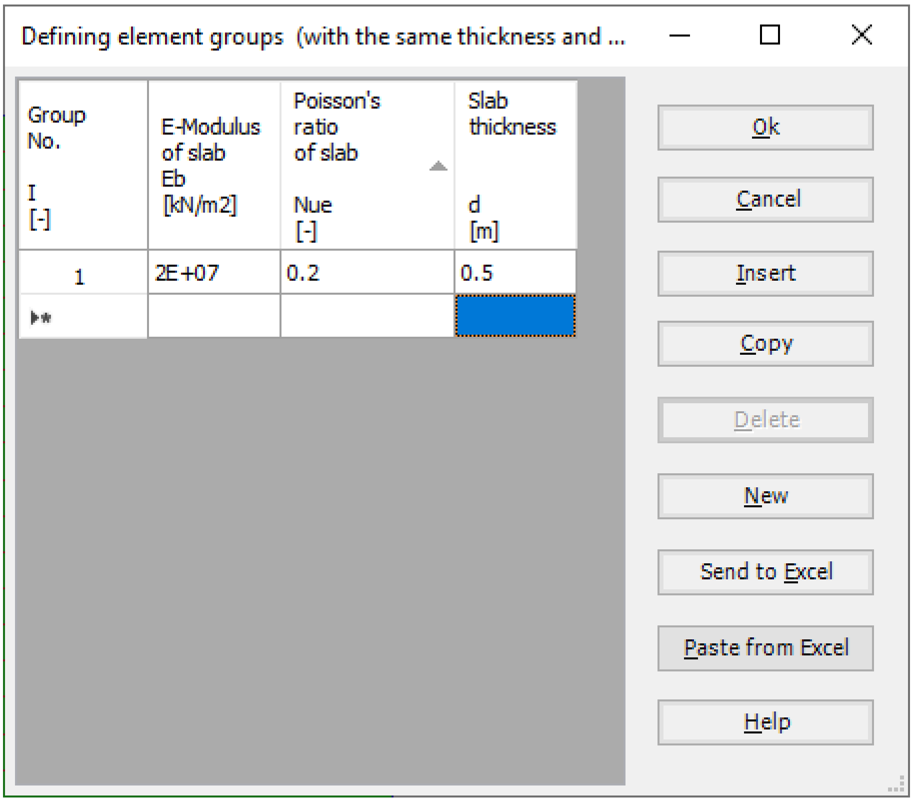


Figure 7.13 "Defining element groups" list box

To enter the unit weight of the tank, choose "Unit weight" command from "Shell Properties" menu in Figure 7.12. The following dialog box in Figure 7.14 with a default unit weight of 25 [kN/m^3] appears, click "OK" button.

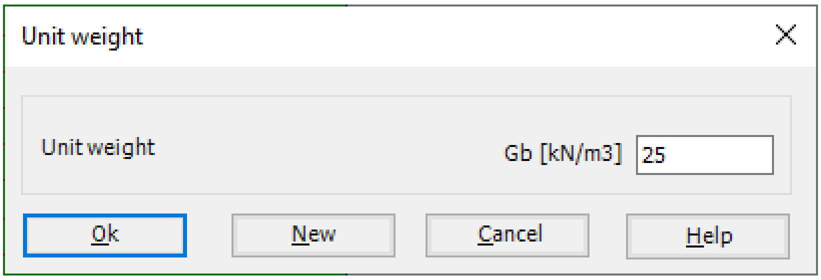


Figure 7.14 "Unit weight" dialog box

To define the filled material properties of the tank, choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 7.12. The following form in Figure 7.15 appears.

To define the filled material properties of the Tank:

- Select "Liquid container" option
- Type 6.85 in the "Height of the liquid" edit box
- Type 9.81 in the "Unit weight of the liquid" edit box

To define the element size of the tank:

- Check the "Constant element sizes in z-direction" check box
- Type 0.2 in the "Element size in each shell segment" edit box
- Click "OK" button

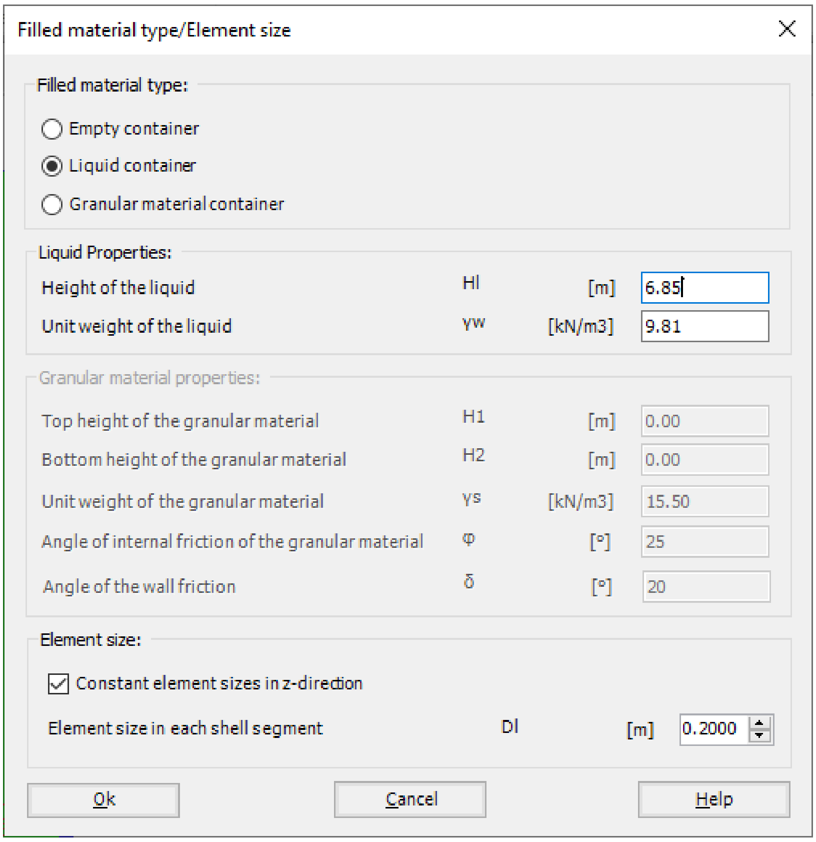


Figure 7.15 "Liquid properties/Element size" Form

After defining the shell properties, do the following two steps:

- Choose "Save" command from "File" menu in Figure 7.12 to save the shell properties
- Choose "Close" command from "File" menu in Figure 7.12 to close the "Shell properties" window and return to *ELPLA* main window

4.5 Soil Properties

To define the soil properties, choose "Soil Properties" command from “Data” Tab. The following "Soil Properties" form in Figure 7.16 appears, enter Modulus of subgrade reaction of the soil and the ground water under the surface. Other data in the example are not required.

Soil data:

Modulus of subgrade reaction	k_s	= 100 000	[kN/ m^3]
Ground water depth under the surface	Gw	= 1	[m]

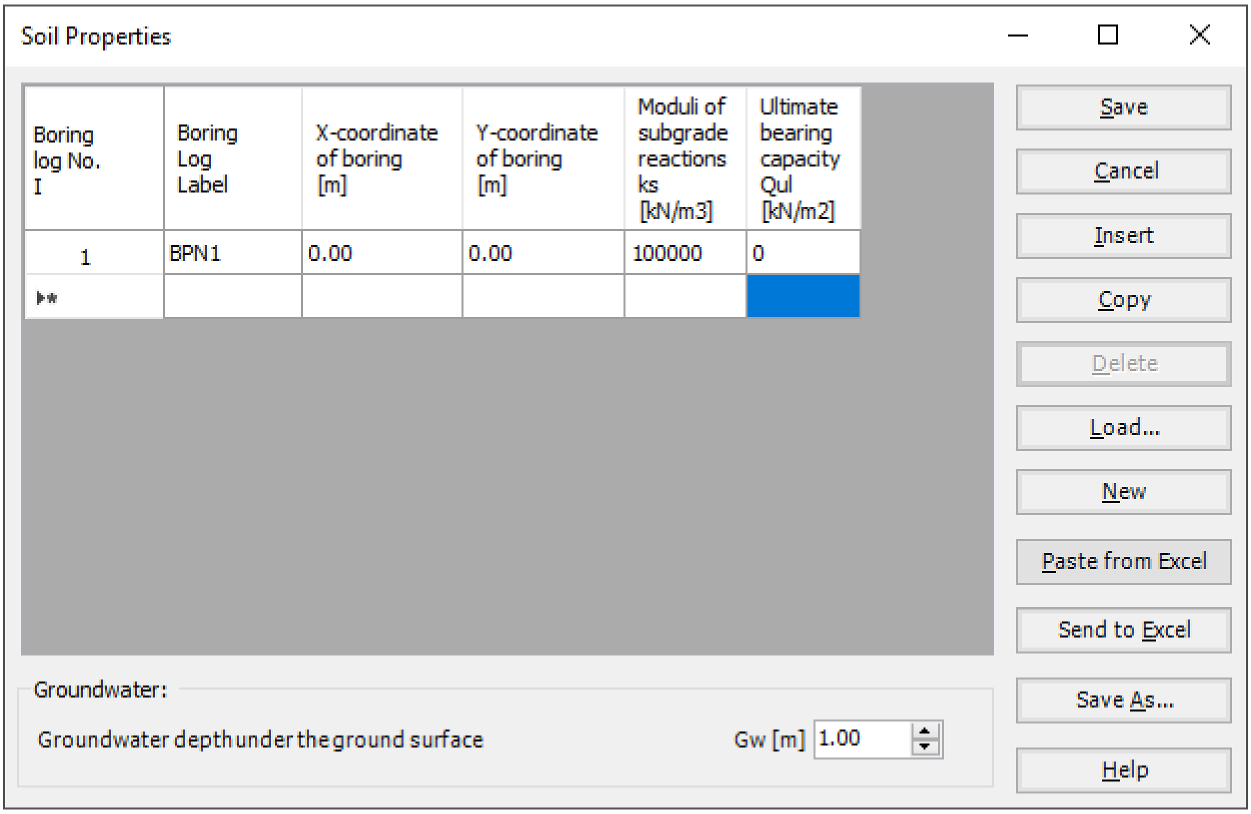


Figure 7.16 "Soil Properties" form

4.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 7.17 appears.

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 7.17. In this example, there is not applied load, as the vertical load has been already defined by the unit weight of the tank, while the hydrostatic pressure on the tank wall is defined by the unit weight of water.

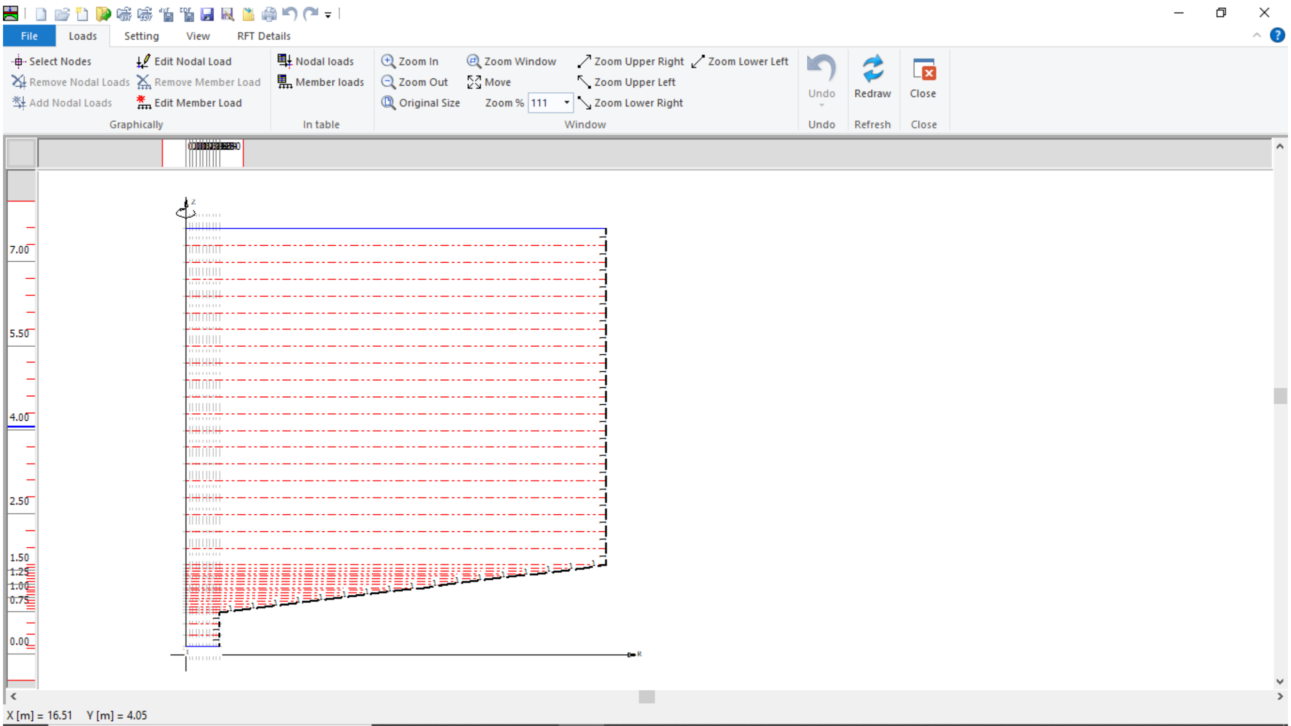


Figure 7.17 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 7.17 to save the load data
- Choose "Close" command from "File" menu in Figure 7.17 to close the "Loads" window and return to *ELPLA* main window

4.7 Spring Supports

To define the Spring supports, choose "Spring Supports" command from “Data” Tab. The following window in Figure 7.18 appears.

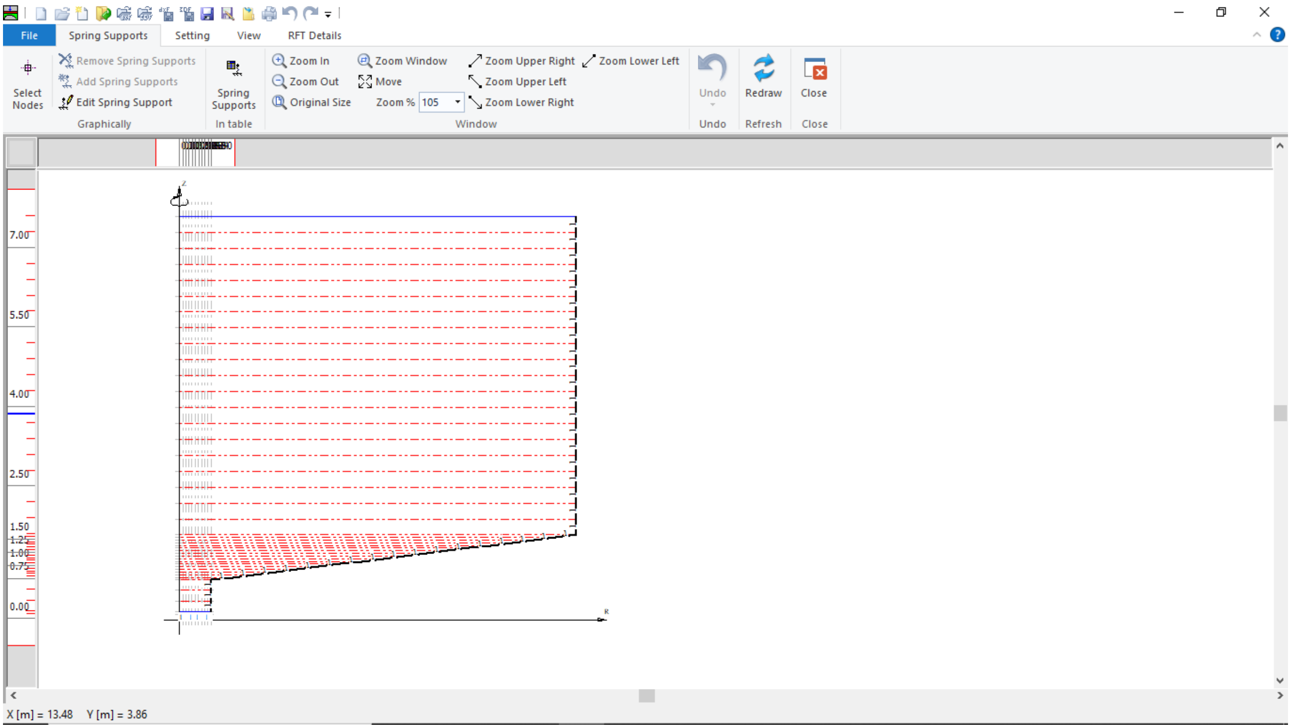


Figure 7.18 "Spring Supports" Window

To define the spring supports on the net:

- Choose "Select Nodes" command from "Graphically" menu in Figure 7.18. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the node that has the spring support as shown in Figure 7.19
- After selecting the nodes, choose "Add Spring Supports" command from "Graphically" menu (Figure 7.18). The "Edit Spring Support" dialog box in Figure 7.20 appears.

Example 7

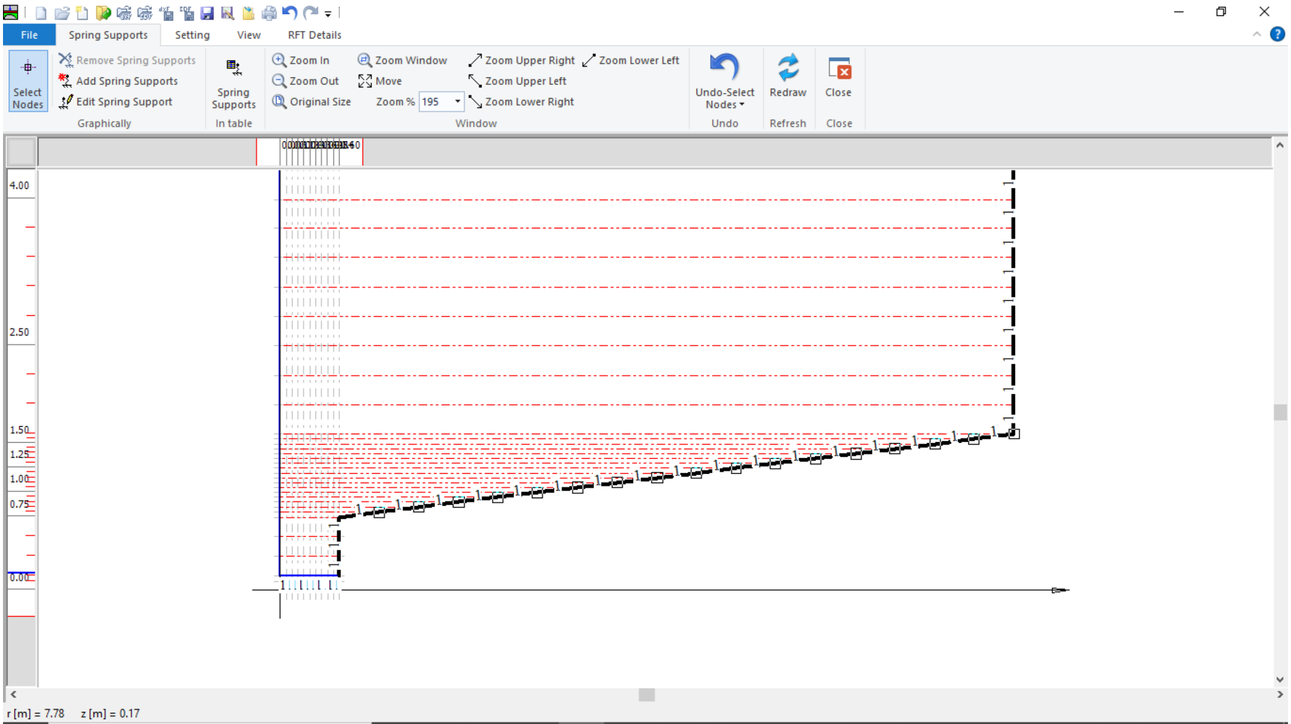


Figure 7.19 Selection of the nodes that have the spring support

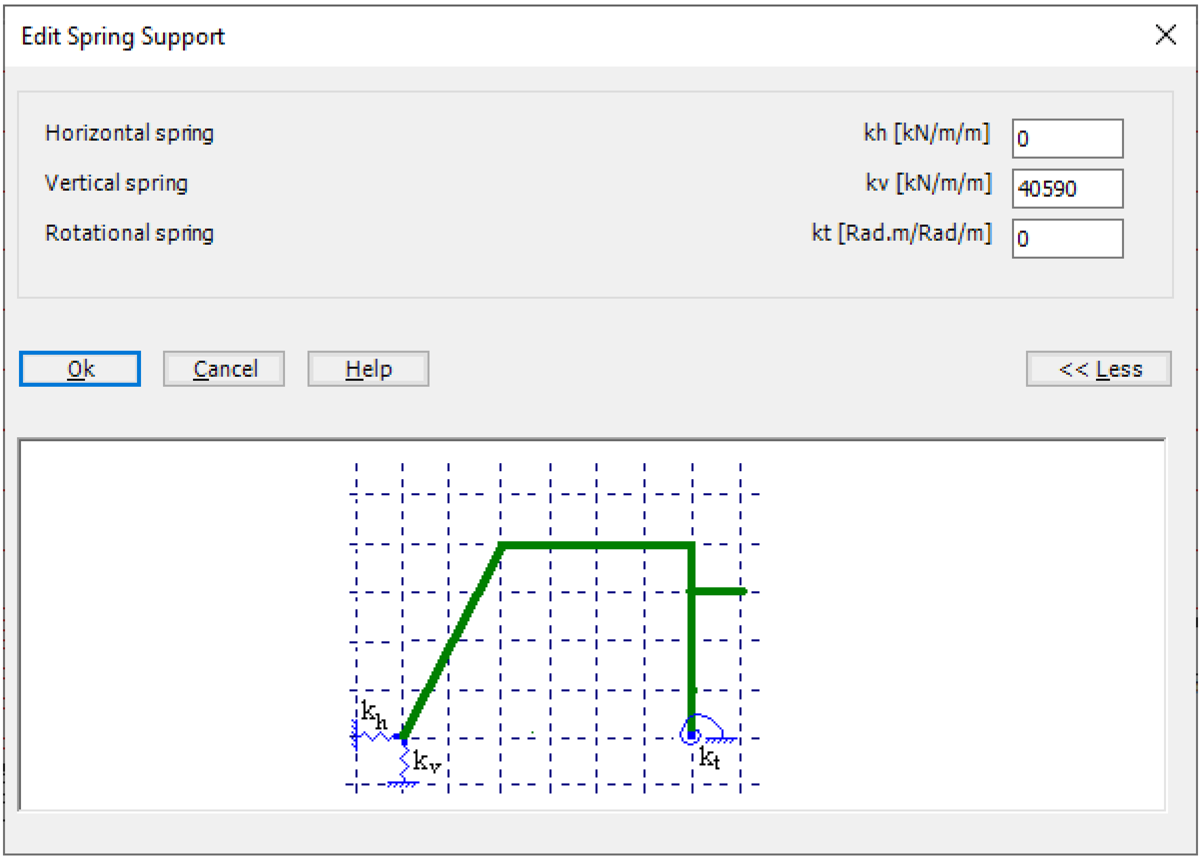


Figure 7.20 "Edit Spring Support" dialog box

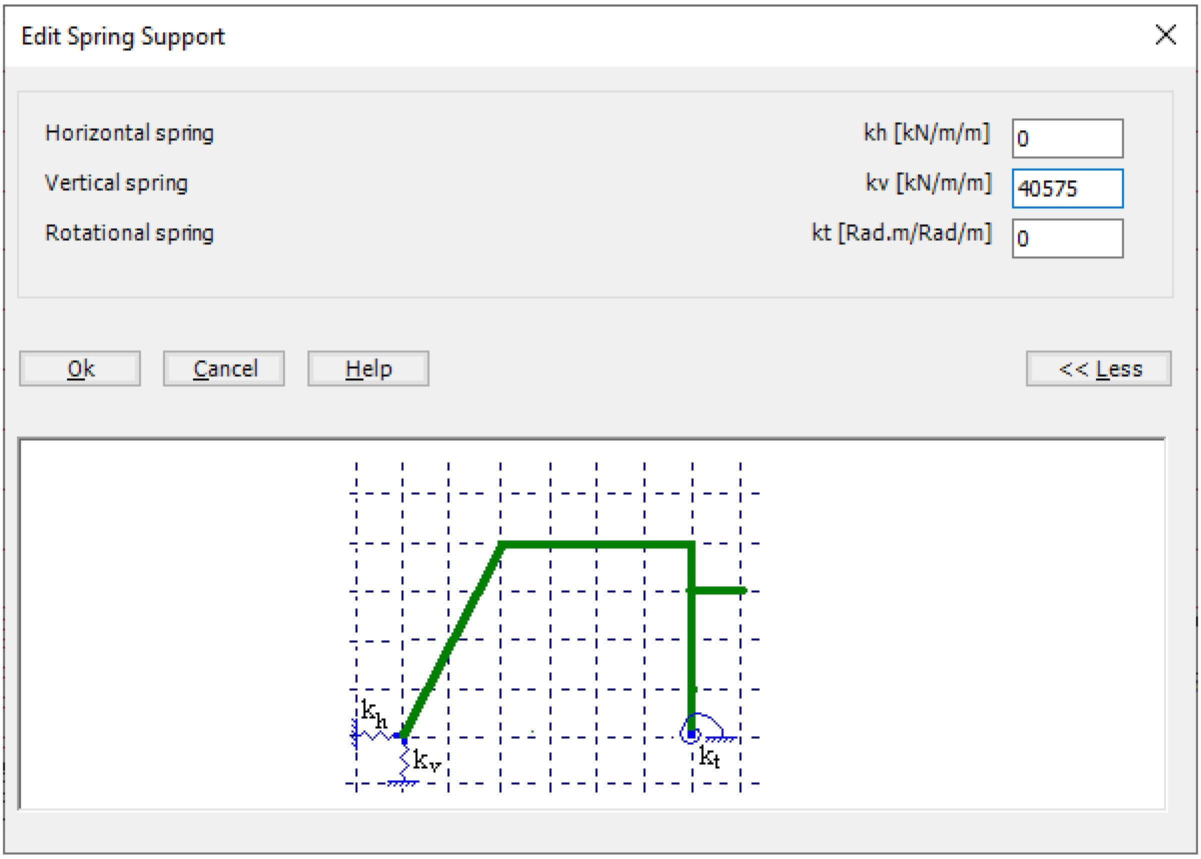


Figure 7.21 "Edit Spring Support" dialog box

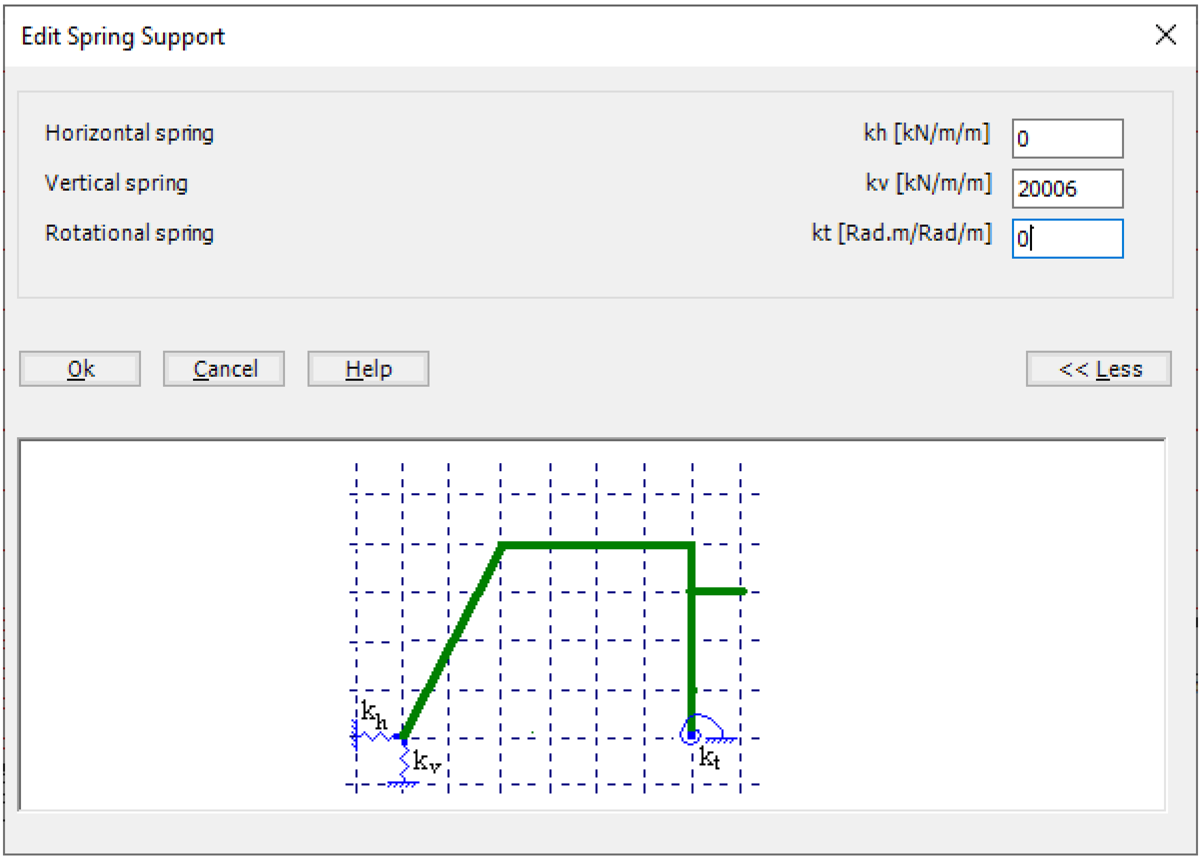


Figure 7.22 "Edit Spring Support" dialog box

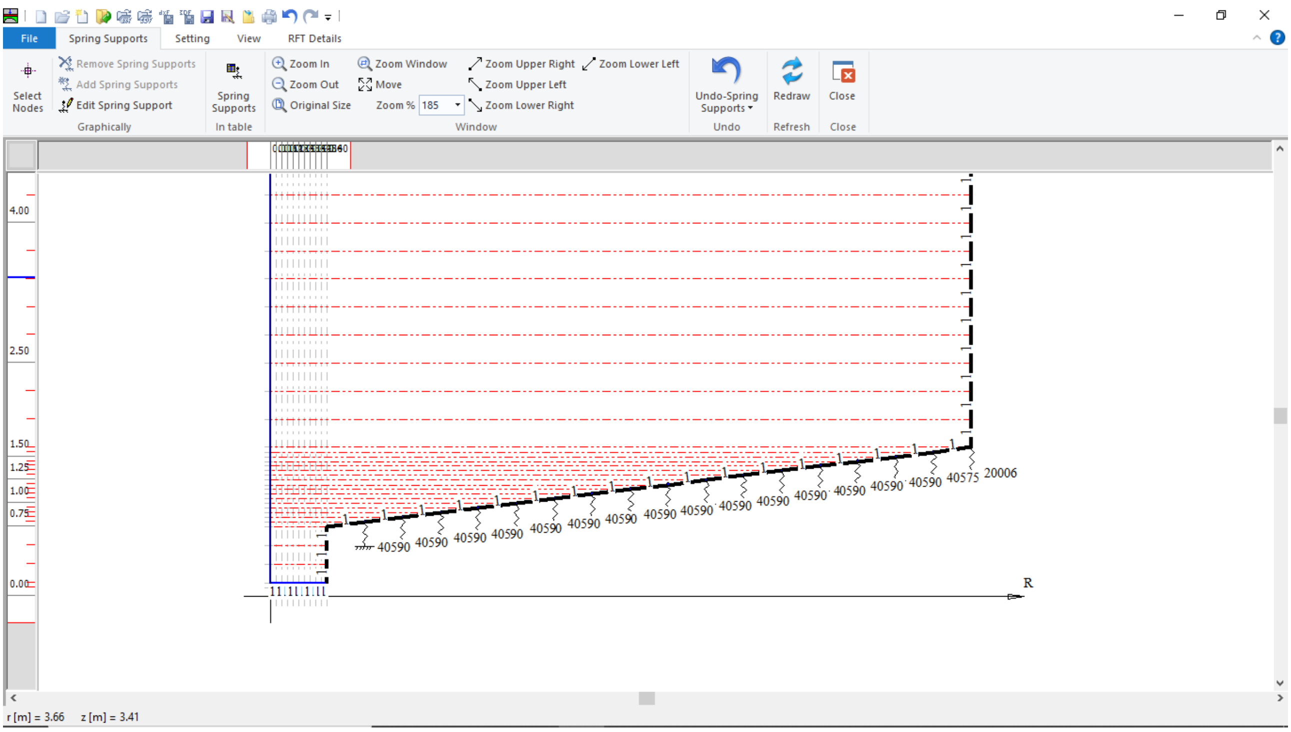


Figure 7.23 "Spring Supports" window after defining the springs

After defining the spring supports, do the following two steps

- Choose "Save " command from "File" menu in Figure 7.23 to save the data of the springs
- Choose "Close" command from "File" menu in Figure 7.23 to close the "Spring Supports" window and return to the main window

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

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 7.24.

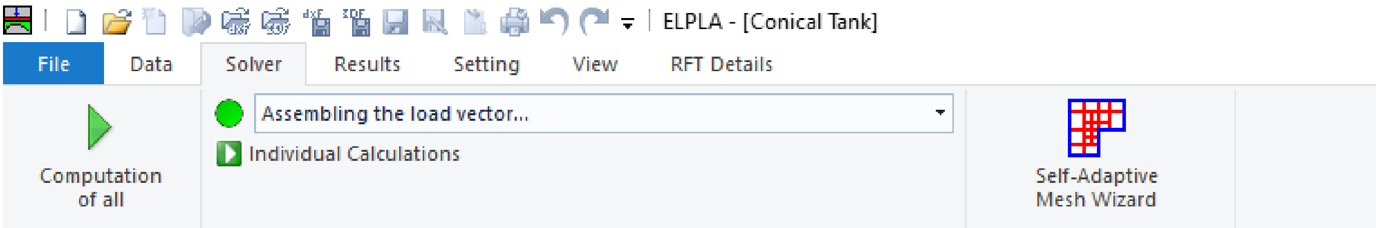


Figure 7.24 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

- Assembling the load vector
- Determining the modulus of subgrade reaction
- Assembling the slab stiffness matrix
- Solving the system of linear equations (band matrix)
- Determining deformation, internal forces, contact pressures

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

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab Window.

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

Analysis progress

Analysis progress menu in Figure 7.25 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

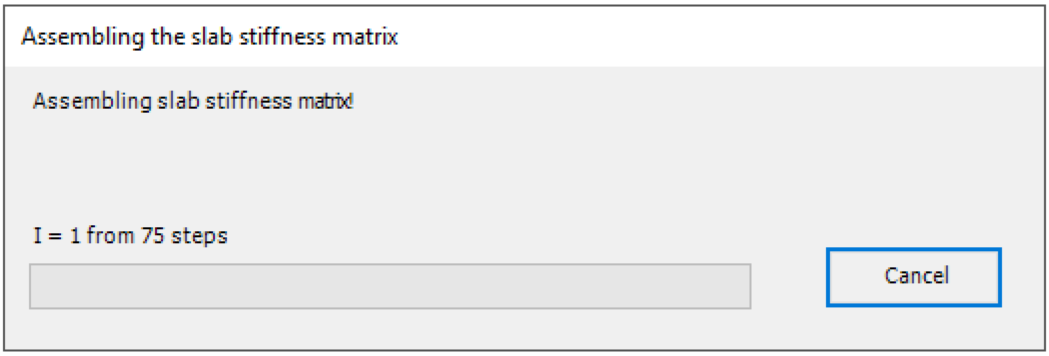


Figure 7.25 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 7.26. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

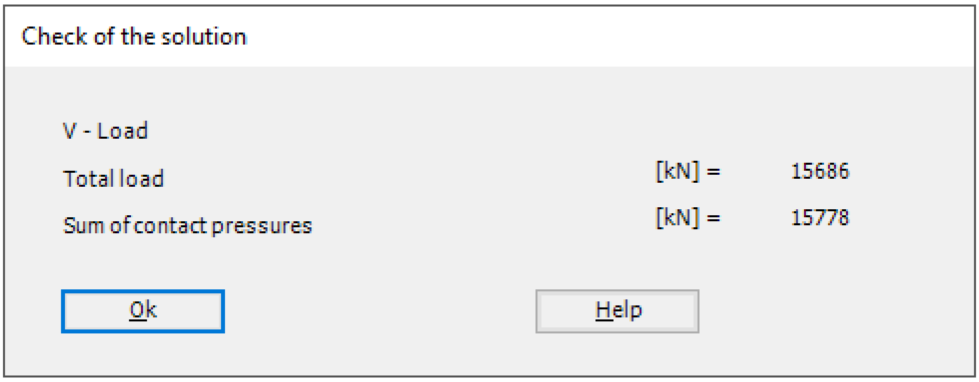


Figure 7.26 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab.

To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 7.27).

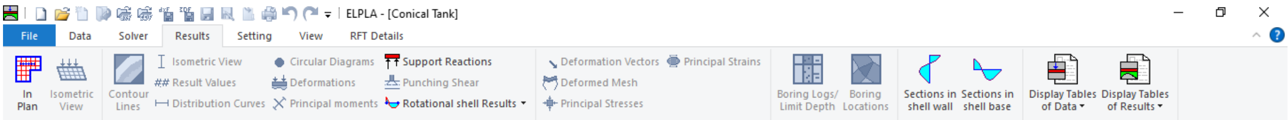


Figure 7.27 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Sections in shell wall
- Sections in shell base

To view the radial forces in the shell wall:

- Choose "Sections in shell wall" command from "Section" menu. The following option box in Figure 7.28 appears
- In the "Sections in shell wall" option box, select "Radial forces Nr" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 7.29.

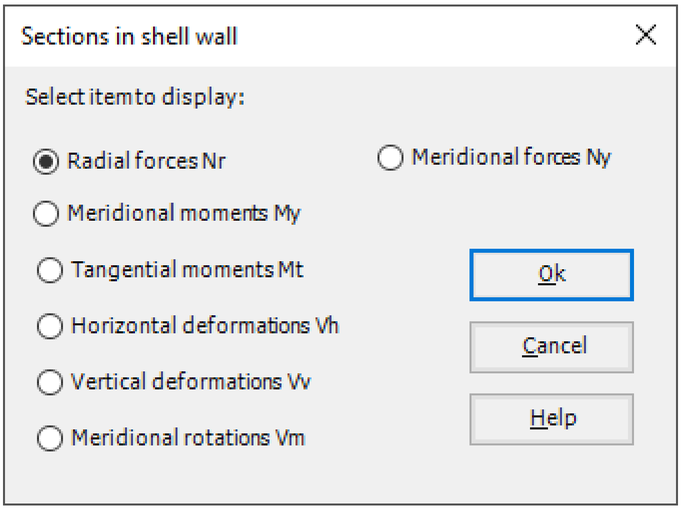


Figure 7.28 "Sections in shell wall" option box

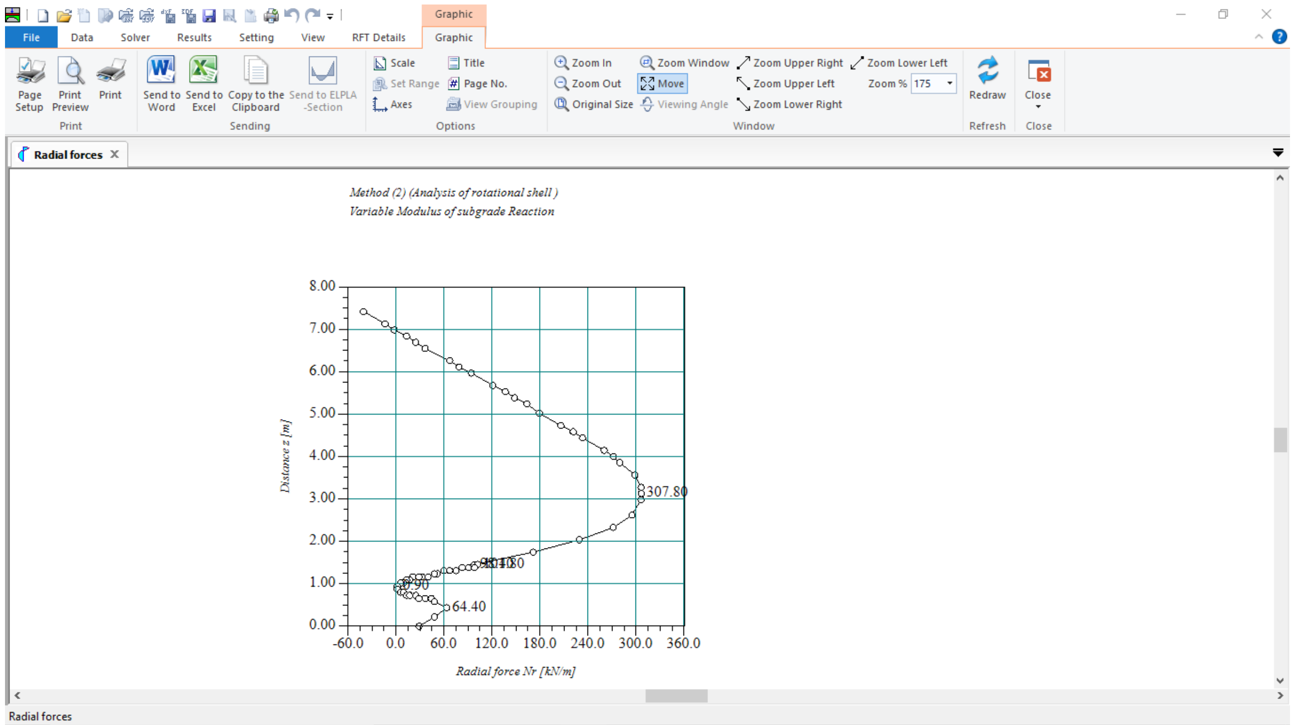


Figure 7.29 Radial forces in shell wall

To view element groups of the tank

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 7.30 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

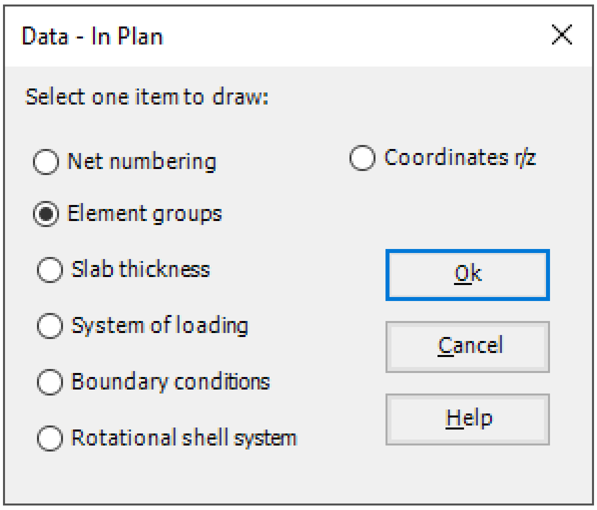


Figure 7.30 "Data – In Plan" option box

To view the meridional moments on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 7.31 appears
- In this check group box, check "Meridional moments *My*" check box
- The user can choose any other data to be viewed
- Click "OK" button

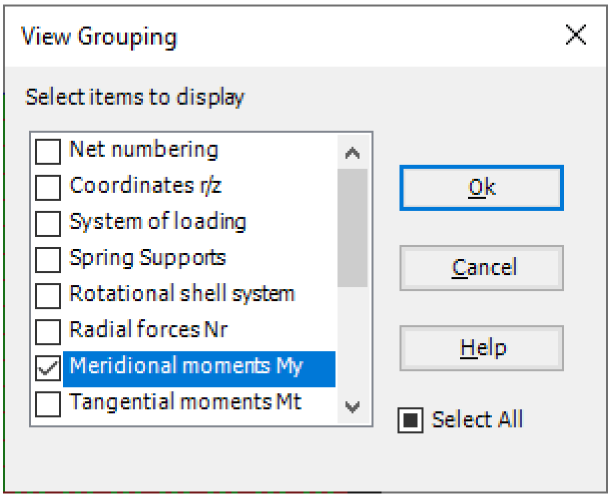


Figure 7.31 "View Grouping" check group box

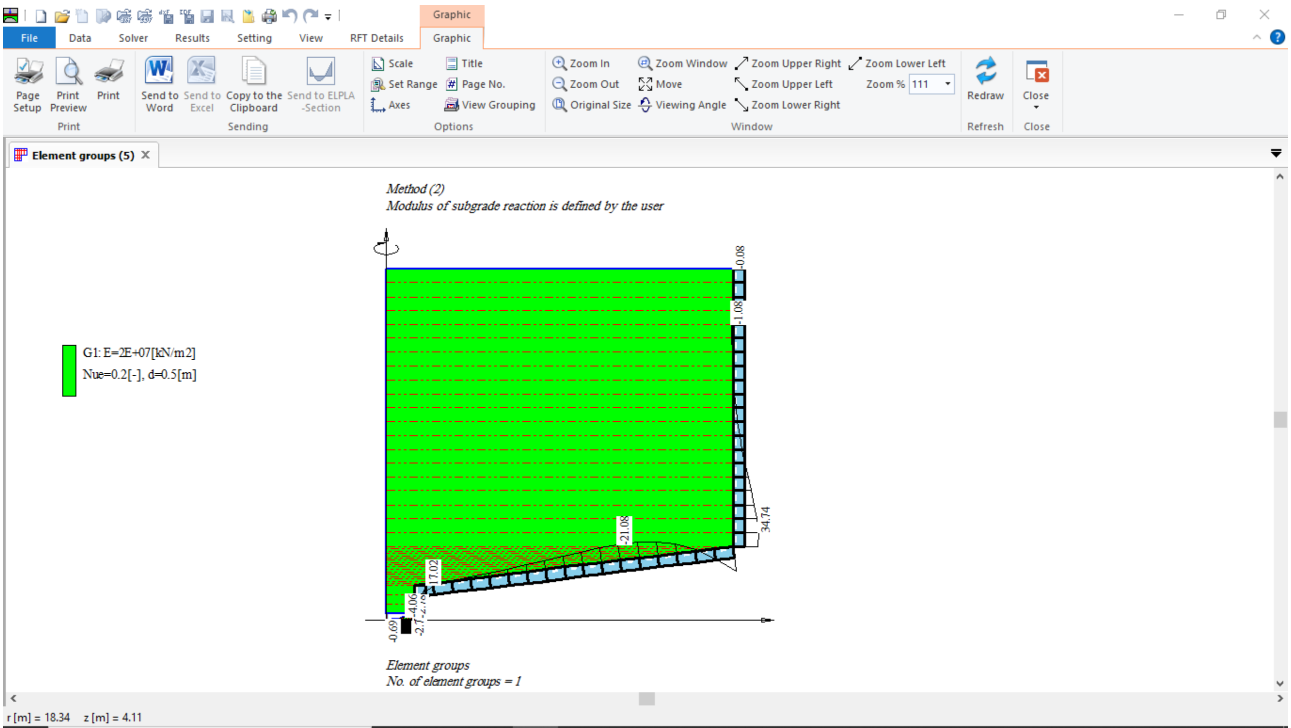


Figure 7.32 Element groups of the tank

Example 8

Analysis of a tank resting on half space soil medium

Contents Page

1 Description of the problem

An example of a circular cylindrical tank resting on an isotropic elastic half space soil medium is selected to illustrate some features of *ELPLA* for analyzing shell elements.

2 Tank geometry and properties

A circular cylindrical tank of an inner diameter of d = 18 [m] and a height of H = 7.5 [m] is considered as shown in Figure 8.1. The thickness of the tank wall and base is t = 0.36 [m]. The tank is filled with water. Figure 8.1 shows the storage tank with dimensions, while the tank material and unit weight of the water are listed in Table 8.1. The data of soil medium under the base of the tank are shown in Table 8.2.

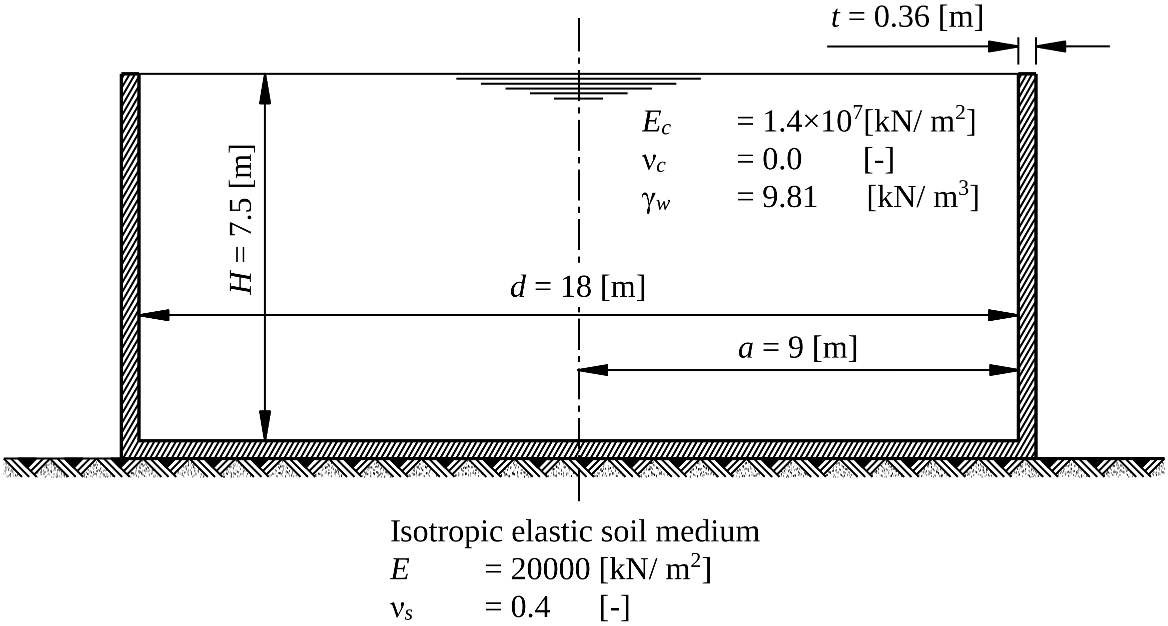


Figure 8.1 Circular cylindrical tank resting on an isotropic elastic soil medium

Table 8.1 Tank material and water unit weight

Modulus of Elasticity of the tank material	E_c	= 1.4×10^7	[kN/ m^2]
Poisson's ratio of the tank material	ν_c	= 0.2	[-]
Unit weight of the water	γ_w	= 9.81	[kN/ m^3]
Unit weight of the tank material	γ_b	= 25	[kN/ m^3]

Table 8.2 Soil data

Modulus of Elasticity of the soil medium	E	= 20000	[kN/m^2]
Poisson's ratio of the soil medium	ν_s	= 0.4	[-]

3 Numerical Analysis

In order to analyze a water storage tank resting on an isotropic elastic half space soil medium using *ELPLA*. The height of the tank is divided into 30 equal segments, each of 0.25 [m], as shown in Figure 8.2, while the base of the tank is divided into 45 equal segments, each of 0.2 [m].

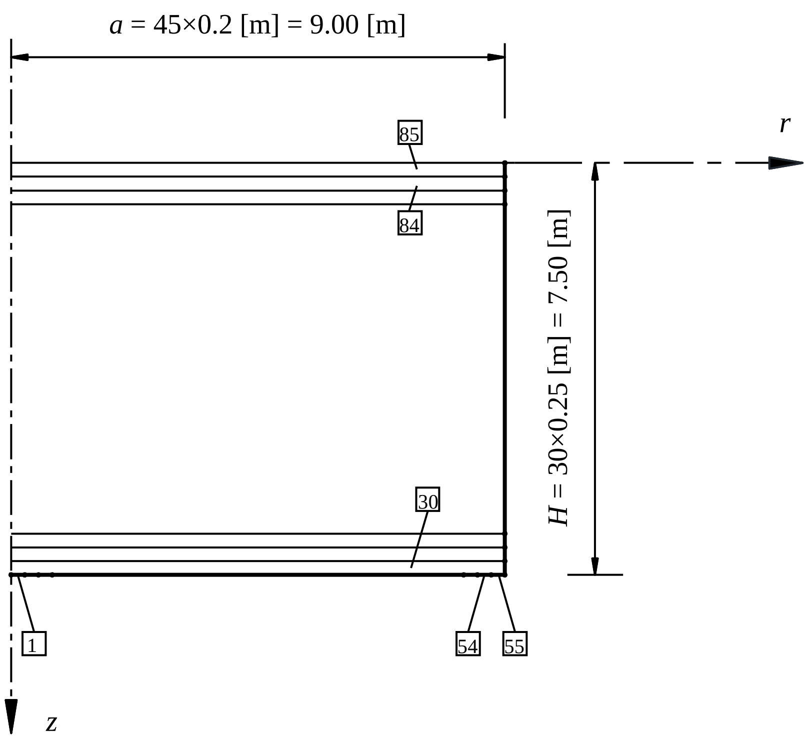


Figure 8.2 Finite element mesh of the tank

4 Creating the project

In this section, the user will learn how to create a project for analyzing a circular cylindrical tank resting on an isotropic elastic soil medium. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 8.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of forms. The first form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 8.3).

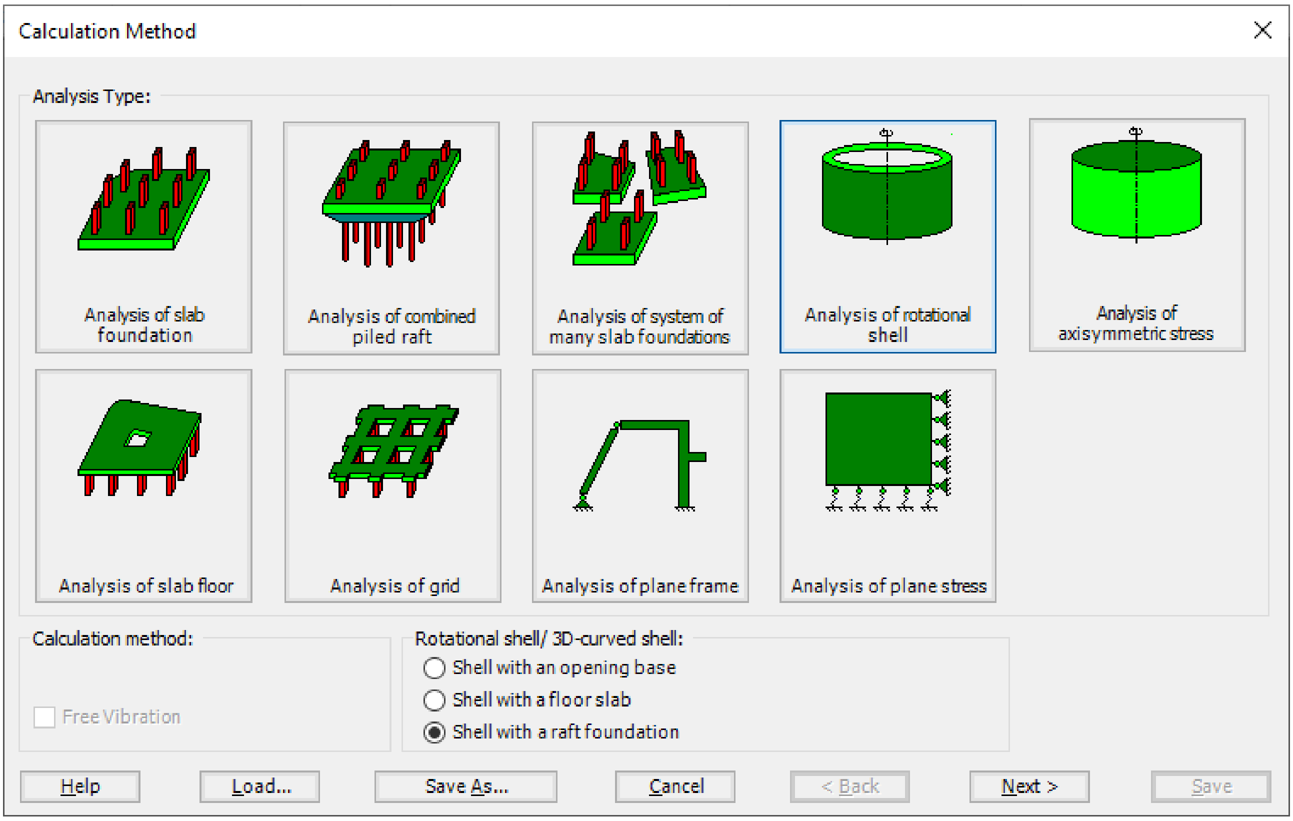


Figure 8.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 8.3, define the analysis type of the problem. As the analysis type is a circular cylindrical tank resting on an isotropic elastic soil medium problem, select "Analysis of rotational Shell" button, and check "Shell with a raft foundation" option then click "Next" button to go to the next form. After clicking "Next" button, the "Calculation Method" Form appears, Figure 8.4.

To define the calculation method:

- Select the calculation method "6-Modulus of Compressibility (Iteration)"
- To determine the subsoil model, select "Half Space model"
- Click "Next" button to go to the next form

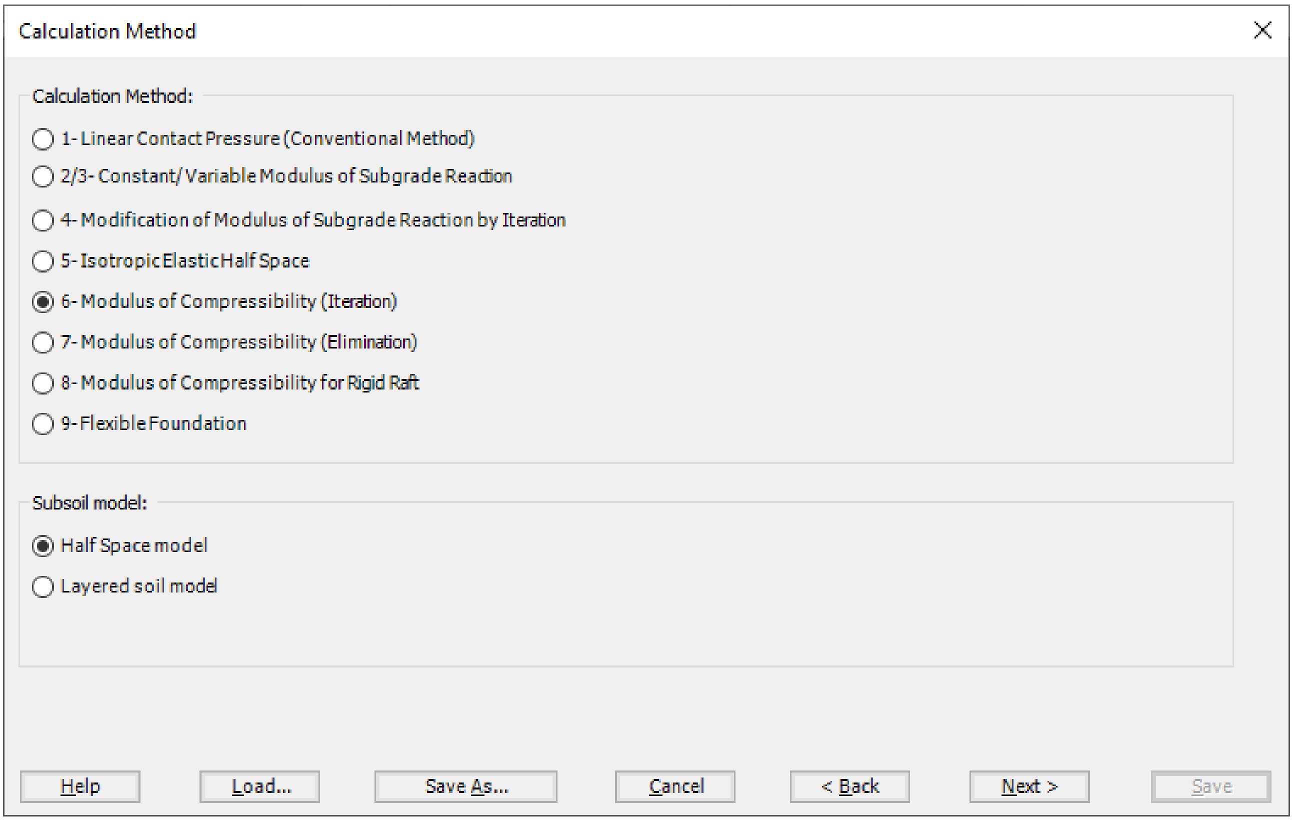


Figure 8.4 "Calculation Method" Form

The last Form in the wizard is the "Options" Form, Figure 8.5. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Since no option will be considered in the analysis, click the "Save" button.

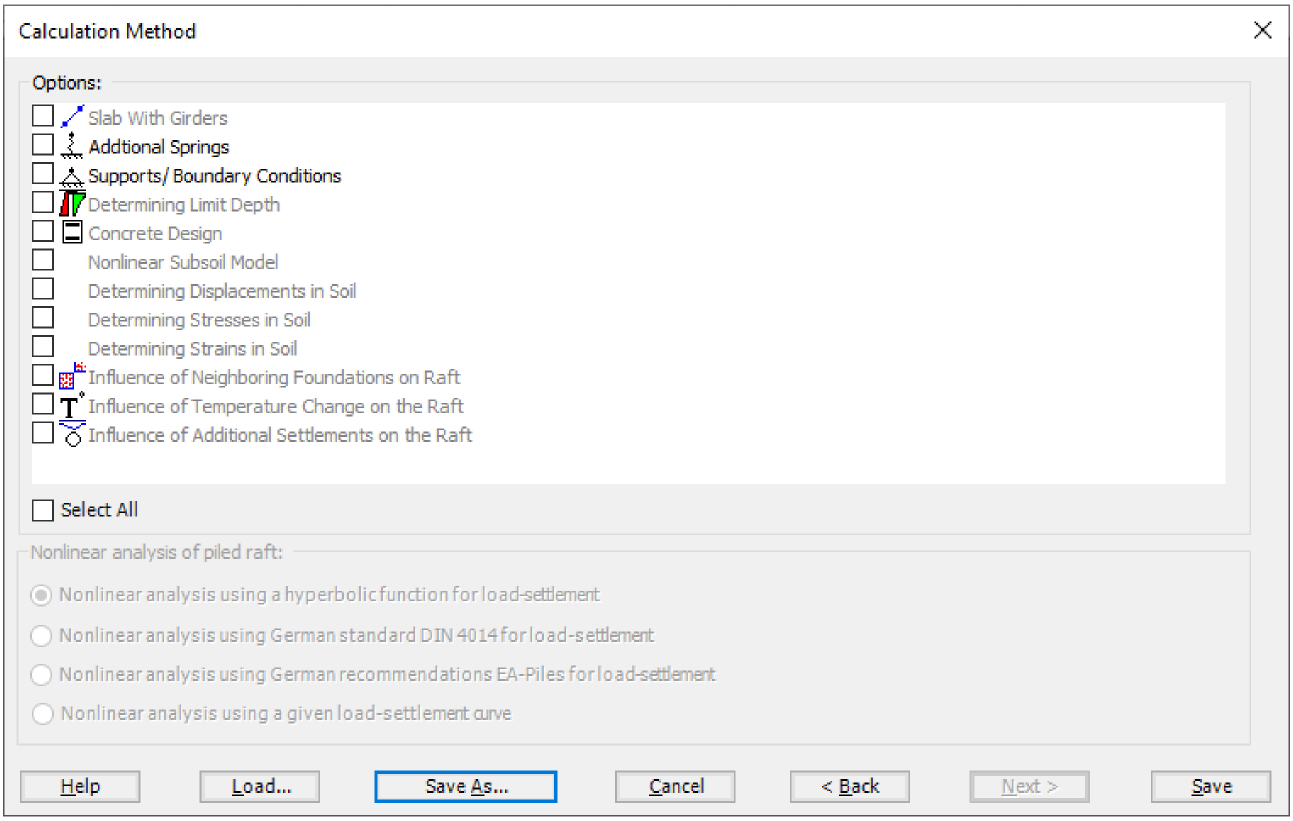


Figure 8.5 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 8.6. In this dialog box type a file name for the current project in "File name" edit box. For example, type "tank resting on half space soil medium". *ELPLA* will use automatically this file name in all reading and writing processes.

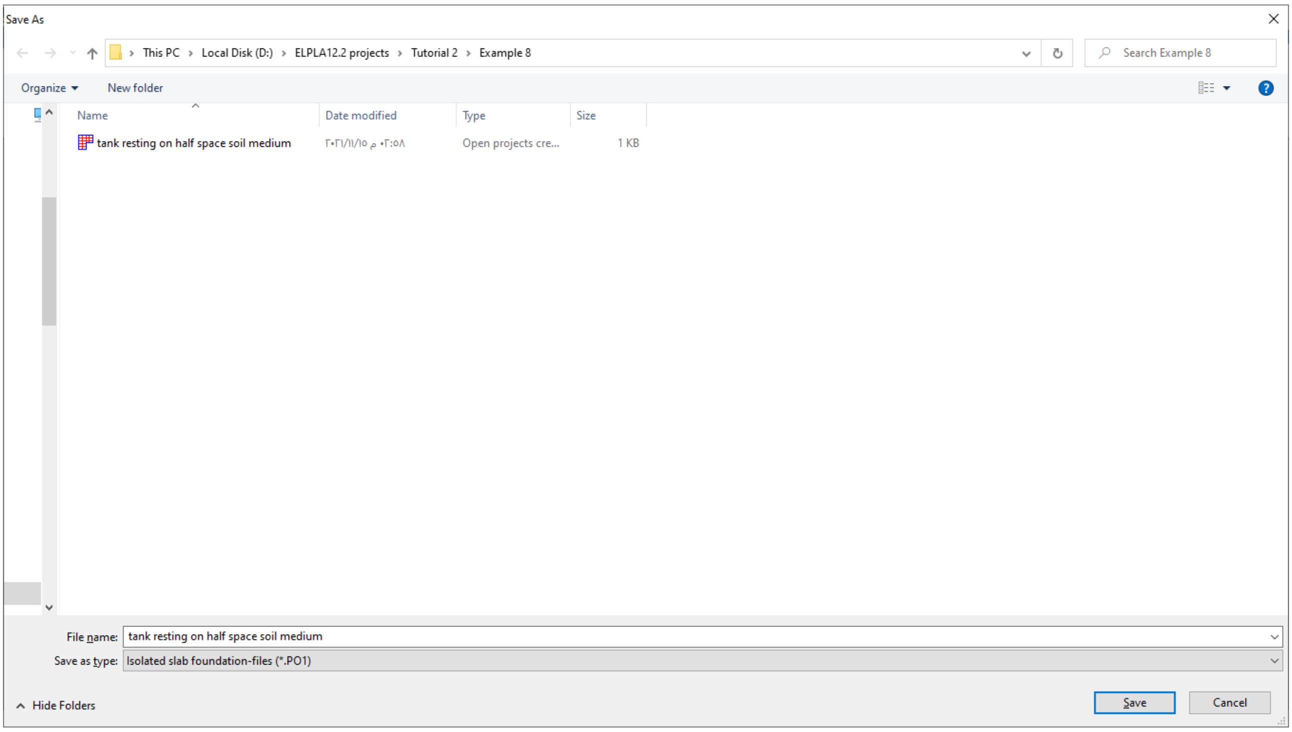


Figure 8.6 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [tank resting on half space soil medium] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 8.7 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a tank resting on a half space soil medium"
- Type the date of the project in the "Date" edit box
- Type "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

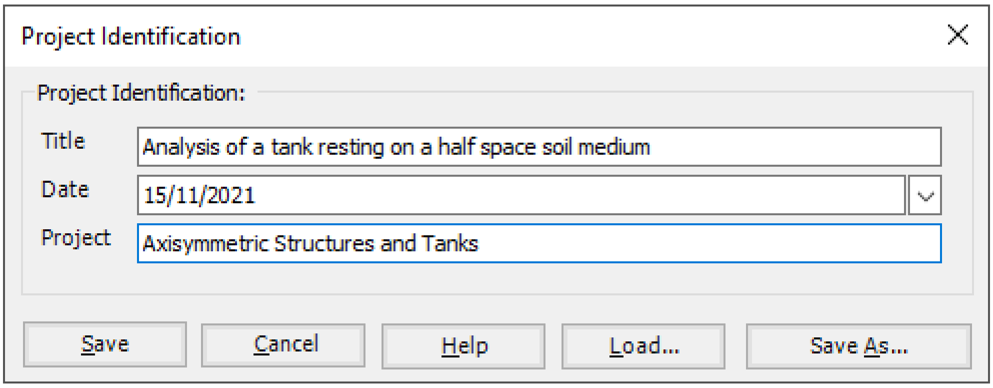


Figure 8.7 "Project Identification" dialog box

4.3 FE-Net data

For the given problem, the tank has an inner diameter of d = 18 [m] and a height of H = 7.5 [m], the height of the tank is divided into 30 equal segments, each of 0.25 [m], and the base of the tank is divided into 45 equal segments, each of 0.2 [m]. To define the FE-Net for this tank, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 8.8. This wizard will guide you through the steps required to generate a FE-Net.

The first form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

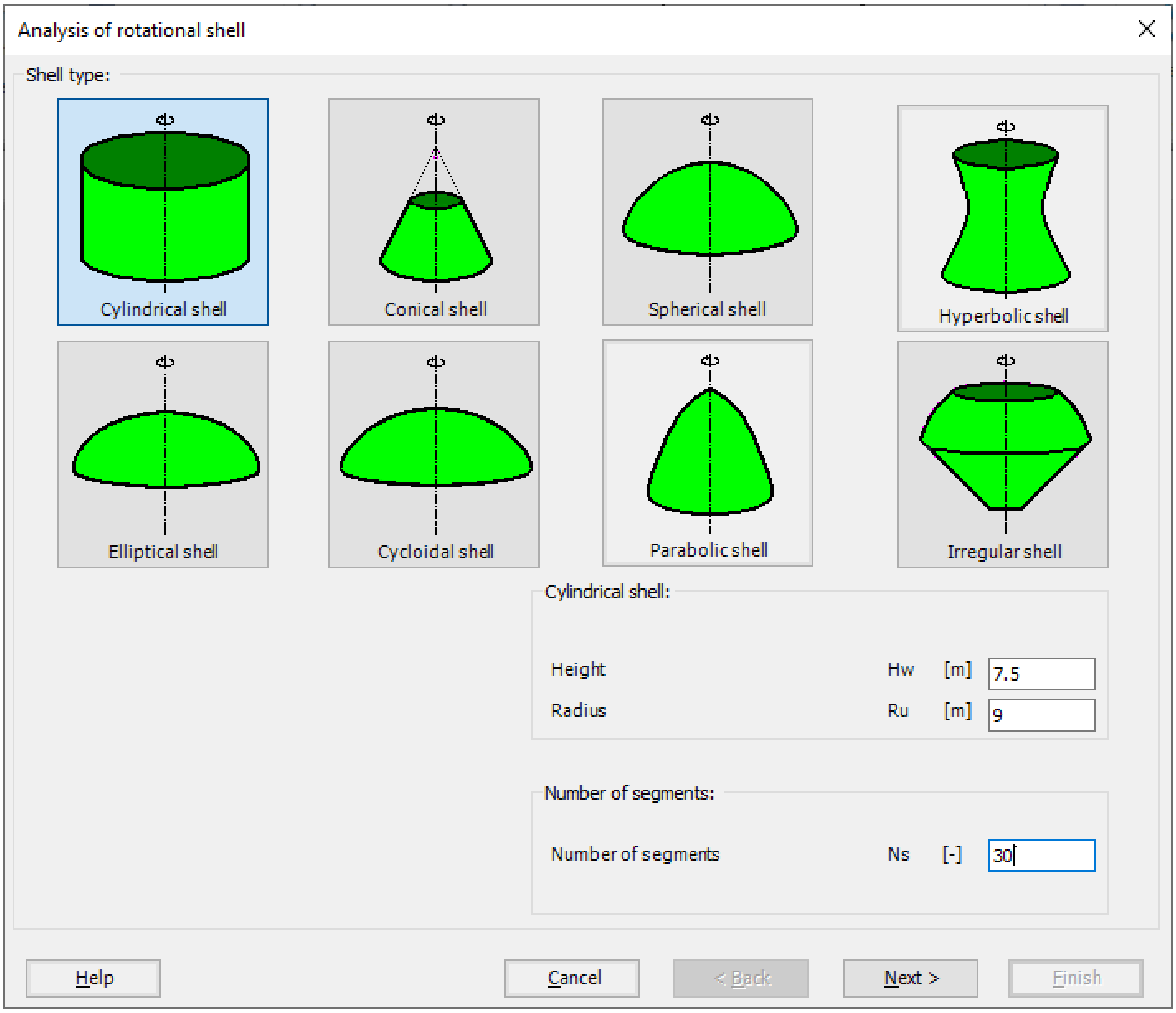


Figure 8.8 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Cylindrical shell" button
- Type 7.5 in the "Height" edit box
- Type 9 in the "Radius" edit box
- Type 30 in the "Number of segments" edit box
- Click "Next" button to go to the next Form
-

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Cylindrical shell" Form appears Figure 8.9, *ELPLA* divides the height of the tank into 30 equal segments, the user can edit the data of the segments individually by using "Modify" button, or all of them by using "In Table" button, if it is necessary.

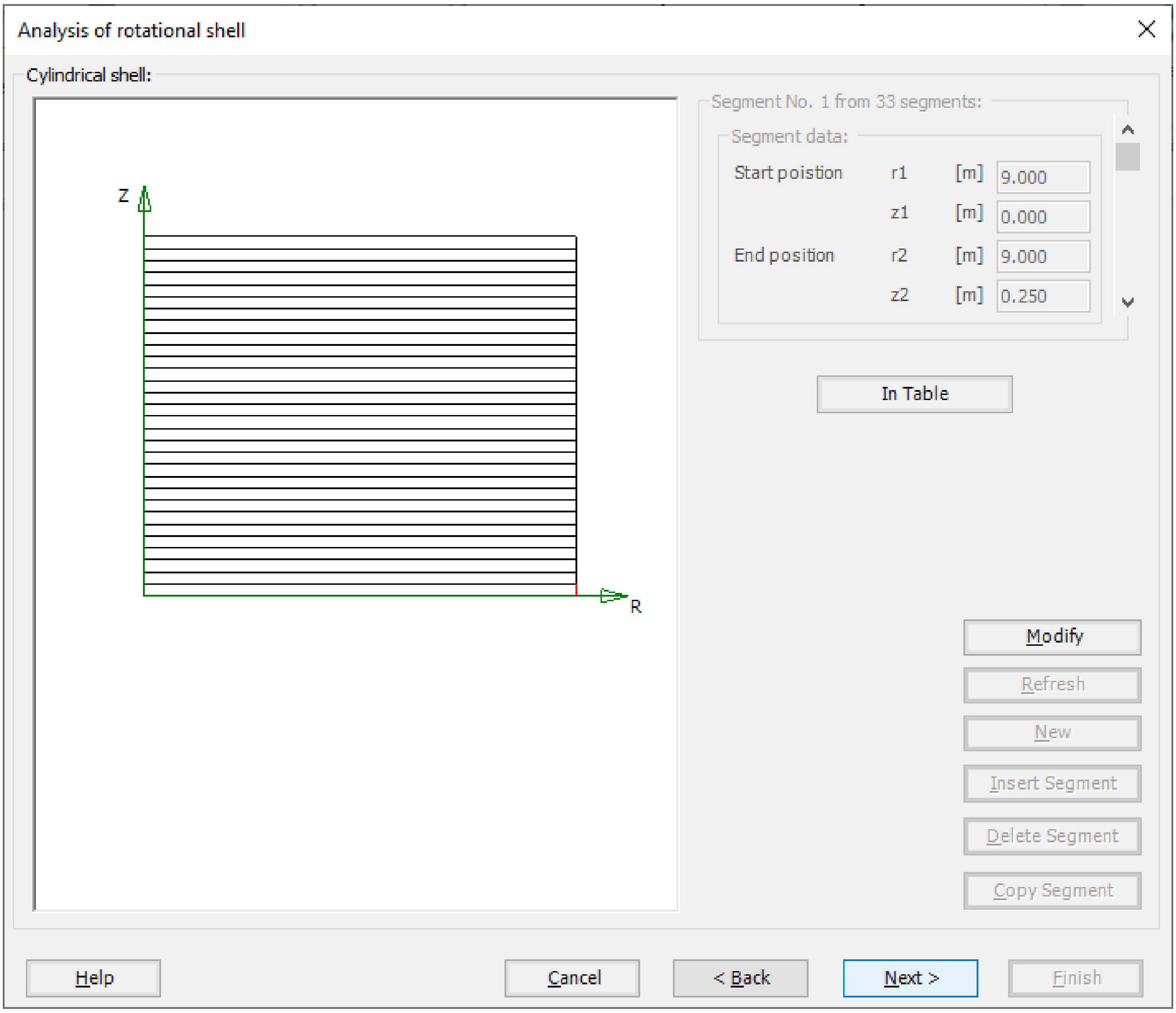


Figure 8.9 "Cylindrical shell" Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Net of Base" Form appears Figure 8.10.

To edit the grid spacing in *x*-direction, do the following steps in "Grid in *x*-direction" frame:

- Choose "Constant grid interval" check box
- Type 45 in the "No. of grid intervals" edit box, the base of the tank is divided into 45 equal segments, each of 0.2 [m]

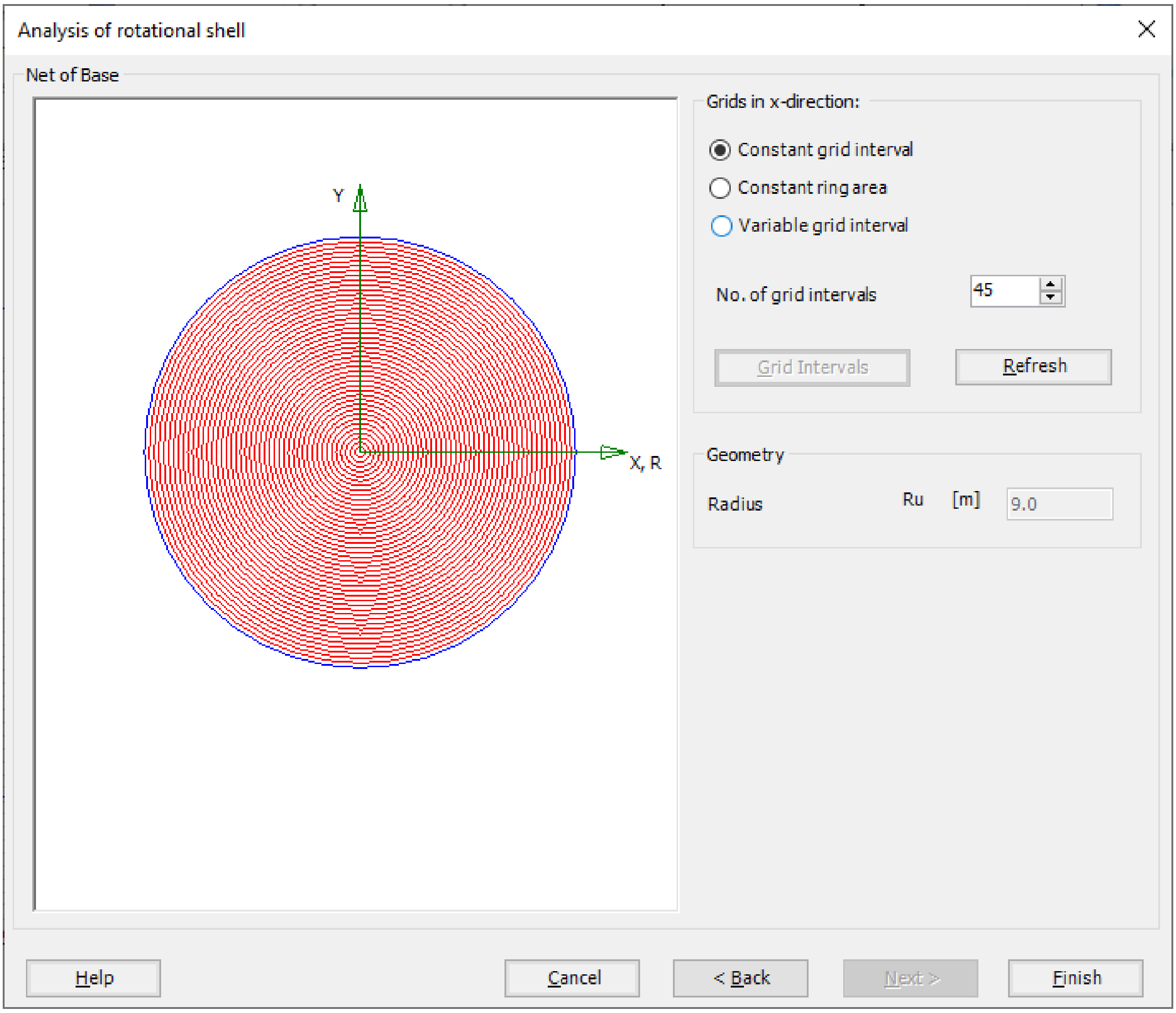


Figure 8.10 "Net of Base" Form

Click "Finish" button, the FE-Net of the tank wall and a sector from the base appears in Figure 8.11.

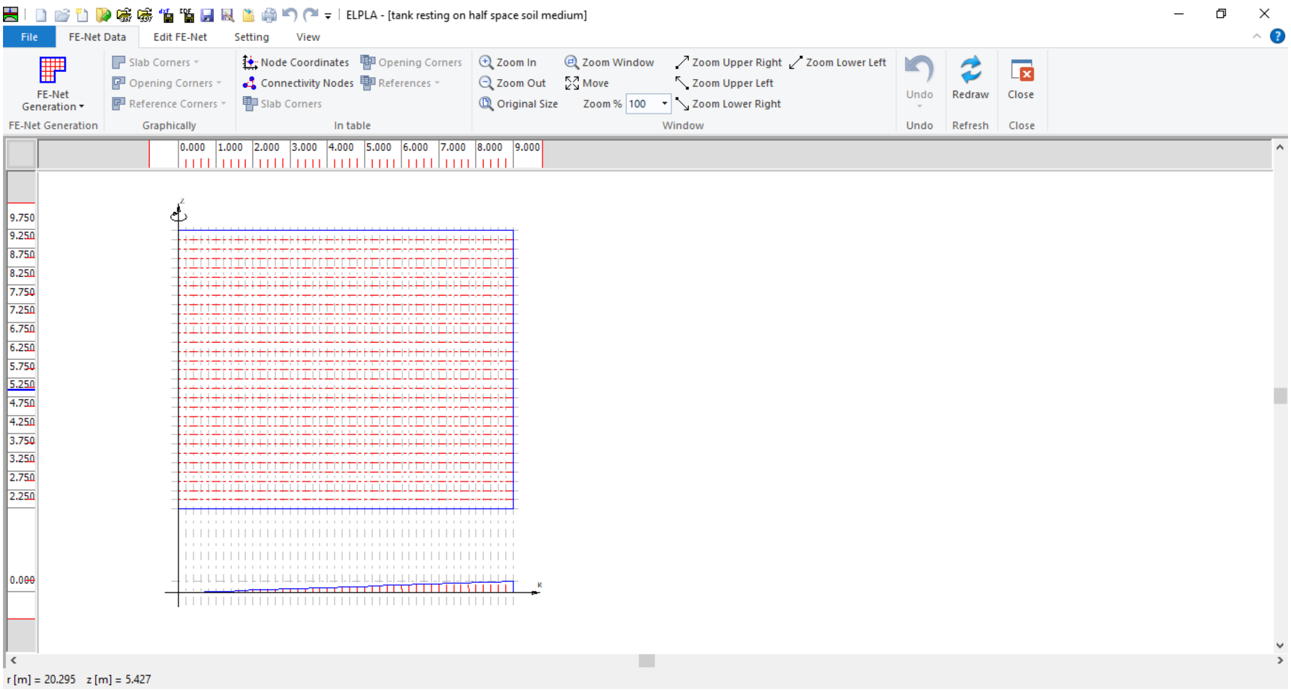


Figure 8.11 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 8.11 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 8.11 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Shell properties

To define the tank properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 8.12 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, unit weight of the tank, and filled material properties.

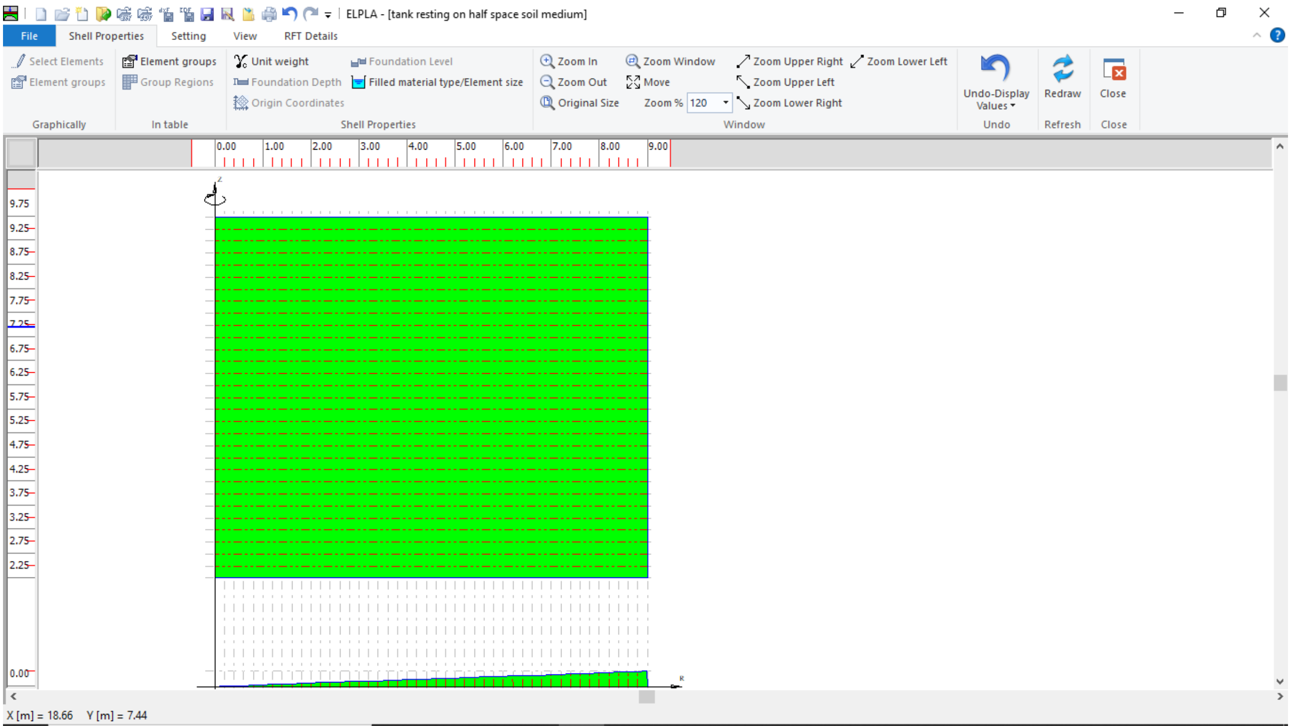


Figure 8.12 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 8.13 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness. Then click "OK" button.

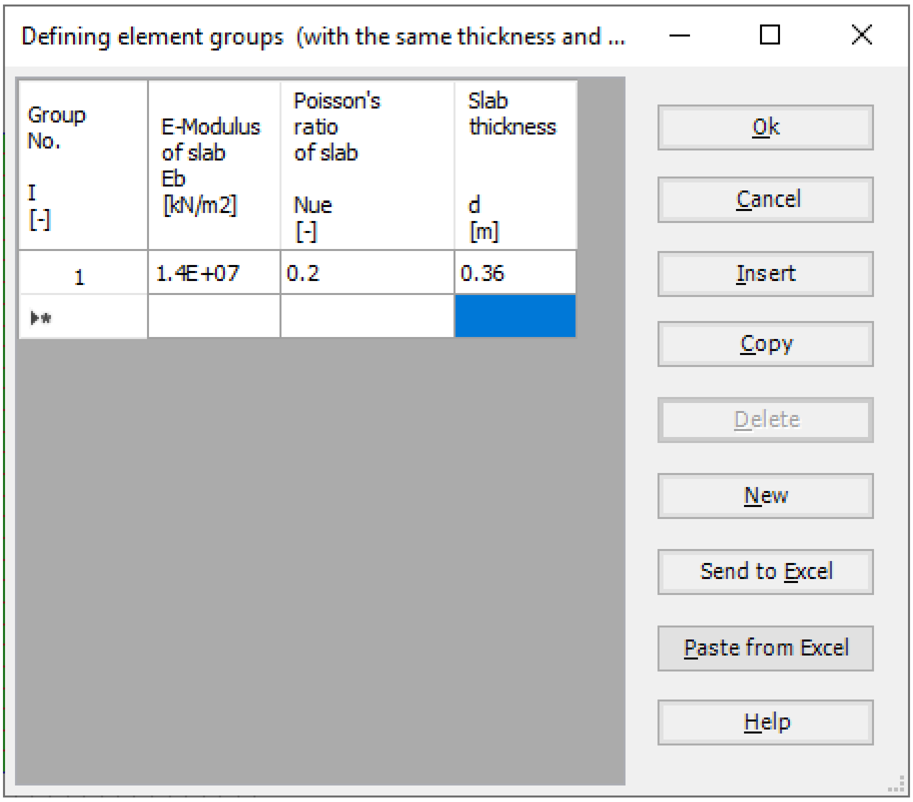


Figure 8.13 "Defining element groups" list box

To enter the unit weight of the tank, choose "Unit weight" command from "Shell Properties" menu in Figure 8.12. The following dialog box in Figure 8.14 with a default unit weight of 25 [kN/m^3] appears, click "OK" button.

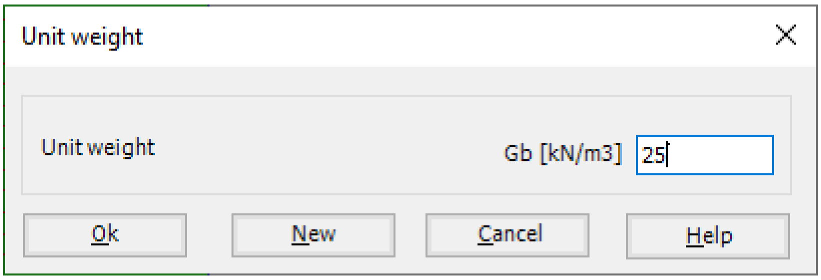


Figure 8.14 "Unit weight" dialog box

To define the liquid properties of the shell, choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 8.12. The following form in Figure 8.15 appears.

To define the filled material properties of the tank:

- Select the "Liquid container" option
- Type 7.5 in the "Height of the liquid" edit box
- Type 9.81 in the "Unit weight of the liquid" edit box

To define the element size of the ring:

- Check the "Constant element sizes in z-direction" check box
- Type 0.2 in the "Element size in each shell segment" edit box
- Click "OK" button

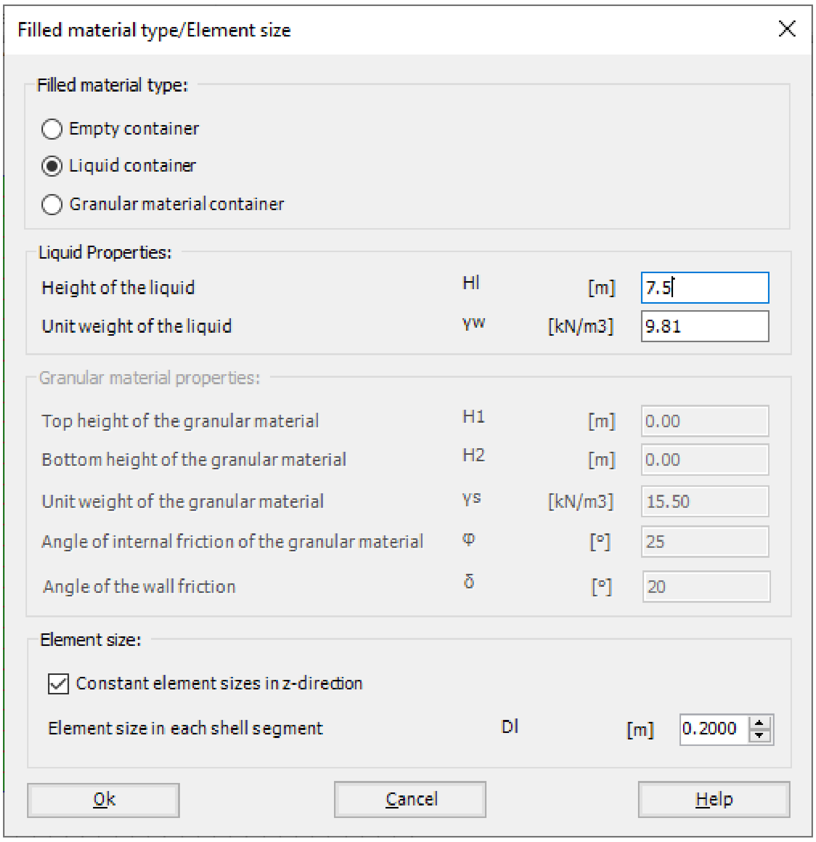


Figure 8.15 "Liquid Properties/Element size" Form

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

- Choose "Save" command from "File" menu in Figure 8.12 to save the shell properties
- Choose "Close" command from "File" menu in Figure 8.12 to close the "Shell properties" window and return to *ELPLA* main window

4.5 Soil Properties

To define the soil properties, choose "Soil Properties" command from “Data” Tab. The following "Soil Properties" form in Figure 8.16 appears, the soil properties are defined by Modulus of Elasticity "E", and is supposed to have the following parameters:

Modulus of Elasticity of the soil	*E*	= 20000	[kN/m^2]
Unit weight of the soil	*GAM*	= 18	[kN/m^3]
Angle of internal friction	*FHI*	= 30	[°]
Cohesion of the soil	c	= 0	[kN/m^2]
Poisson's ratio of the soil medium	*vs*	= 0.4	[-]

Other data in the example is not required, the user can use the default values.

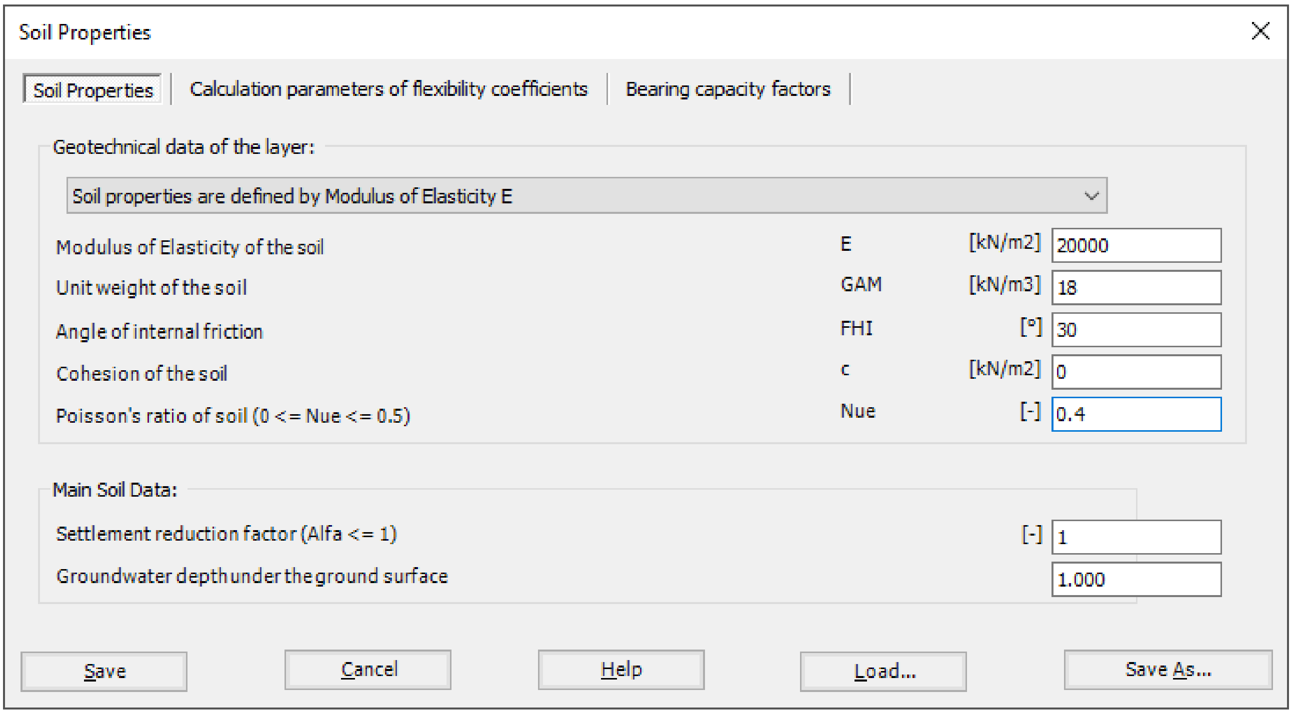


Figure 8.16 "Soil Properties" Form

After defining the soil properties, click "Save" button.

4.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 8.17 appears. In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 8.17. In this example, there is not applied load, as the vertical load has been already defined by the unit weight of the tank material, while the hydrostatic pressure on the tank wall is defined by the unit weight of water.

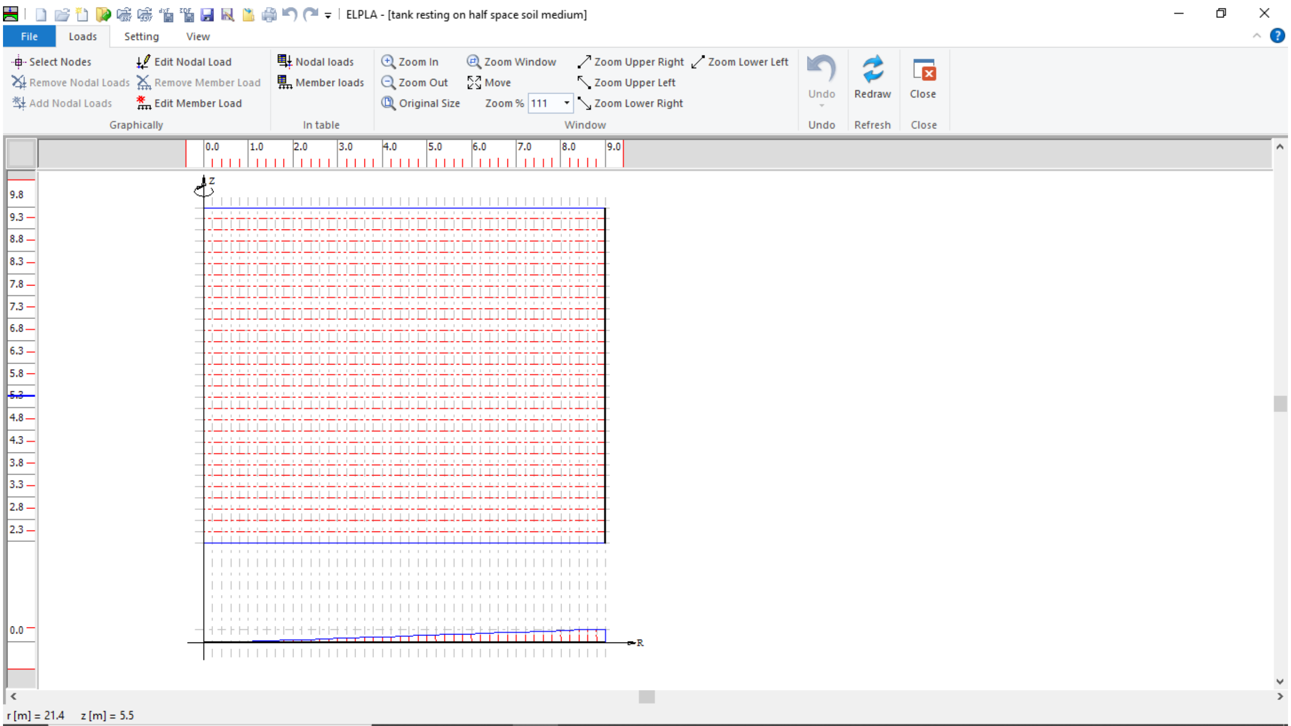


Figure 8.17 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 8.17 to save the load data
- Choose "Close" command from "File" menu in Figure 8.17 to close the "Loads" window and return to *ELPLA* main window

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

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 8.18.

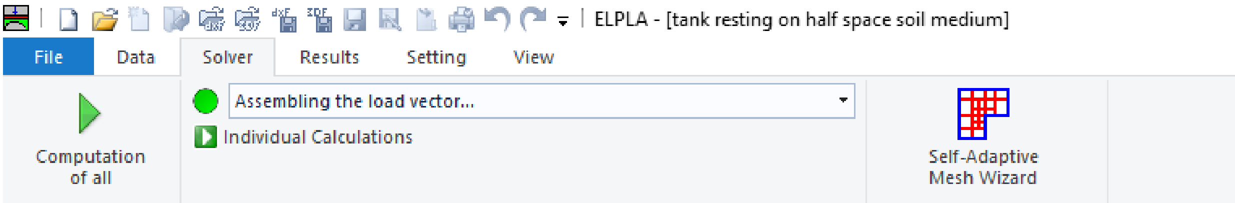


Figure 8.18 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

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

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

Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab Window. The following "Iteration parameters" option box in Figure 8.19 appears
- For this example, choose an accuracy of 0.0001 [m] to end the iteration process
- Click "OK" button

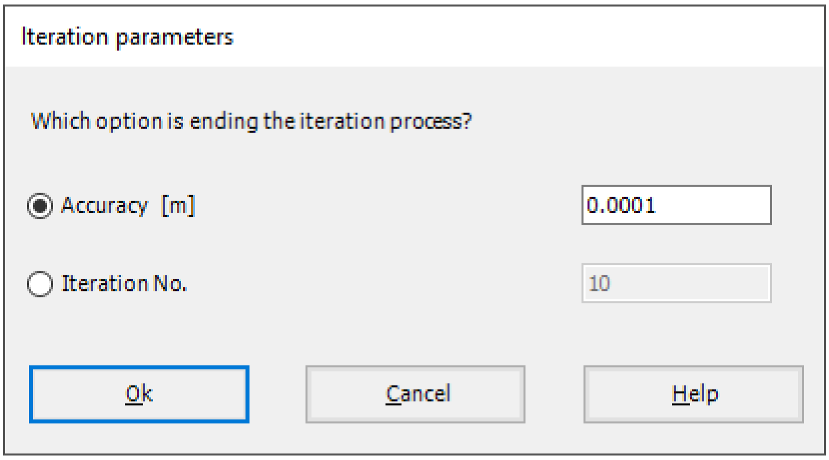


Figure 8.19 "Iteration parameters" option box

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

Analysis progress

Analysis progress menu in Figure 8.20 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

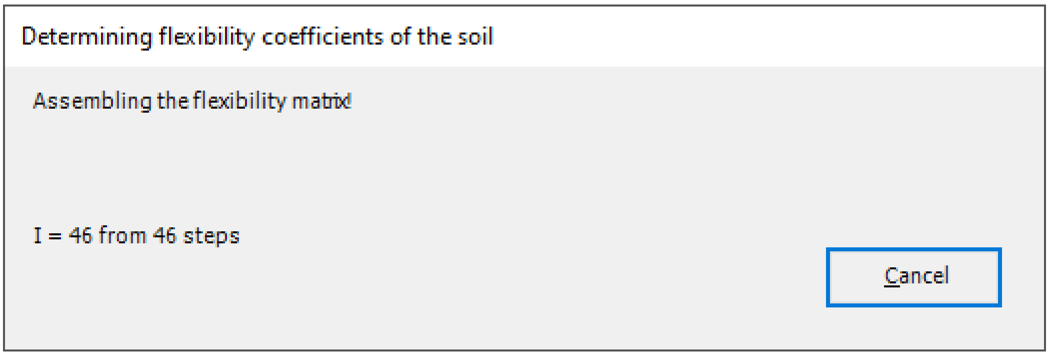


Figure 8.20 Analysis progress menu

"Check of convergence" message Figure 8.21 appears showing that no convergence is reached at the last step, click "Ok" button.

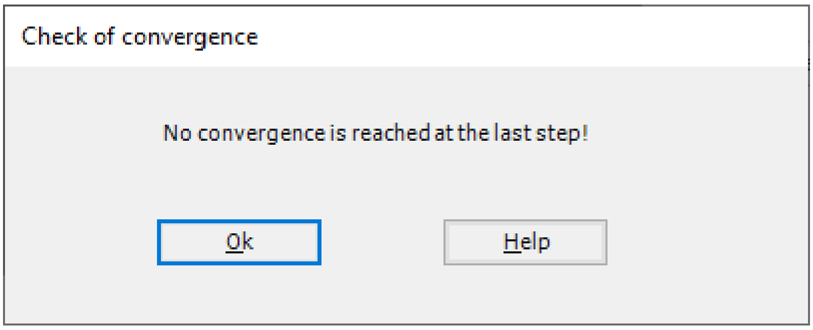


Figure 8.21 "Check of convergence" message

Click "Stop" button (Figure 8.22), to stop the iteration process as no convergence has reached.

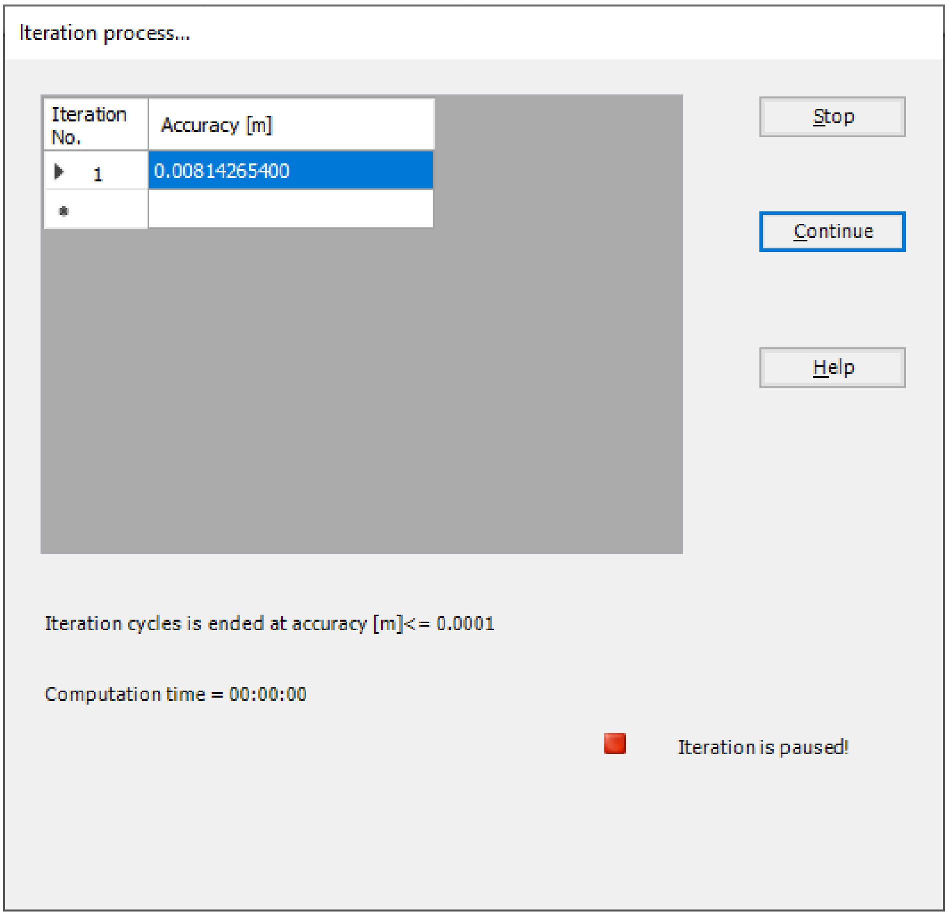


Figure 8.22 "Iteration process" list box

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 8.23. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

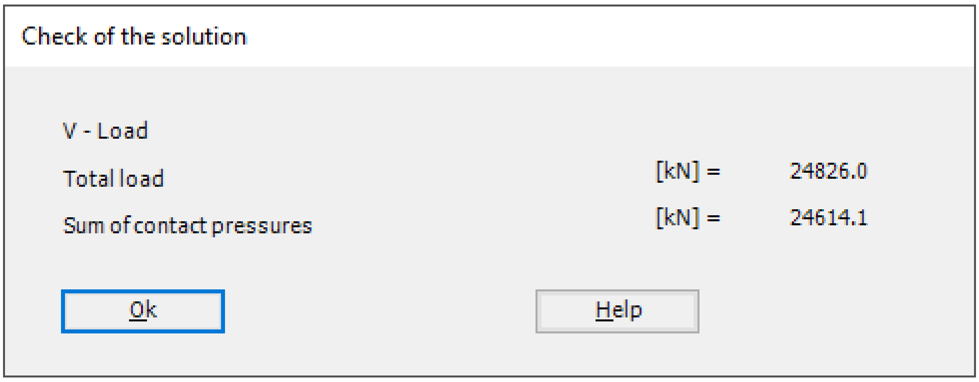


Figure 8.23 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab. To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 8.24).

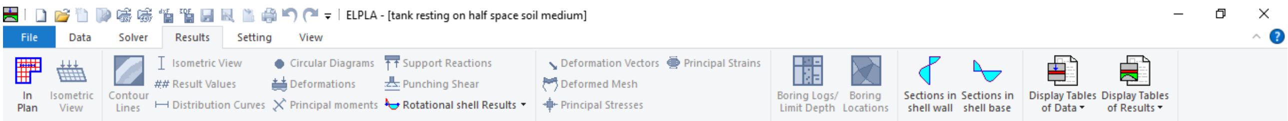


Figure 8.24 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Sections in shell base
- Sections in shell wall
- Display tables of data
- Display tables of results

To view the radial forces in the shell wall

- Choose "Sections in shell wall" command from "Section" menu. The following option box in Figure 8.25 appears
- In the "Sections in shell wall" option box, select "Radial forces *Nr*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 8.26.

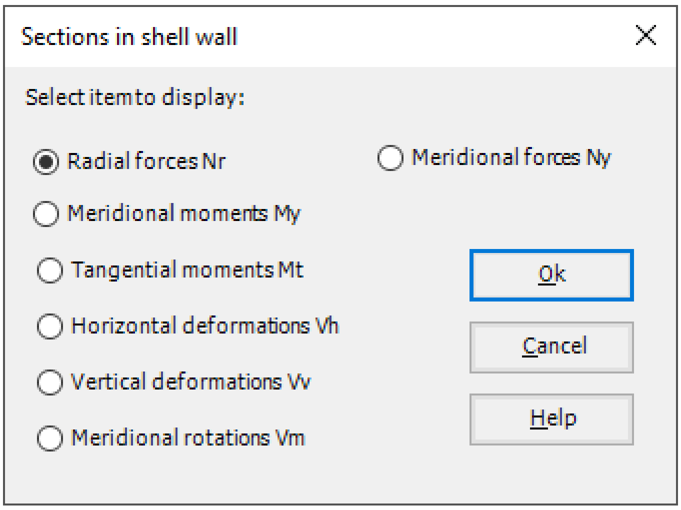


Figure 8.25 "Sections in shell wall" option box

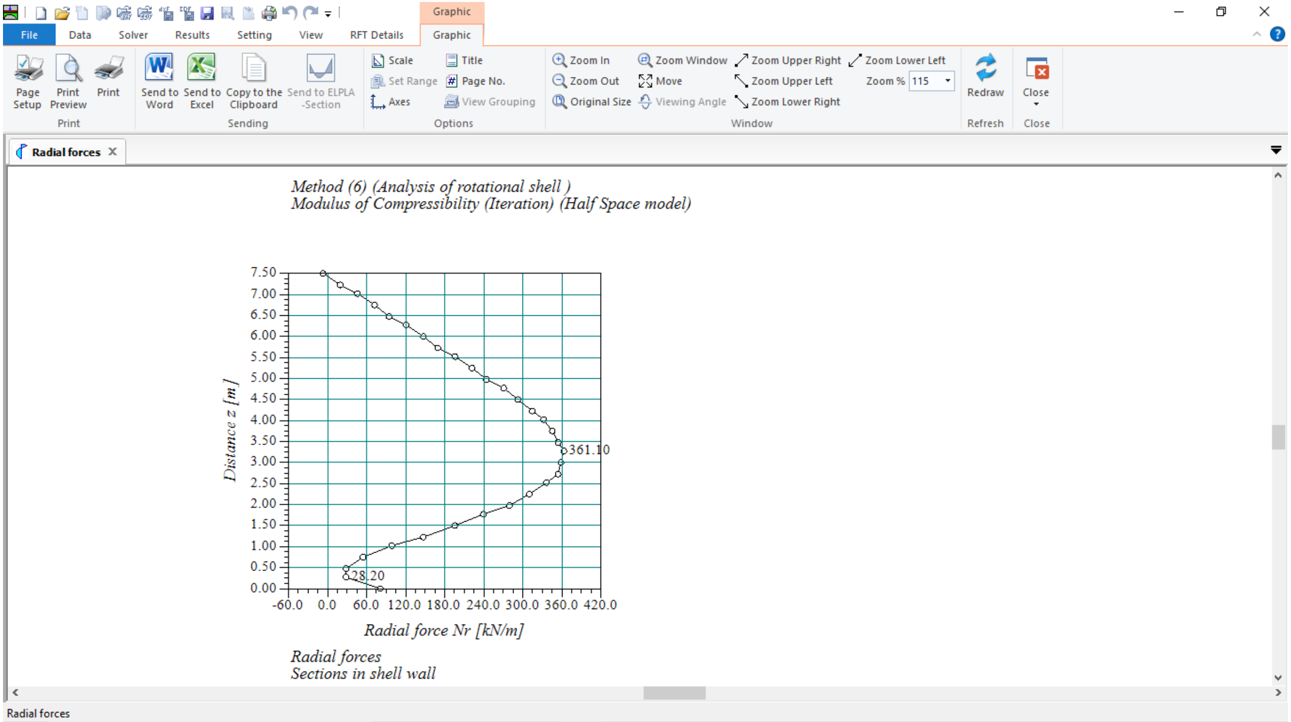


Figure 8.26 Radial forces in shell wall

To view the meridional moments in the shell base

- Choose "Sections in shell base" command from "Section" menu. The following option box in Figure 8.27 appears
- In the "Sections in shell base" option box, select "Meridional moments *My*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 8.28.

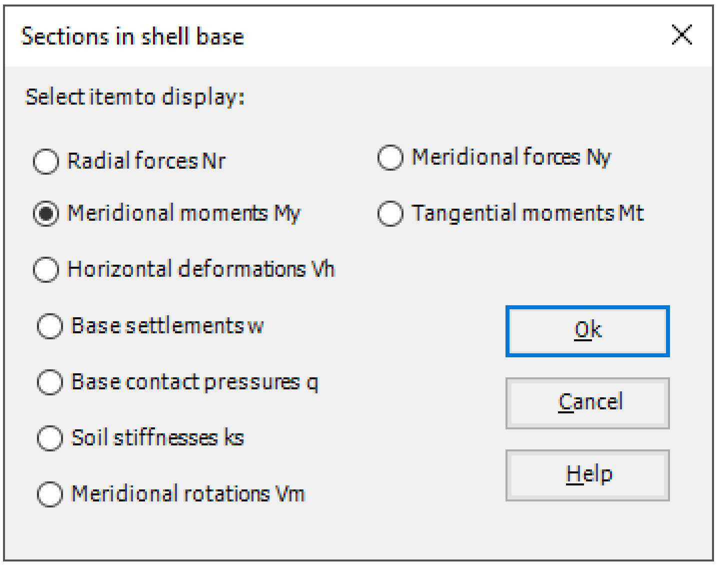


Figure 8.27 "Sections in shell base" option box

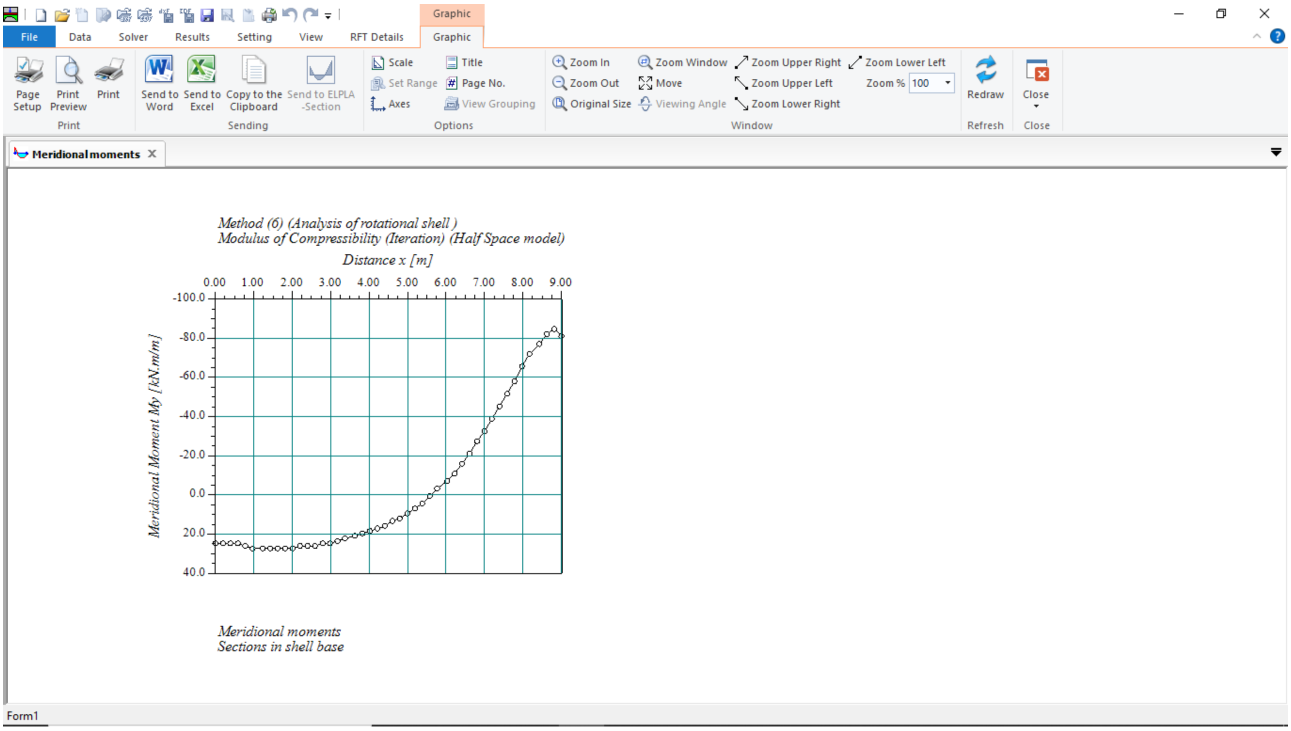


Figure 8.28 Meridional moments in shell base

To view element groups of the tank

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 8.29 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

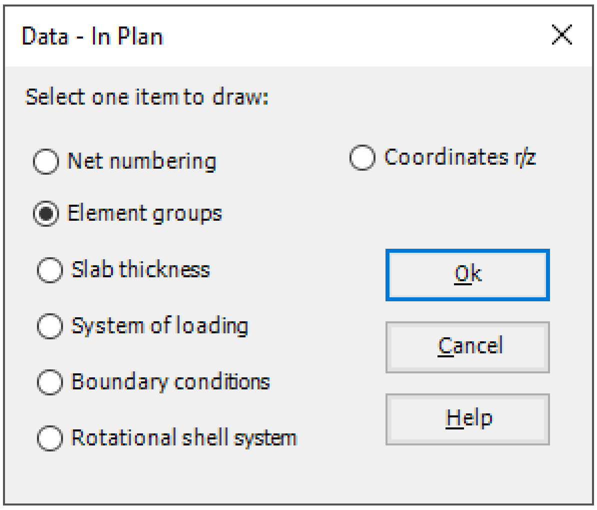


Figure 8.29 "Data – In Plan" option box

To view the meridional moments on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 8.30 appears
- In this check group box, check "Meridional moments *My*" check box
- The user can choose any other data to be displayed
- Click "OK" button

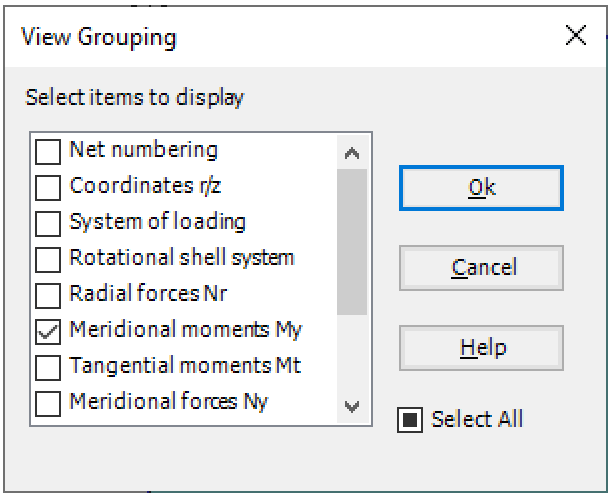


Figure 8.30 "View Grouping" check group box

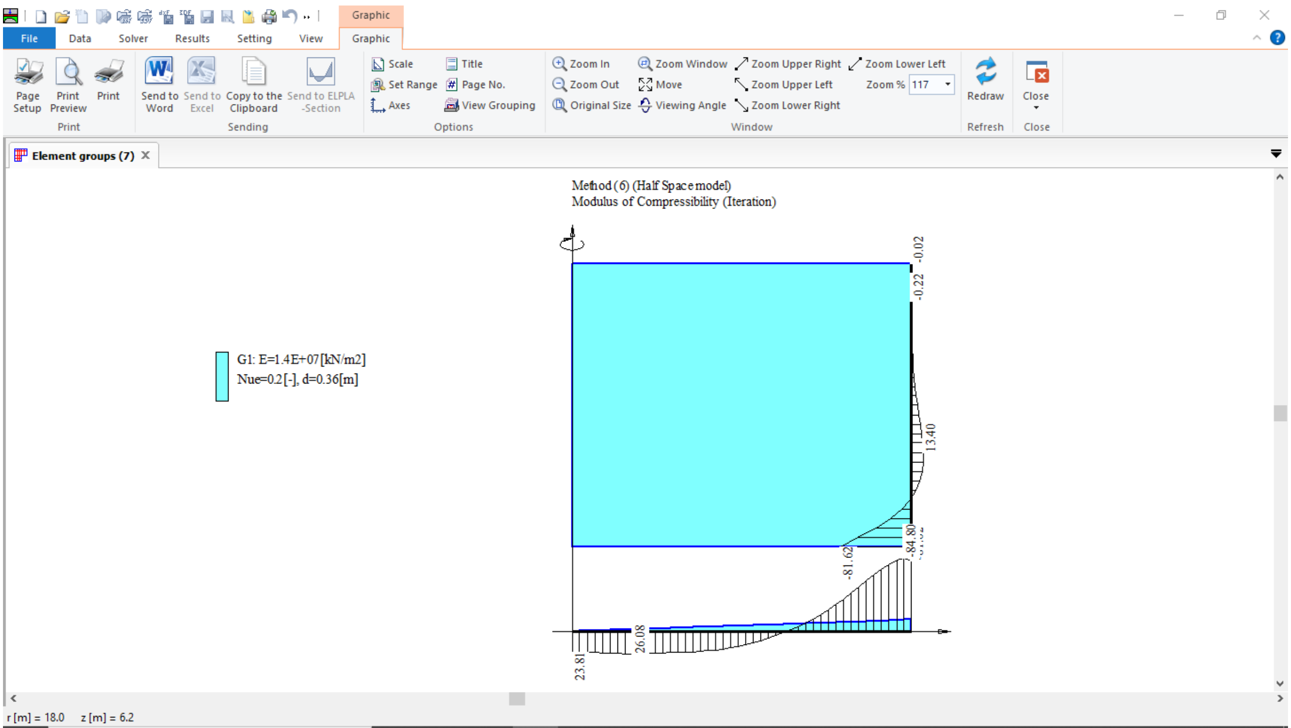


Figure 8.31 Element groups

Example 9

Analysis of a tank with a different base thickness resting on half space soil medium

Contents Page

1 Description of the problem

An example of a circular cylindrical storage tank resting on an isotropic elastic half space soil medium is selected to and illustrate some features of *ELPLA* for analyzing circular cylindrical shell elements.

2 Tank geometry and properties

A circular cylindrical tank of an inner diameter of d = 20 [m] and a height of H = 10 [m] is considered as shown in Figure 9.1. Thicknesses of the wall and the base are different. The thickness of the tank wall is t_w = 0.2 [m] and that of the base is t_b = 0.5 [m]. The tank is filled with water. Figure 9.1 shows the storage tank, while the tank material and unit weight of the water are listed in Table 9.1. The data of soil medium under the base of the tank are shown in Table 9.2.

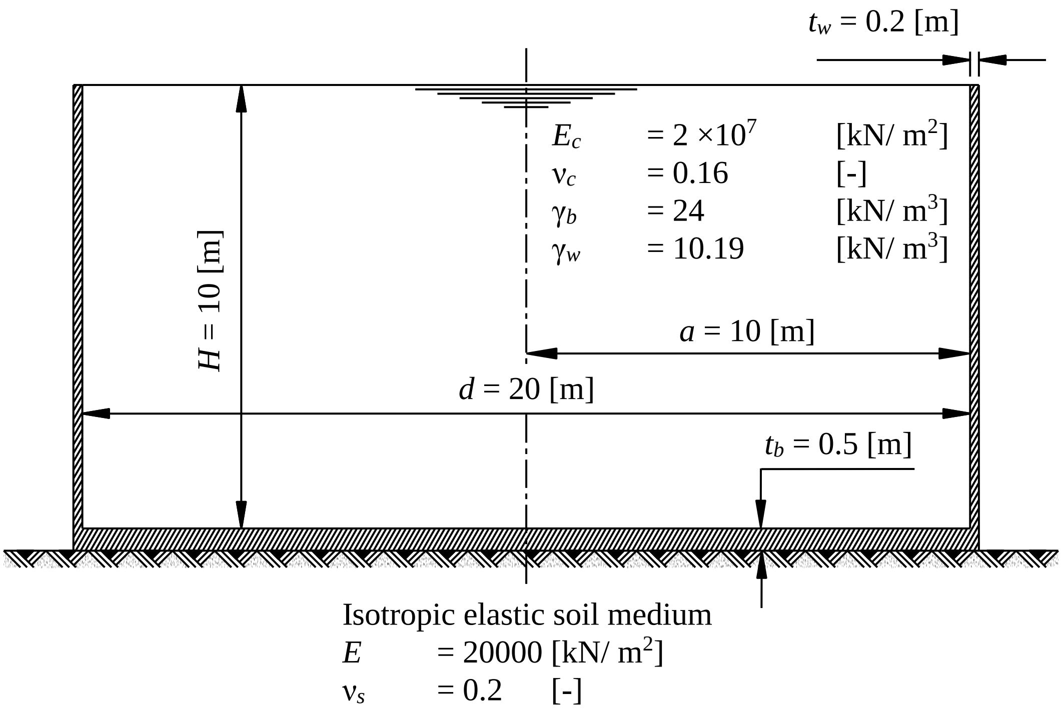


Figure 9.1 Circular cylindrical tank resting on an isotropic elastic soil medium

Table 9.1 Tank material and water unit weight

Modulus of Elasticity of the tank material	E_c	= 2×10^7	[kN/ m^2]
Poisson's ratio of the tank material	ν_b	= 0.16	[-]
Unit weight of the tank material	γ_b	= 24	[kN/ m^3]
Unit weight of the water	γ_w	= 10.19	[kN/ m^3]

Table 9.2 Soil data

Modulus of Elasticity of the soil medium	E	= 20000	[kN/m^2]
Poisson's ratio of the soil medium	ν_s	= 0.2	[-]

3 Numerical Analysis

In order to analyze a water storage tank resting on an isotropic elastic half space soil medium using *ELPLA*, this example shown in Figure 9.1 is analyzed. The height of the tank is divided into 50 equal segments, each of 0.20 [m], as shown in Figure 9.2, while the half base of the tank is divided into 50 equal segments, each of 0.20 [m].

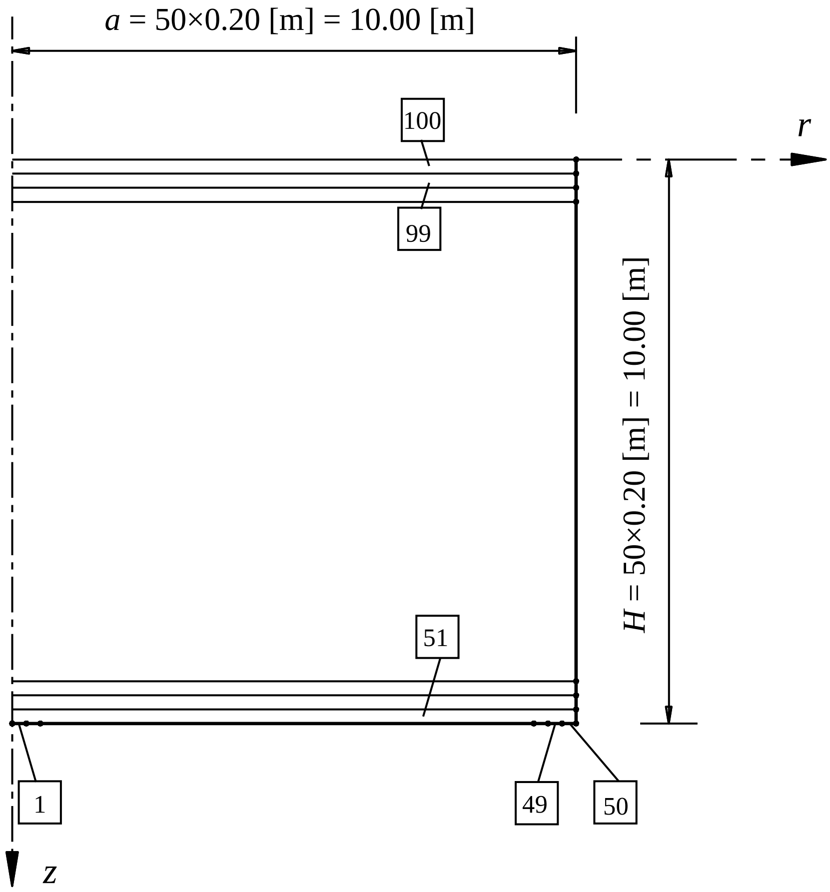


Figure 9.2 Finite element mesh of the tank

4 Creating the project

In this section, the user will learn how to create a project for analyzing a tank resting on an isotropic elastic half space soil medium. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 9.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of forms. The first form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 9.3).

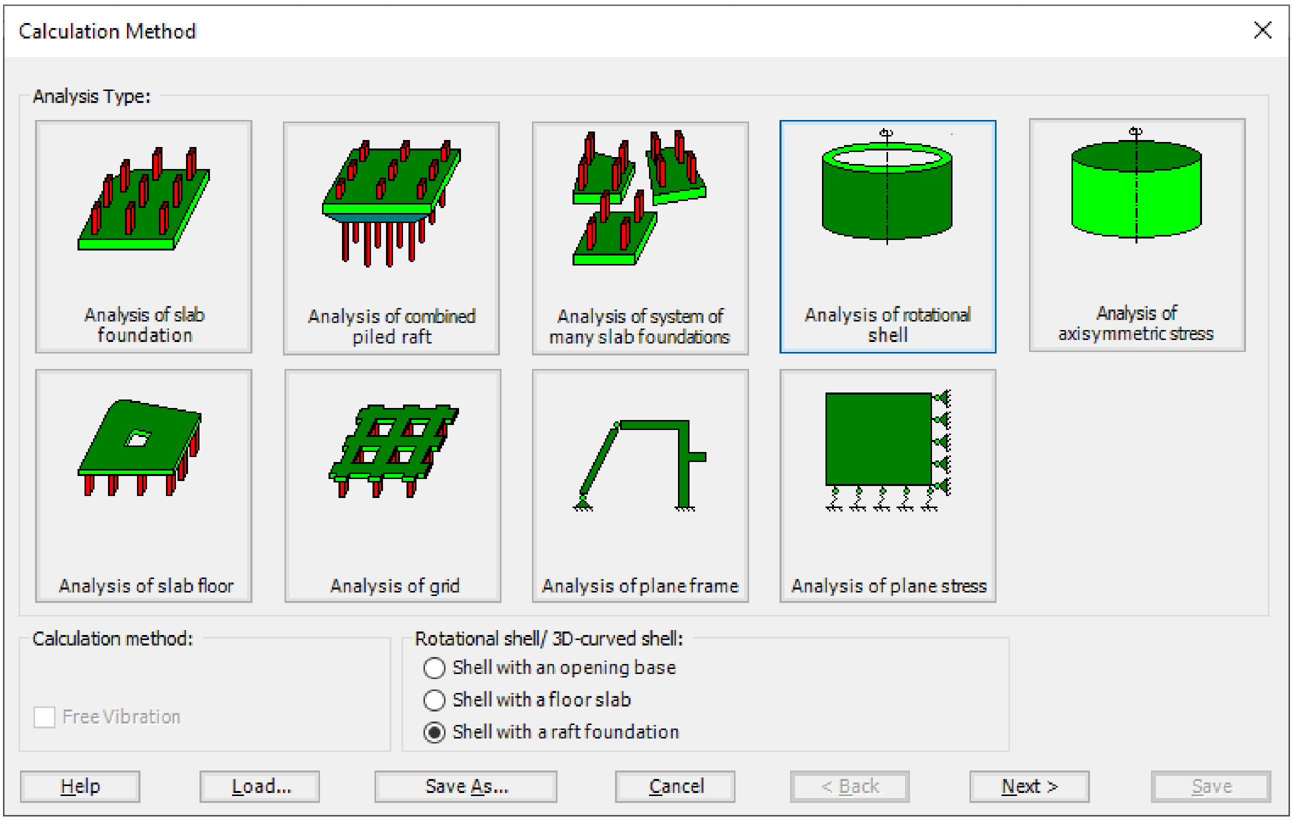


Figure 9.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 9.3, define the analysis type of the problem. As the analysis type is a tank resting on an isotropic elastic half space soil medium problem, select "Analysis of rotational Shell" button, and check "Shell with a raft foundation" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Calculation Method" Form appears, Figure 9.4.

To define the calculation method:

- Select the calculation method "4-Modification of Modulus of Subgrade Reaction by Iteration"
- To determine the subsoil model, select "Half Space model"
- Click "Next" button to go to the next Form

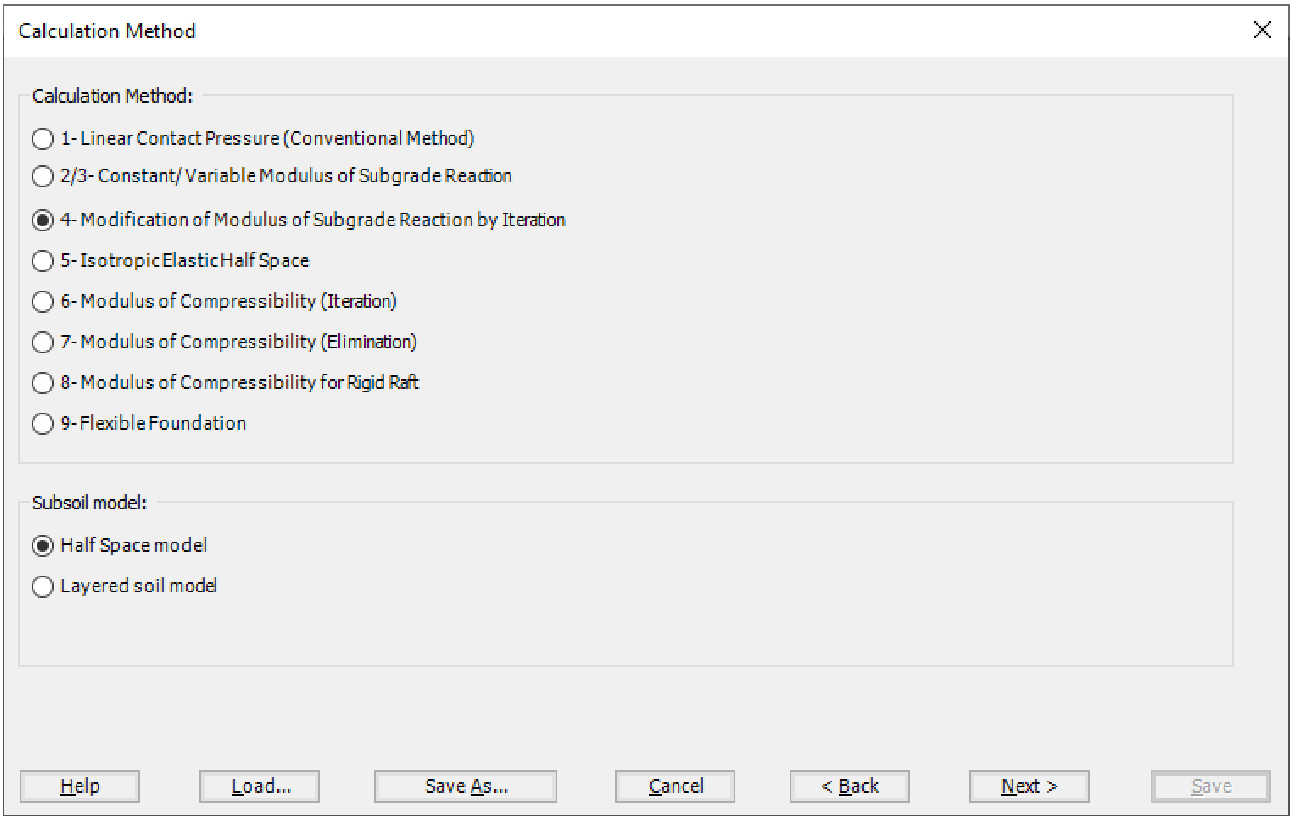


Figure 9.4 "Calculation Method" Form

The last Form in the wizard is the "Options" Form, Figure 9.5. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Since no option will be considered in the analysis, click the "Save" button.

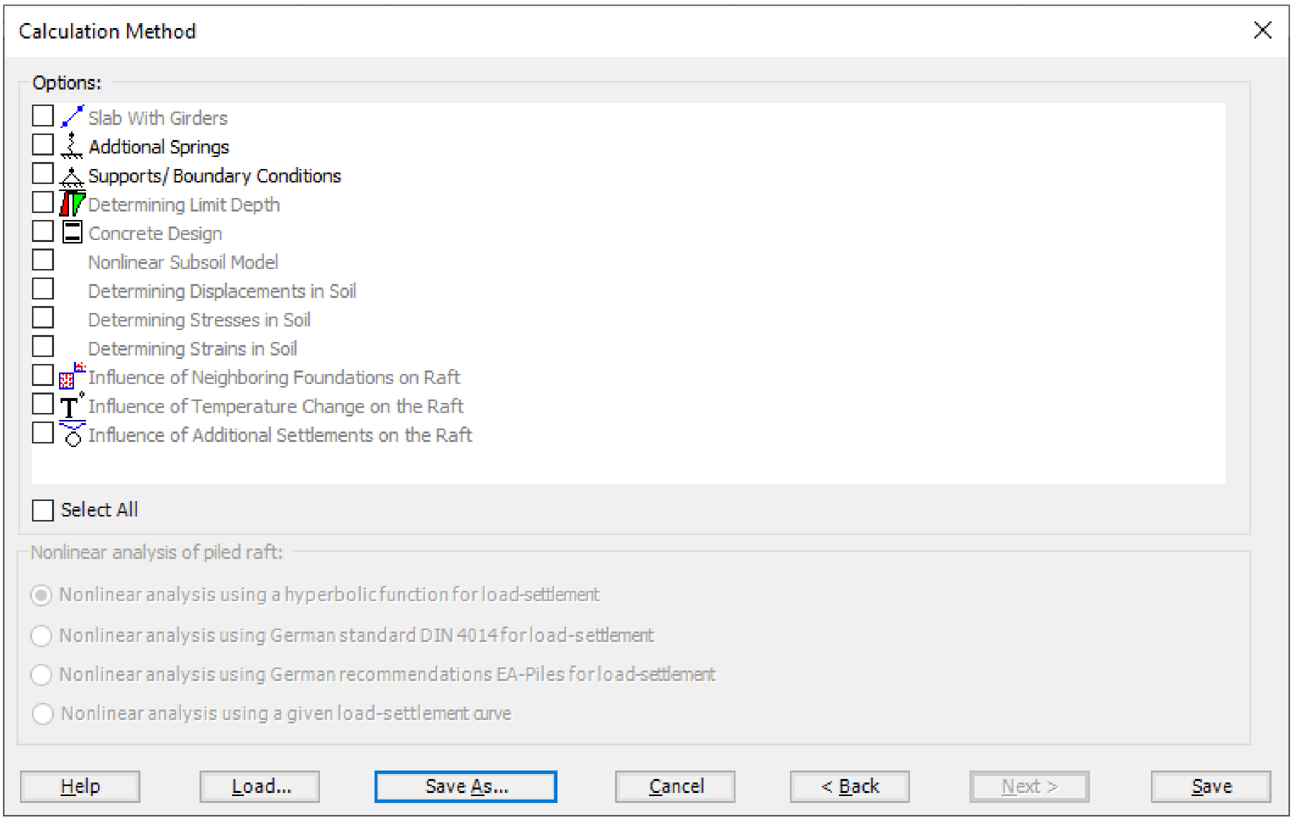


Figure 9.5 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 9.6. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Tank with different base thickness". *ELPLA* will use automatically this file name in all reading and writing processes.

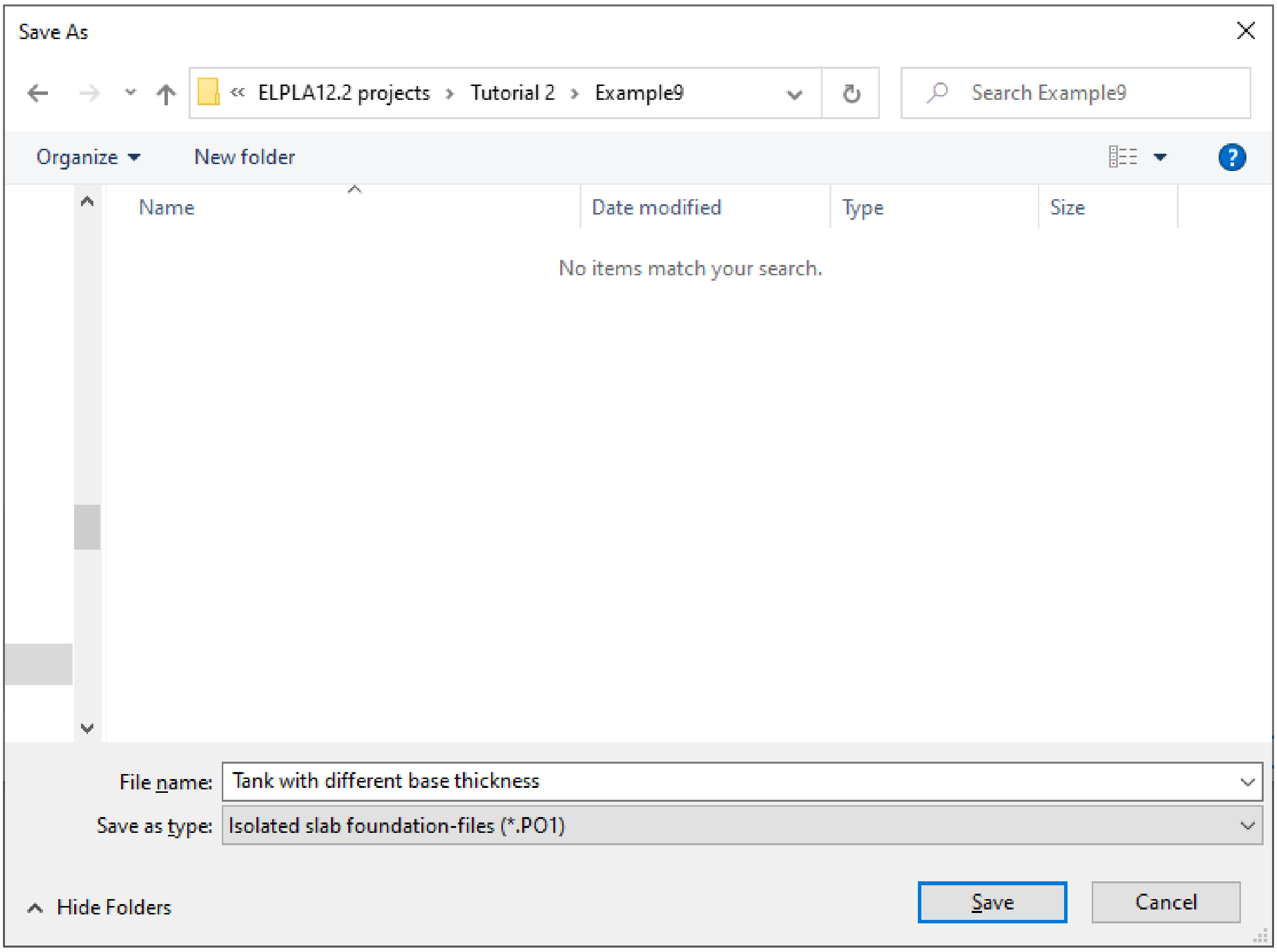


Figure 9.6 "Save as" dialog box

ELPLA will activate the "Data" Tab. In addition, the file name of the current project [Tank with different base thickness] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 9.7 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a tank resting on a half space soil medium"
- Type the date of the project in the "Date" edit box
- Type "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

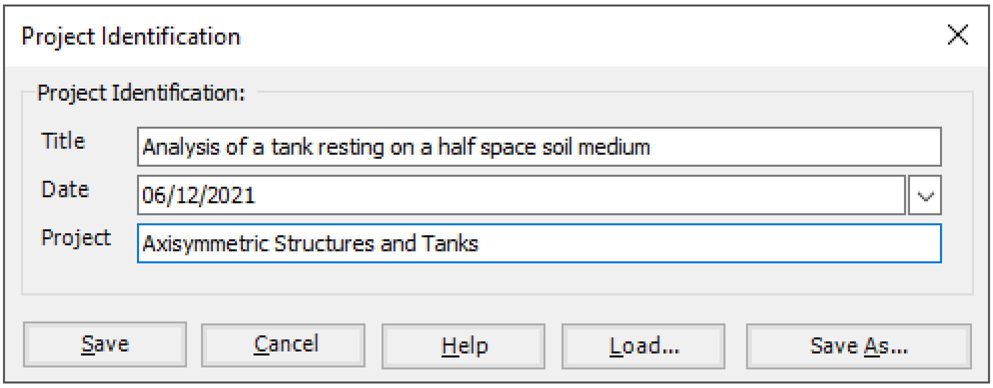


Figure 9.7 "Project Identification" dialog box

4.3 FE-Net data

For the given problem, a circular cylindrical tank of an inner diameter of d = 20 [m] and a height of H = 10 [m], the height of the tank is divided into 50 equal segments, each of 0.20 [m], while the half base of the tank is divided into 50 equal segments, each of 0.20 [m]. To define the FE-Net for this tank, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 9.8. This wizard will guide you through the steps required to generate a FE-Net.

The first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

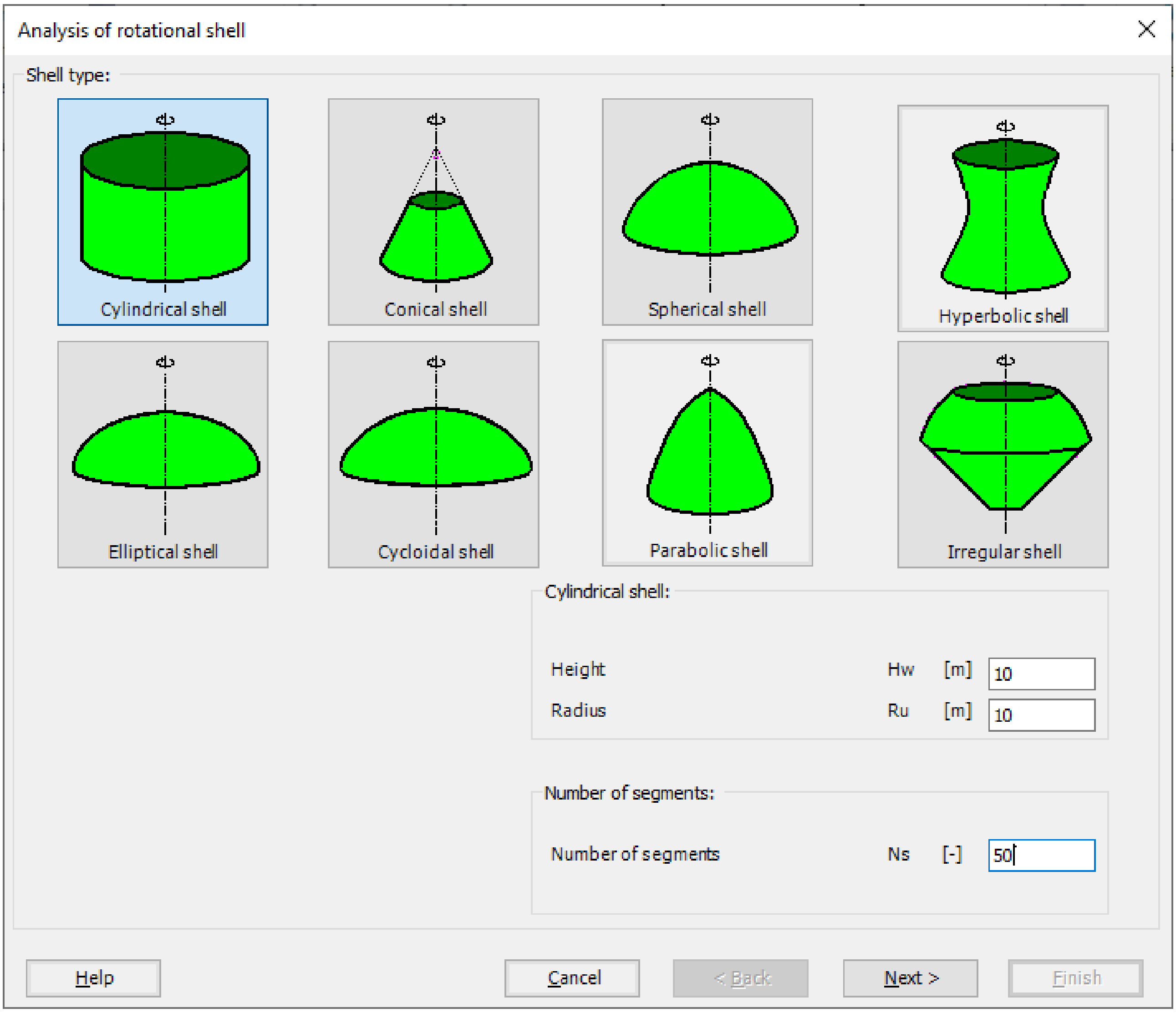


Figure 9.8 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Cylindrical shell" button
- Type 10 in the "Height" edit box
- Type 10 in the "Radius" edit box
- Type 50 in the "Number of segments" edit box
- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Cylindrical shell" Form appears Figure 9.9, *ELPLA* divides the height of the tank into 50 equal segments, the user can edit the data of the segments individually by using "Modify" button, or all of them by using "In Table" button, if it is necessary.

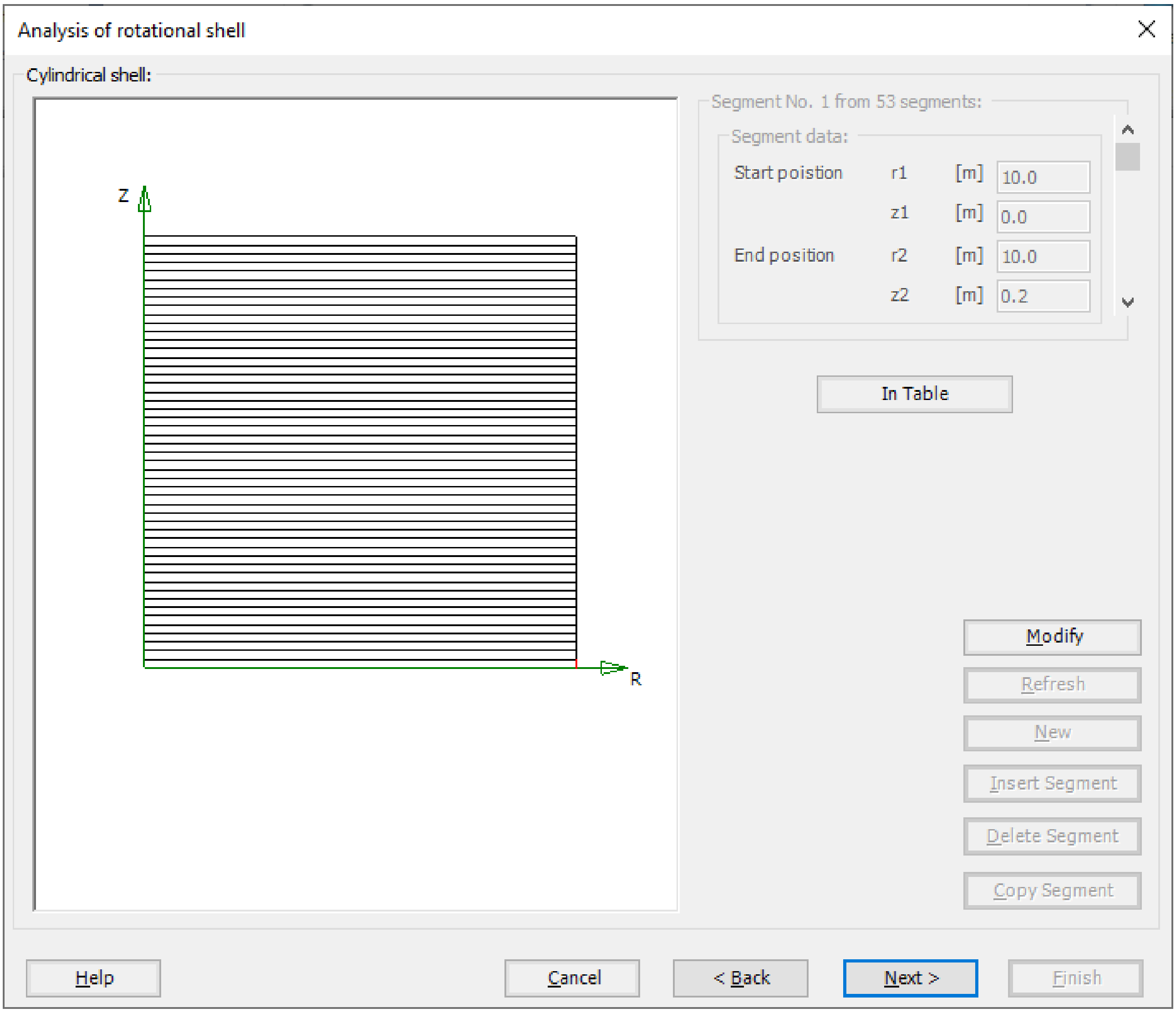


Figure 9.9 "Cylindrical shell" Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Net of Base" Form appears Figure 9.10.

To edit the grid spacing in *x*-direction, do the following steps in "Grid in *x*-direction" frame:

- Choose "Constant grid interval" check box
- Type 50 in the "No. of grid intervals" edit box, the base of the tank is divided into 50 equal elements, each of 0.2 [m]

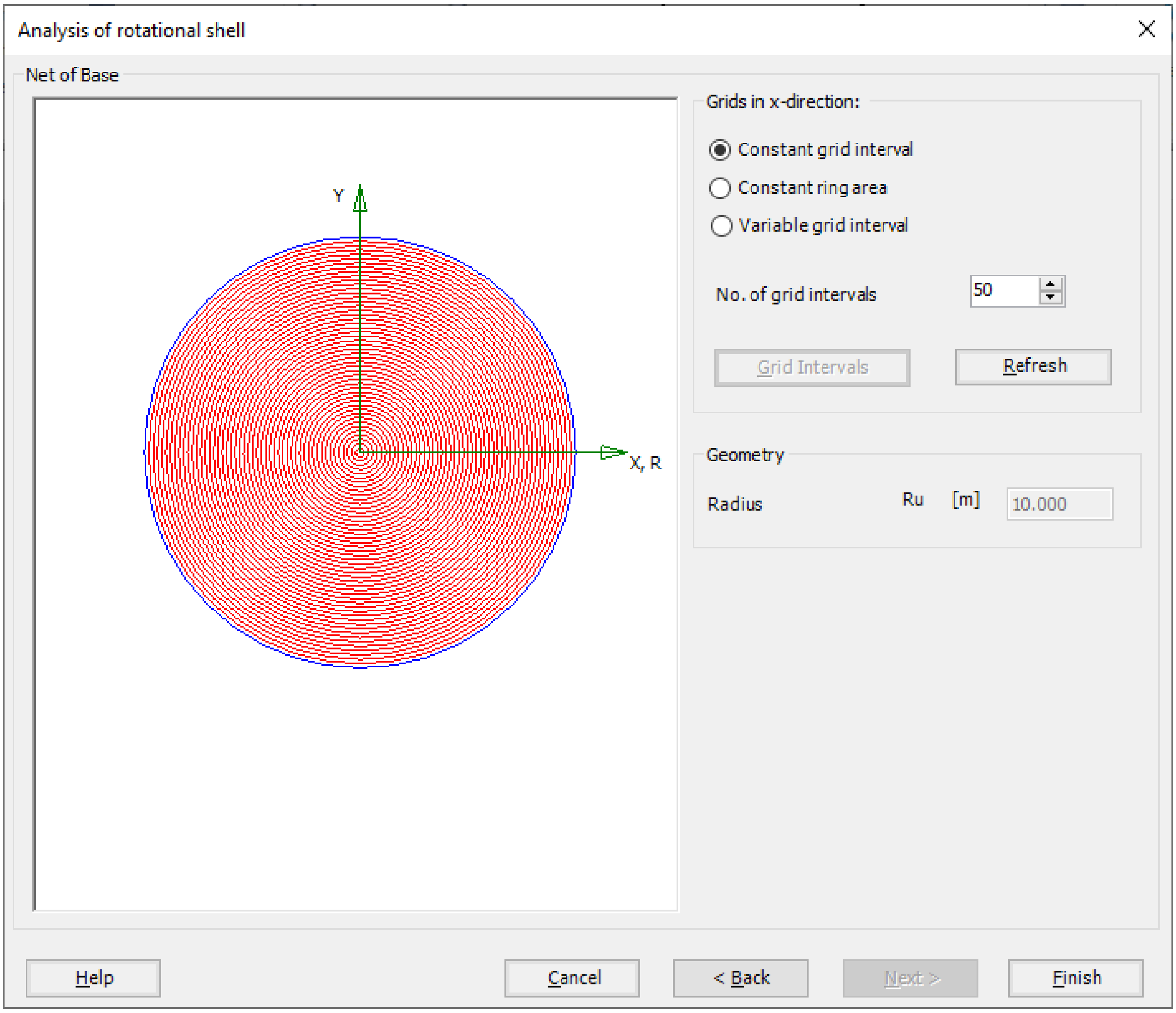


Figure 9.10 "Net of Base" Form

Click "Finish" button, the FE-Net of the tank wall and a sector from the base appears in Figure 9.11.

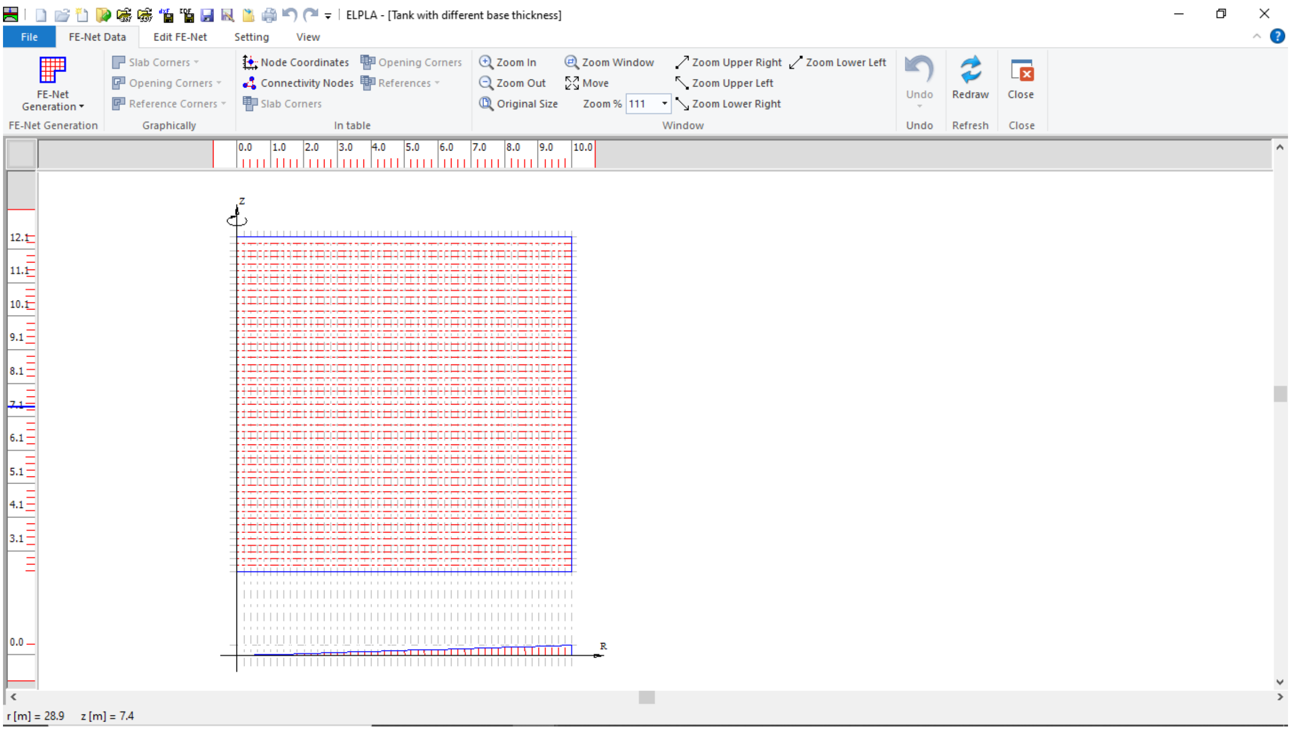


Figure 9.11 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 9.11 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 9.11 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Shell properties

To define the tank properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 9.12 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, group regions, unit weight of the tank, and the filled material properties.

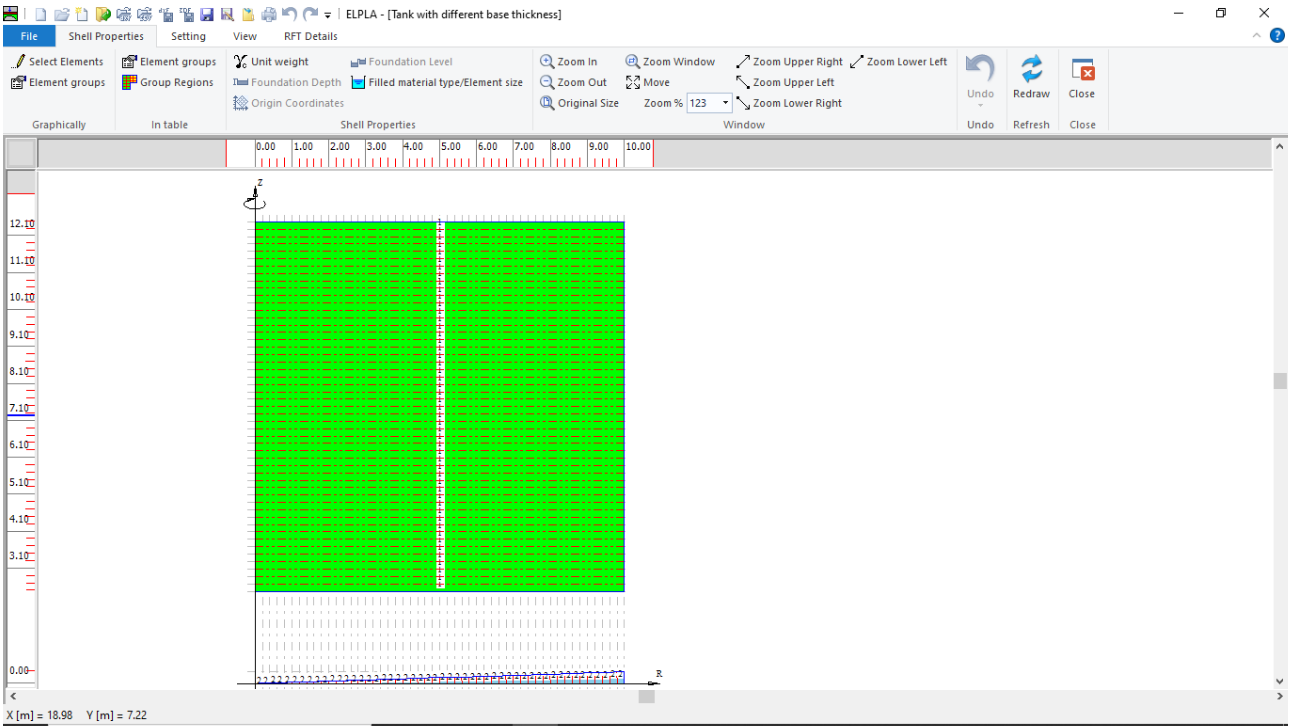


Figure 9.12 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 9.13 appears. In this list box, define E-Modulus, *Poisson's* ratio and slab thickness for both the tank wall and the tank base, as they differ in thickness. Then click "OK" button.

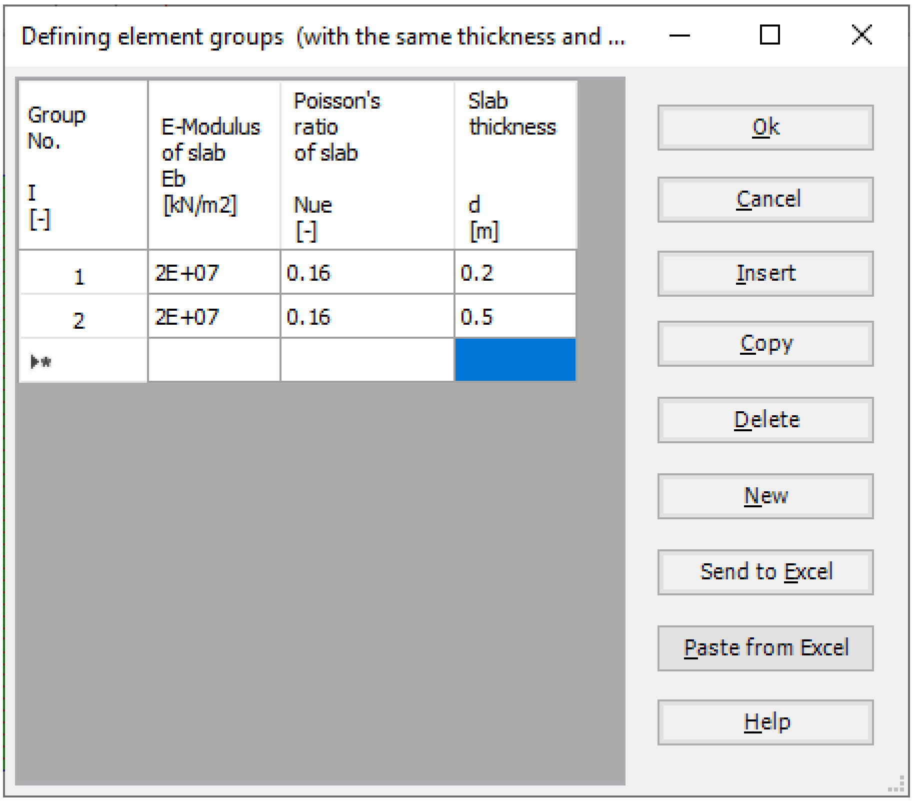


Figure 9.13 "Defining element groups" list box

Defining the slab thickness for materials on the net may be carried out either graphically or numerically (in a table). In the current example, the user will define the slab thickness on the net graphically.

To define the slab thickness for the tank base

- Choose "Select Elements" command from "Graphically" menu in the window of Figure 9.12.
- When "Select Elements" command is chosen, the cursor will change from an arrow to a cross hair. A group of elements can be selected by holding the left mouse button down at the corner of the region. Then, drag the mouse until a rectangle encompasses the required group of elements. When the left mouse button is released, all elements in the rectangle are selected
- Select the elements that include the tank base as shown in Figure 9.14
- Choose "Elements Groups" command from "Graphically" menu in the window of Figure 9.12, "Group Regions" dialog box Figure 9.15 appears
- Define the "Group No." as type "2", while "Group No." of the tank wall elements will be as type "1", where type "1" is the default "Group No.", then click "OK" button

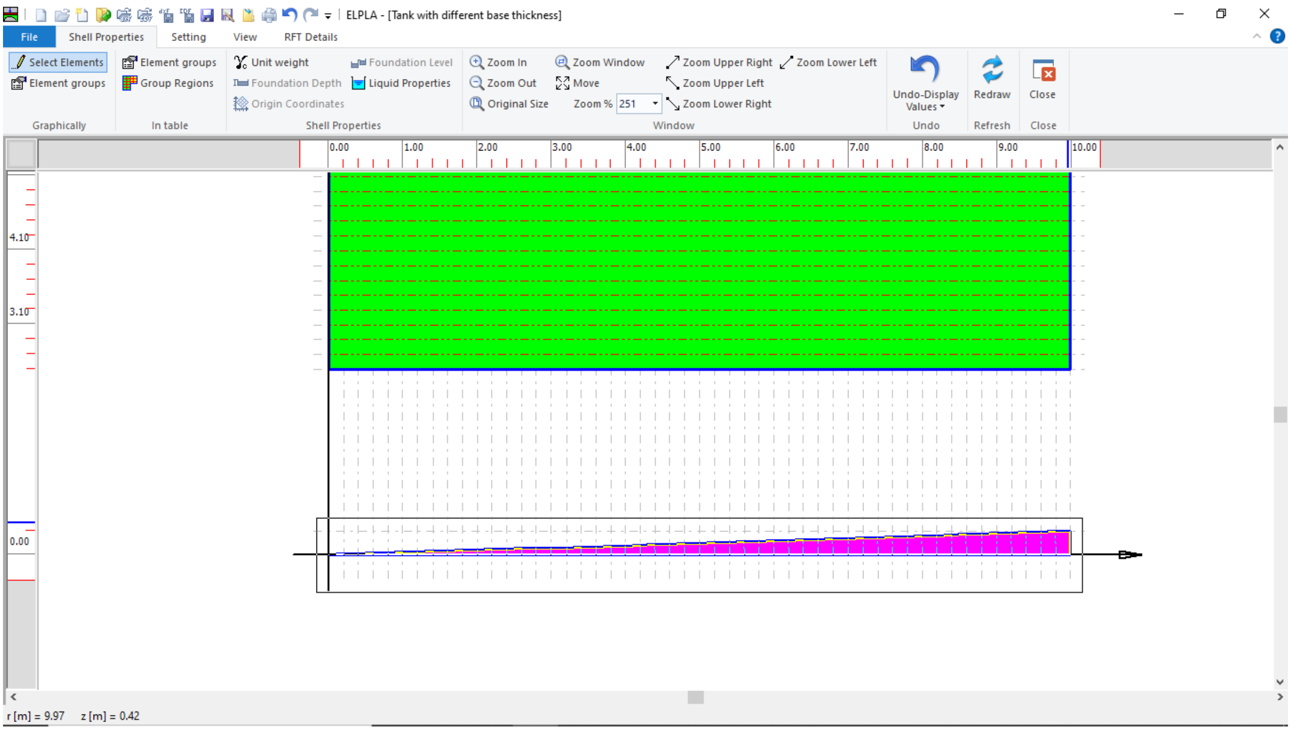


Figure 9.14 Selecting the nodes that include the tank base

Figure 9.15 "Group Regions" dialog box

To enter the unit weight of the tank, choose "Unit weight" command from "Shell Properties" menu in Figure 9.12. The following dialog box in Figure 9.16 with a default unit weight of 25 [kN/m^3] appears, type 24 in the "Unit Weight" edit box, then click "OK" button.

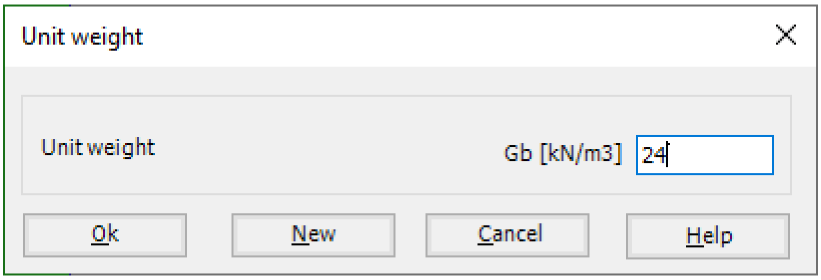


Figure 9.16 "Unit weight" dialog box

To define the filled material properties of the shell, choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 9.12. The following form in Figure 9.17 appears.

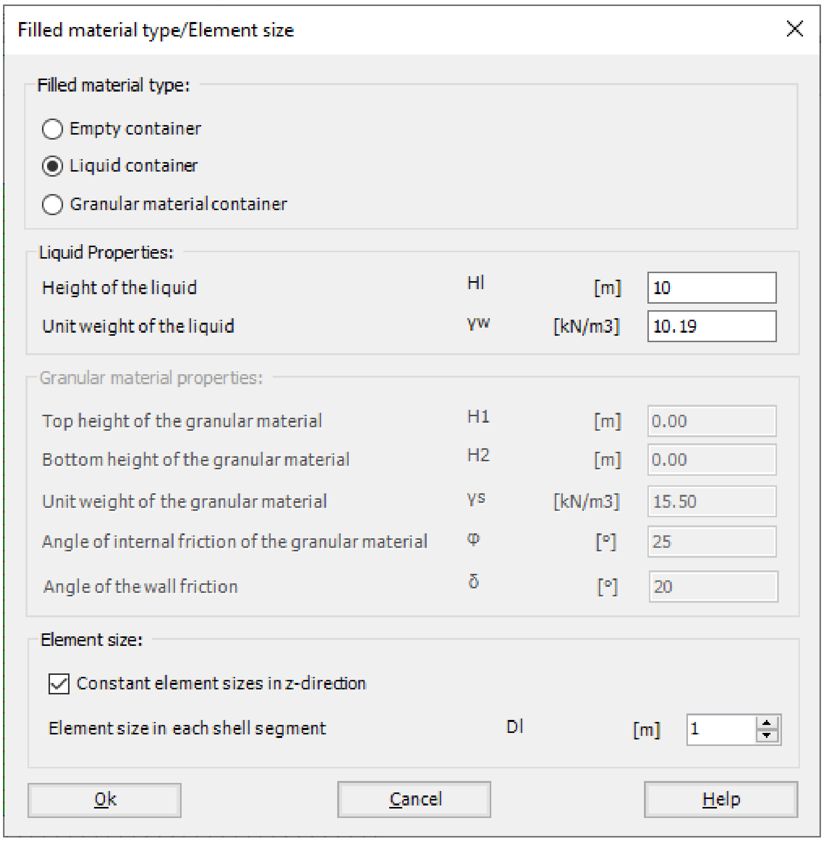


Figure 9.17 "Liquid Properties/Element size" dialog box

To define the filled material properties of the tank:

- Select the "Liquid container" option
- Type 10 in the "Height of the liquid" edit box
- Type 10.19 in the "Unit weight of the liquid" edit box

To define the element size of the ring wall:

- Check the "Constant element sizes in z-direction" check box
- Type 1 in the "Element size in each shell segment" edit box. The element size is chosen to be 1 [m] larger than the segment size in order to ignore further subdivision of the segments into smaller elements. In some cases, it is necessary to divide the segment into smaller elements in order to make the analysis more precise. Nevertheless, the final results of the internal forces appear only at nodes of segments
- Click "OK" button

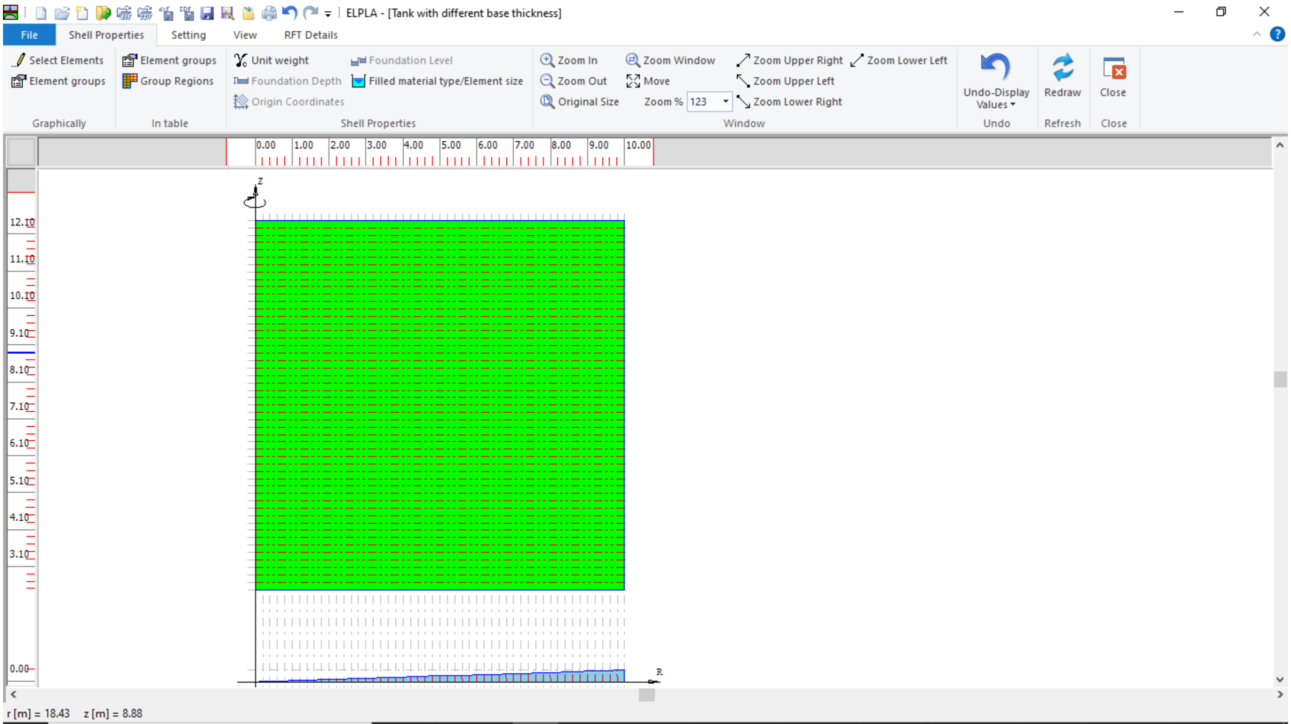


Figure 9.18 "Shell Properties" window after defining the shell data

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

- Choose "Save" command from "File" menu in Figure 9.18 to save the shell properties
- Choose "Close" command from "File" menu in Figure 9.18 to close the "Shell properties" window and return to *ELPLA* main window

4.5 Soil properties

To define the soil properties, choose "Soil Properties" command from “Data” Tab. The following "Soil Properties" form in Figure 9.19 appears, the soil properties are defined by Modulus of Elasticity "*E*", and is supposed to have the following parameters:

Modulus of Elasticity of the soil	*E*	= 20000	[kN/m^2]
Unit weight of the soil	*GAM*	= 18	[kN/m^3]
Angle of internal friction	*FHI*	= 30	[°]
Cohesion of the soil	c	= 0	[kN/m^2]
Poisson's ratio of the soil medium	ν_s	= 0.2	[-]

Other data in the example are not required, the user can use the default values.

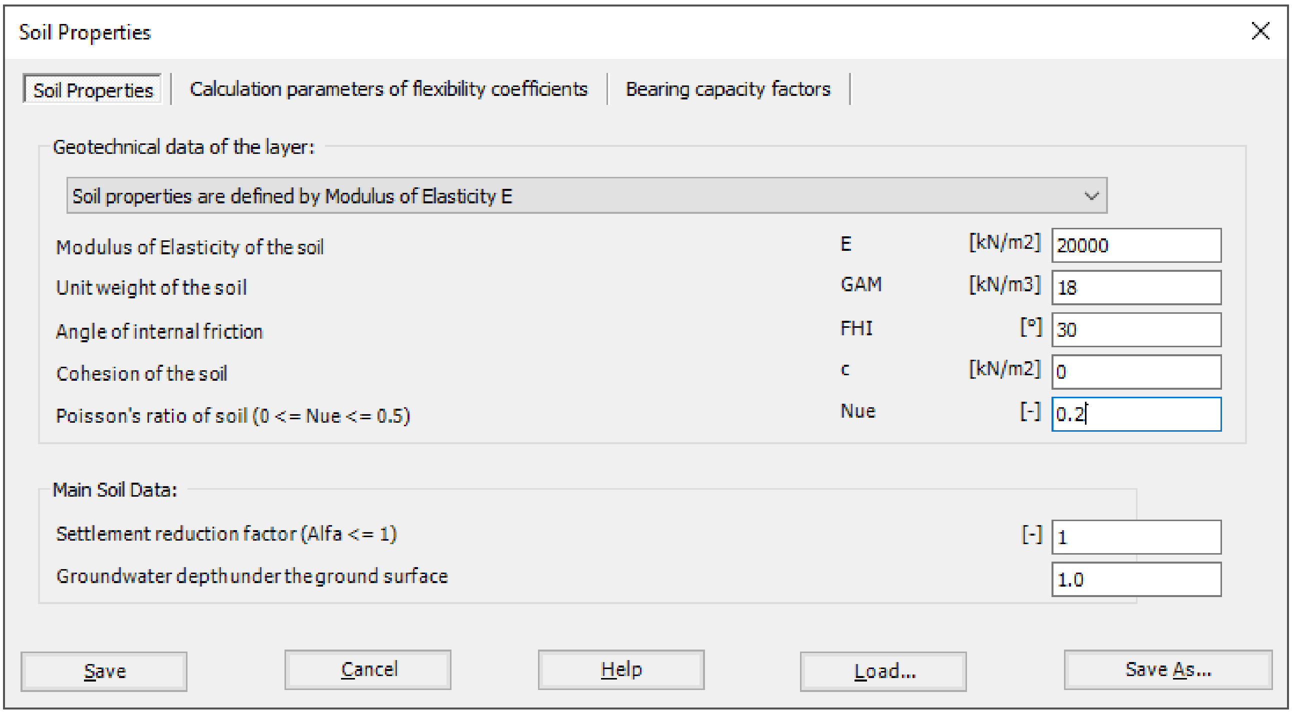


Figure 9.19 "Soil Properties" form

After defining the soil properties, click "Save" button.

4.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 9.20 appears. In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 9.20. In this example, there is not applied load, as the vertical load has been already defined by the unit weight of the tank, while the hydrostatic pressure on the tank wall is defined by the unit weight of water.

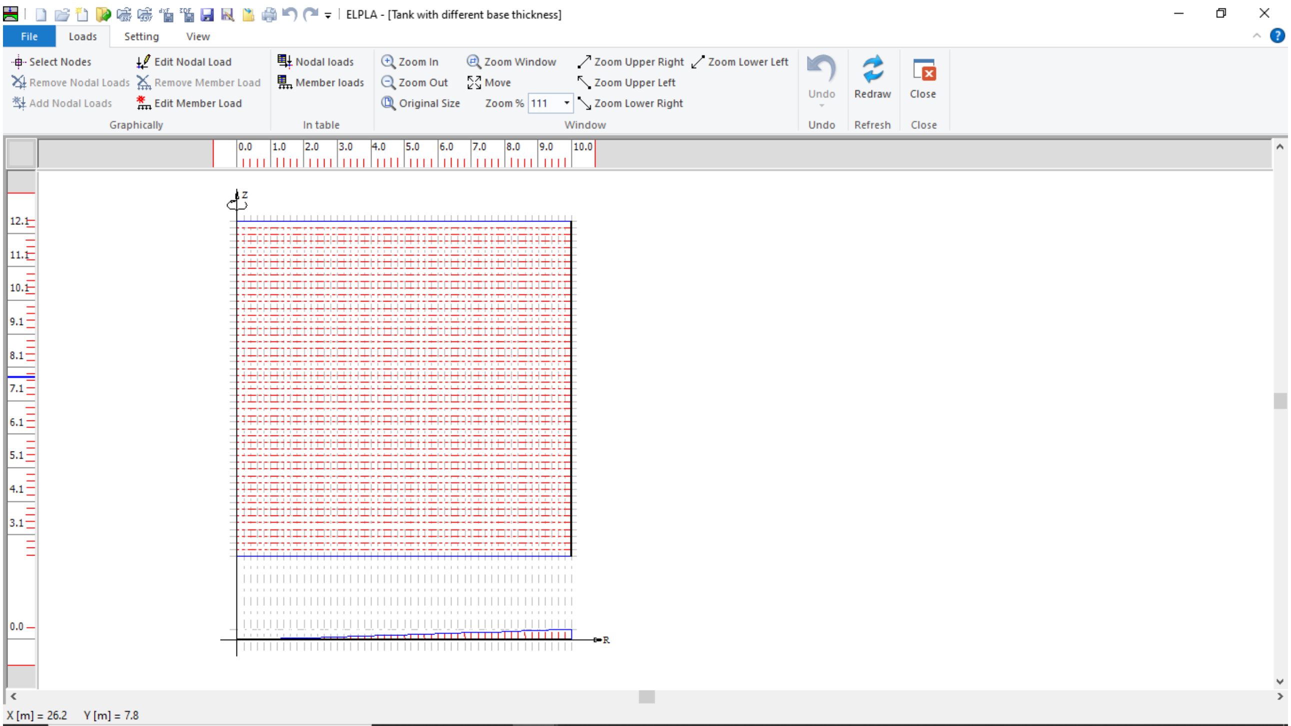


Figure 9.20 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 9.20 to save the load data
- Choose "Close" command from "File" menu in Figure 9.20 to close the "Loads" window and return to *ELPLA* main window

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

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 9.21.

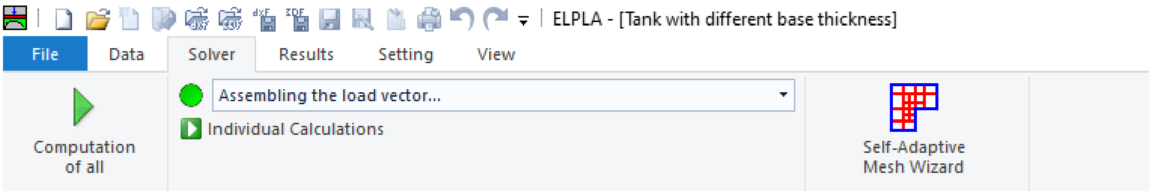


Figure 9.21 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

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

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

Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab Window. The following "Iteration parameters" option box in Figure 9.22 appears
  For this example, choose an accuracy of 0.0001 [m] to end the iteration process
- Click "OK" button

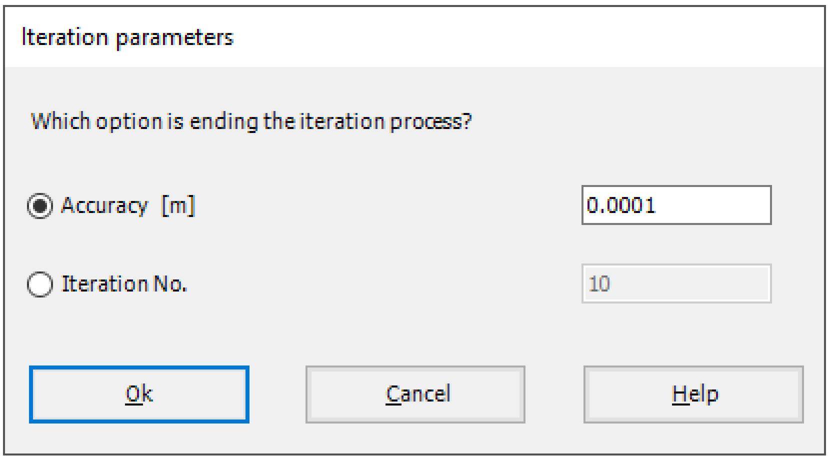


Figure 9.22 "Iteration parameters" option box

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

Analysis progress

Analysis progress menu in Figure 9.23 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

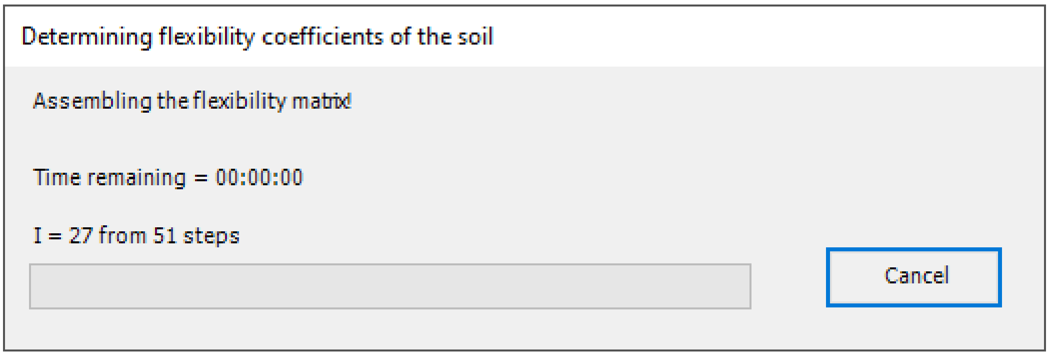


Figure 9.23 Analysis progress menu

"Check of convergence" message Figure 9.24 appears showing that no convergence is reached at the last step, click "Ok" button.

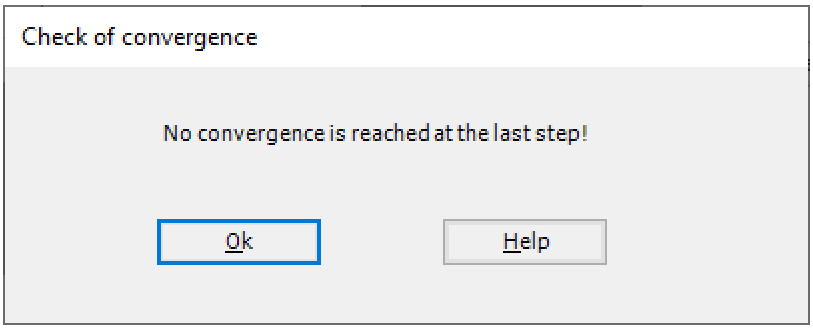


Figure 9.24 "Check of convergence" message

Click "Stop" button in Figure 9.25, to stop the iteration process as no convergence has reached.

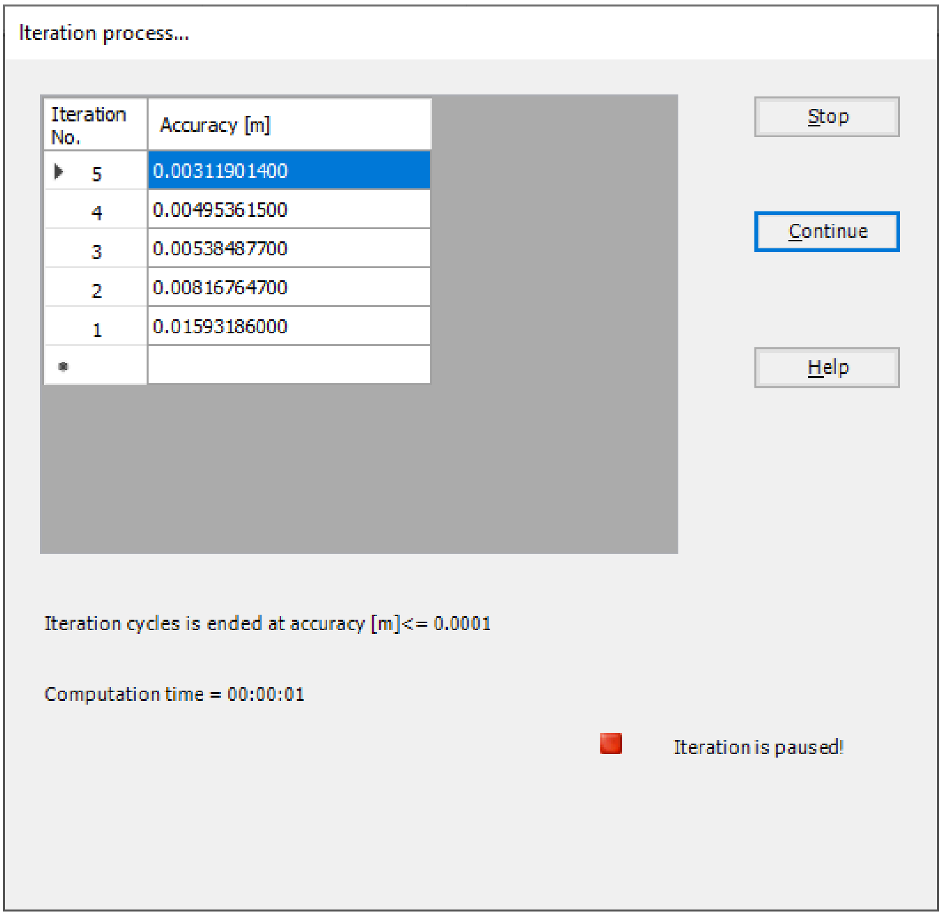


Figure 9.25 "Iteration process" list box

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 9.26. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

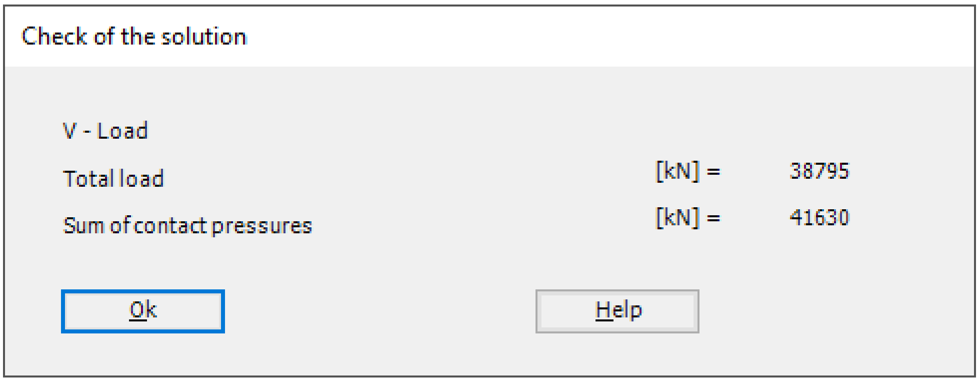


Figure 9.26 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab. To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 9.27).

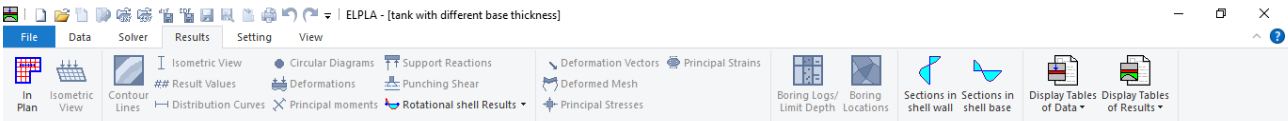


Figure 9.27 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Sections in shell base
- Sections in shell wall
- Display tables of data
- Display tables of results

To view the radial forces in the shell wall

- Choose "Sections in shell wall" command from "Section" menu. The following option box in Figure 9.30 appears
- In the "Sections in shell wall" option box, select "Radial forces *Nr*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 9.29.

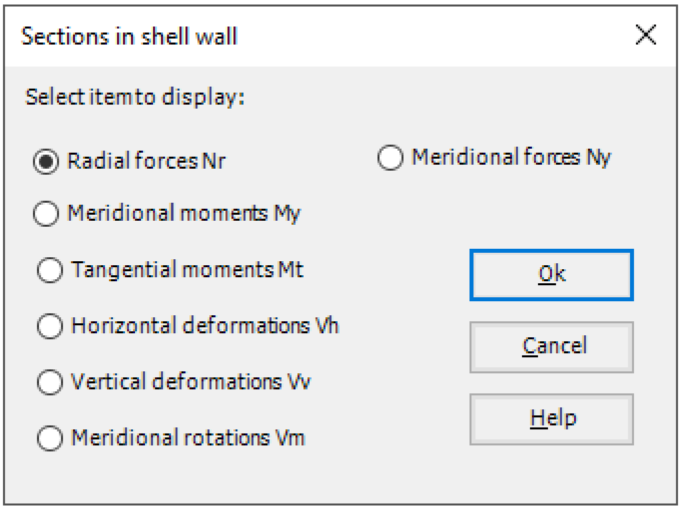


Figure 9.28 "Sections in shell wall" option box

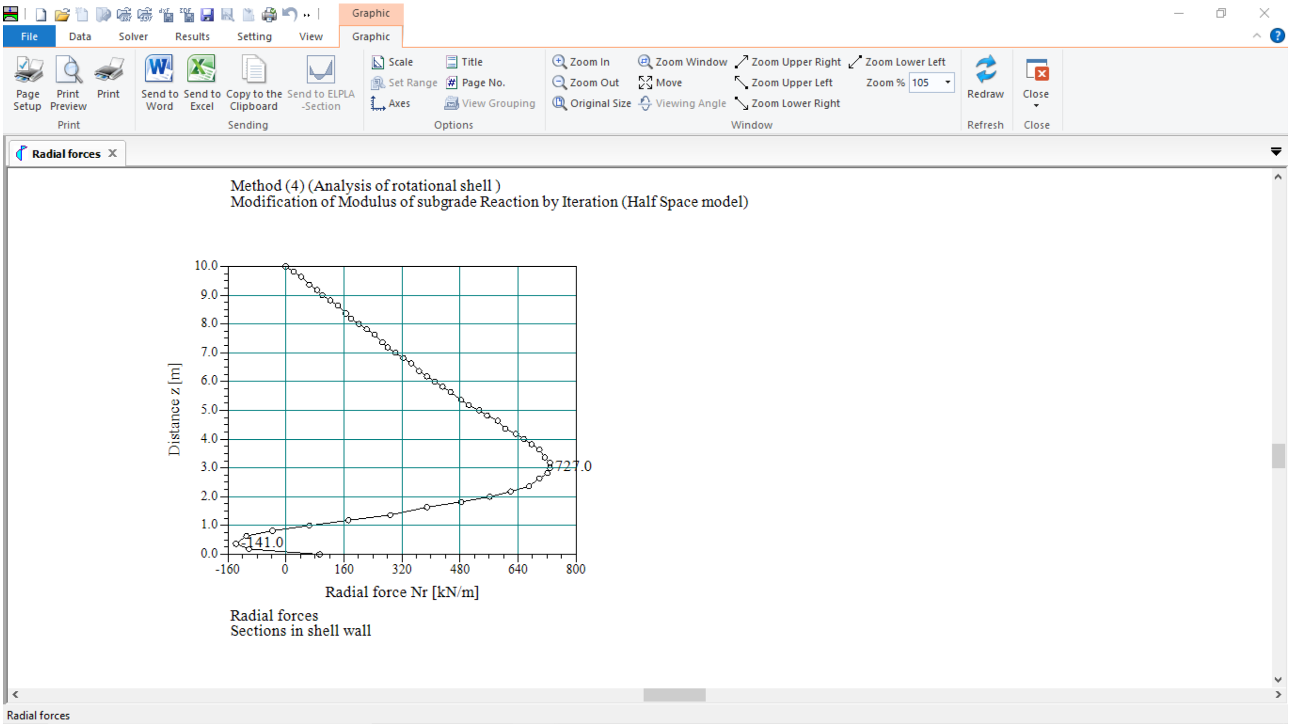


Figure 9.29 Radial forces in shell wall

To view the meridional moments in the shell base

- Choose "Sections in shell base" command from "Section" menu. The following option box in Figure 9.30 appears
- In the "Sections in shell base" option box, select "Meridional moments *My*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 9.31.

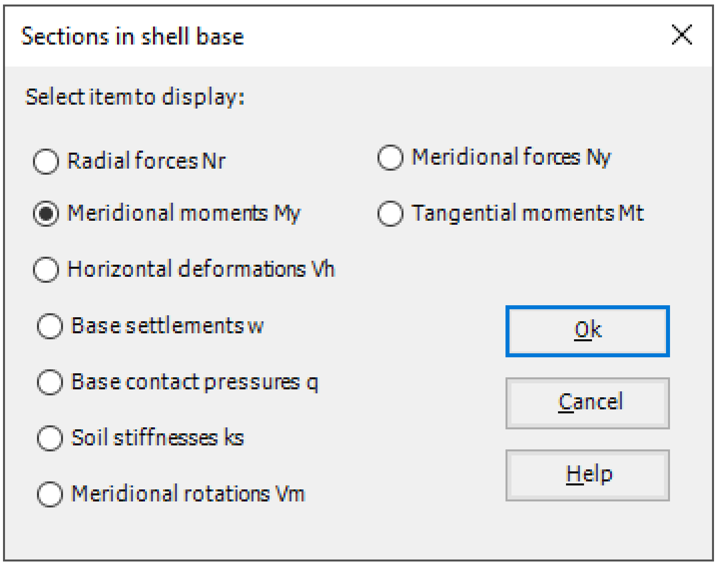


Figure 9.30 "Sections in shell base" option box

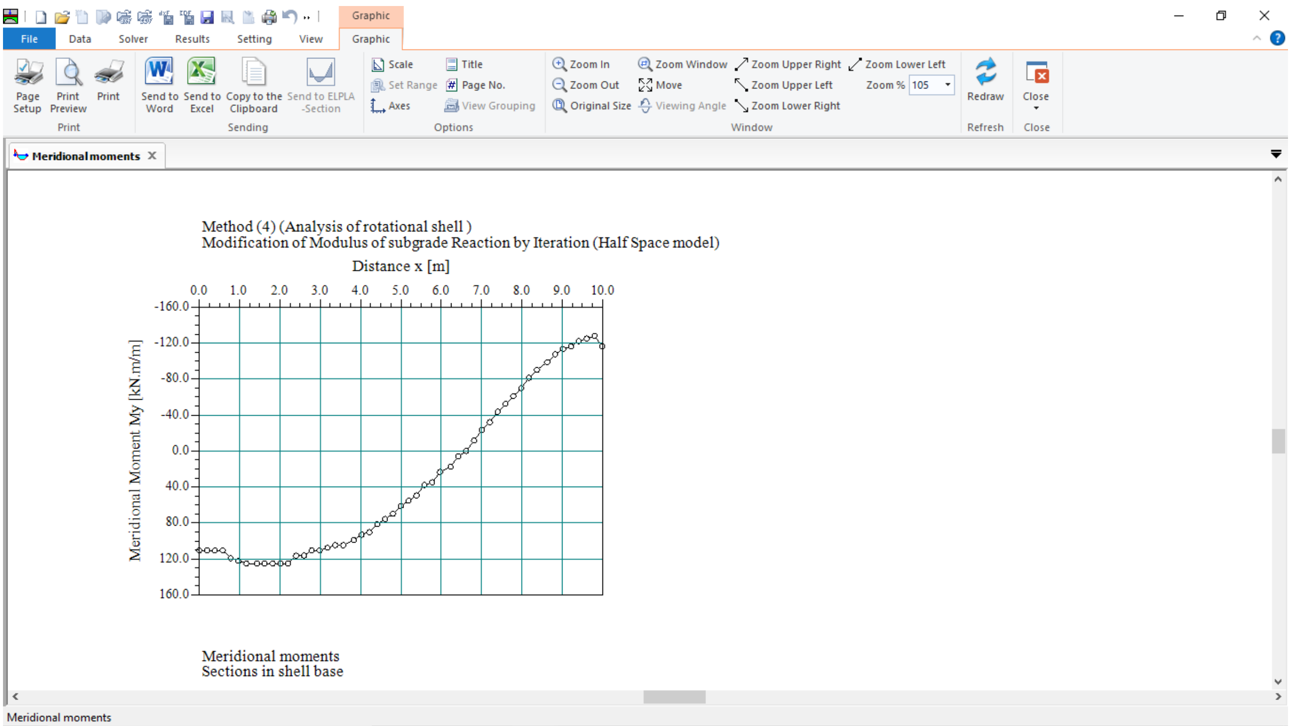


Figure 9.31 Meridional moments in shell base

To view element groups of the tank

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 9.32 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

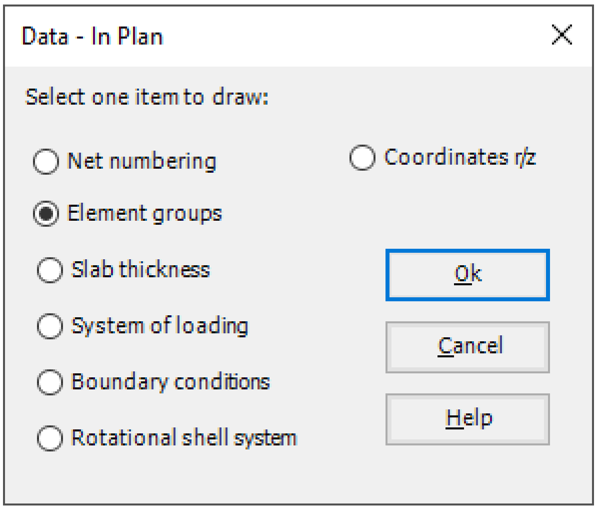


Figure 9.32 "Data – In Plan" option box

To view the meridional moments on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 9.33 appears
- In this check group box, check "Meridional moments *My*" check box
- The user can choose any other data to be displayed
- Click "OK" button

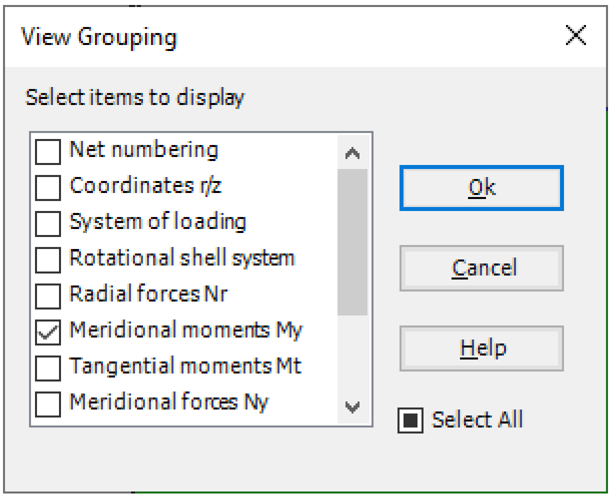


Figure 9.33 "View Grouping" check group box

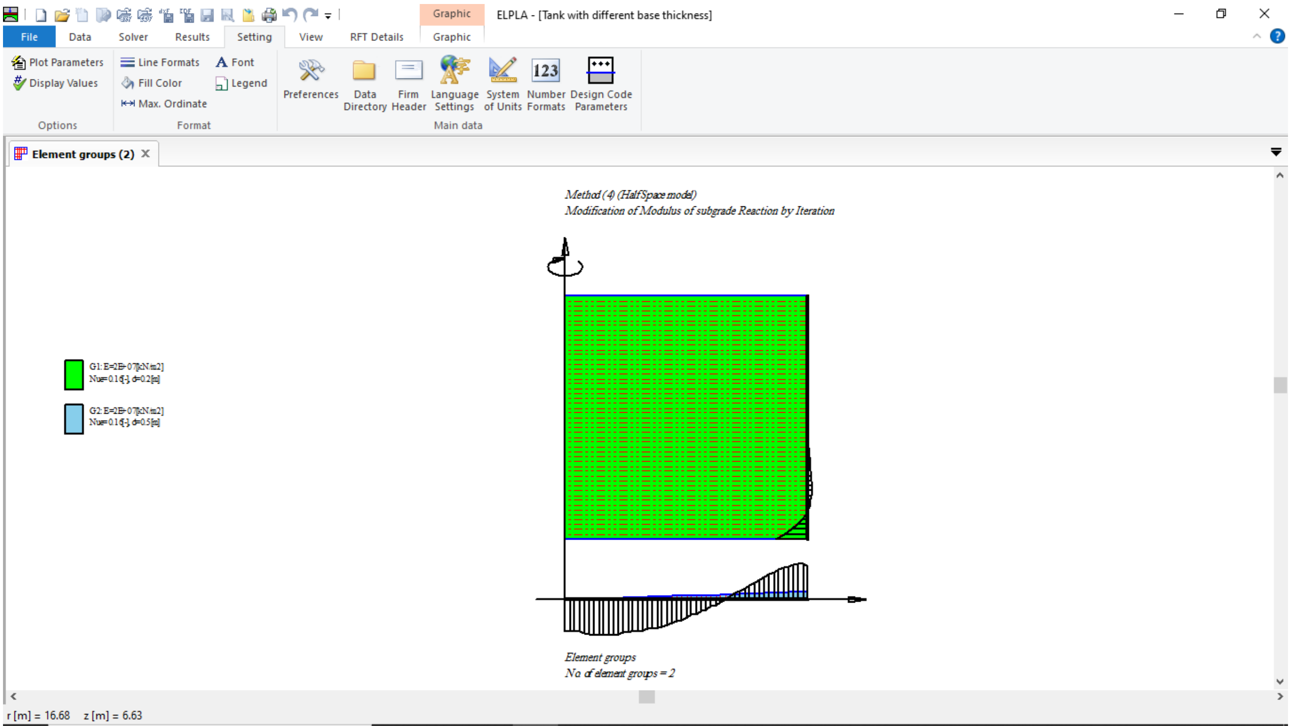


Figure 9.34 Element groups of the tank with the meridional moments

Example 10

Analysis of a cylindrical water container with a conical base

Contents Page

1 Description of the problem

An example of a cylindrical water container with a conical base is selected to illustrate some features of *ELPLA* for analyzing circular cylindrical shell elements.

2 Container geometry and properties

A cylindrical water container with a conical base of a radius of a = 3.0 [m] and a height of H = 12.0 [m] is considered as shown in Figure 10.1. Thickness of the container wall is 0.3 [m], while that for the conical base is 0.2 [m]. Figure 10.1 shows the container with dimensions and supports, while the container material and unit weight of the water are listed in Table 10.1.

Table 10.1 Container material and water unit weight

Modulus of Elasticity of the container material	E_c	= 10000	[kN/m^2]
Poisson's ratio of the container material	ν_c	= 0.17	[-]
Unit weight of the water	γ_w	= 10	[kN/m^3]

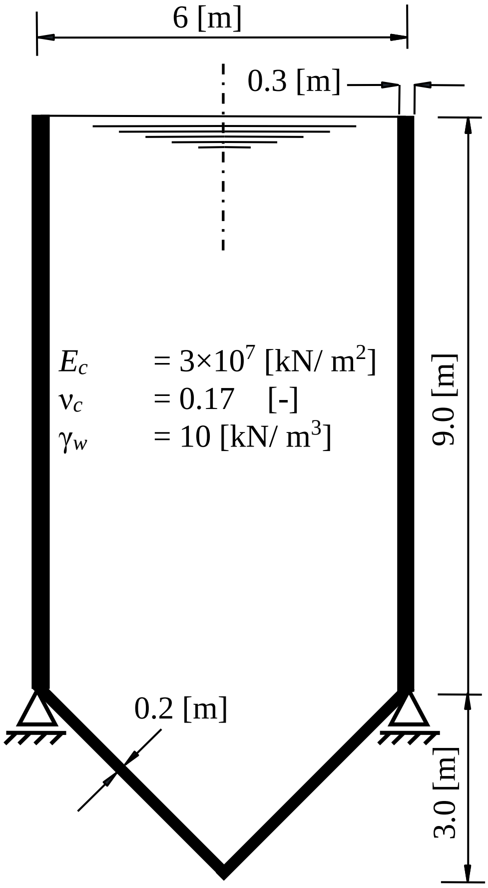


Figure 10.1 Cylindrical water container with dimensions and supports

3 Numerical Analysis

In the analysis, the height of the tank is divided into two main segments, the first one is divided into 30 subsegments (30×0.3 [m]), while the second is divided into 20 subsegments (20×0.15 [m]) as shown in Figure 10.2.

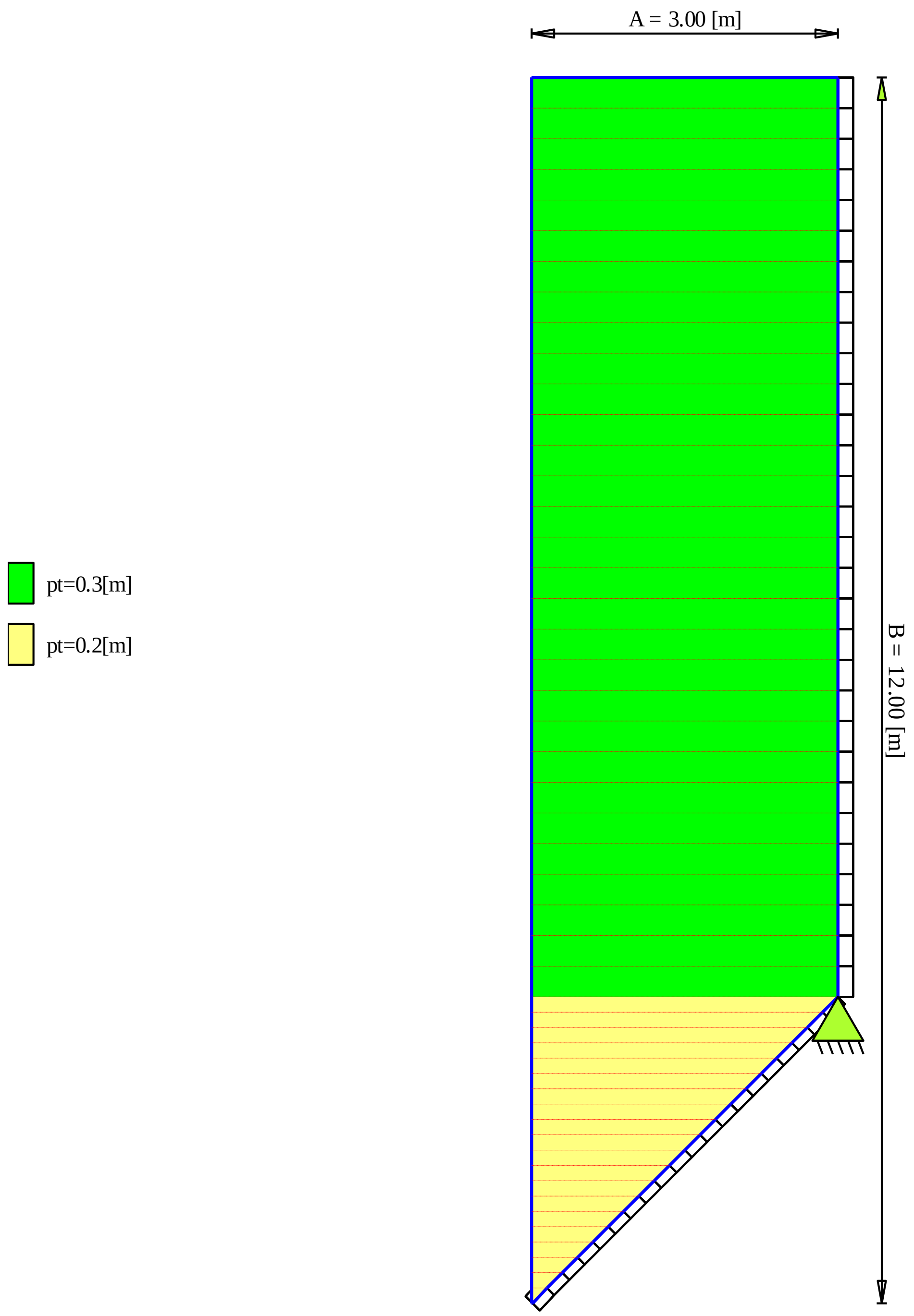


Figure 10.2 Finite element mesh of the container with boundary condition

4 Creating the project

In this section, the user will learn how to create a project for analyzing a cylindrical water container with a conical base. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 10.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of forms. The first form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 10.3).

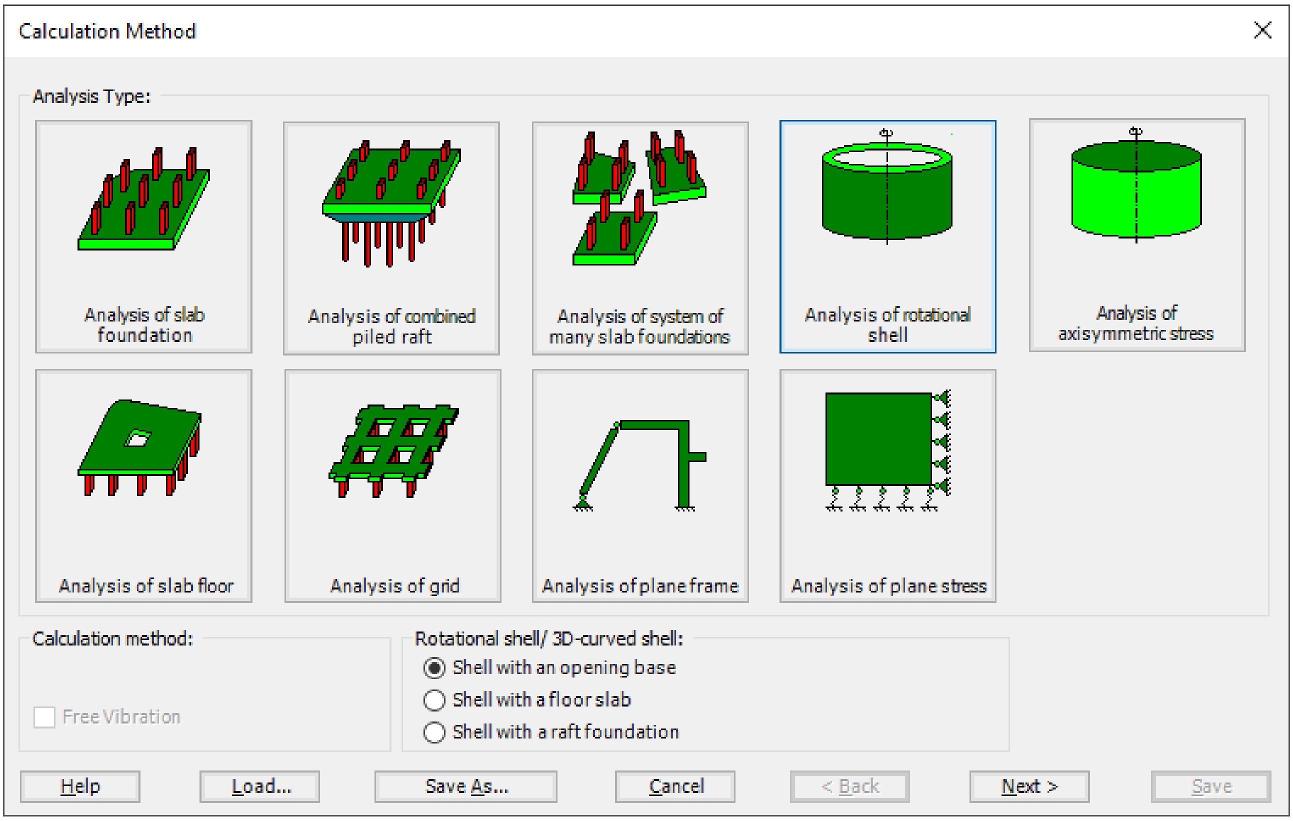


Figure 10.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 10.3, define the analysis type of the problem. As the analysis type is a cylindrical water container with a conical base problem, select "Analysis of rotational Shell" button, and check "Shell with an opening base" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Options" Form appears, Figure 10.4.

The last Form in the wizard is the "Options" Form, Figure 10.4. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Select "Supports/ Boundary Conditions", then click the "Save" button.

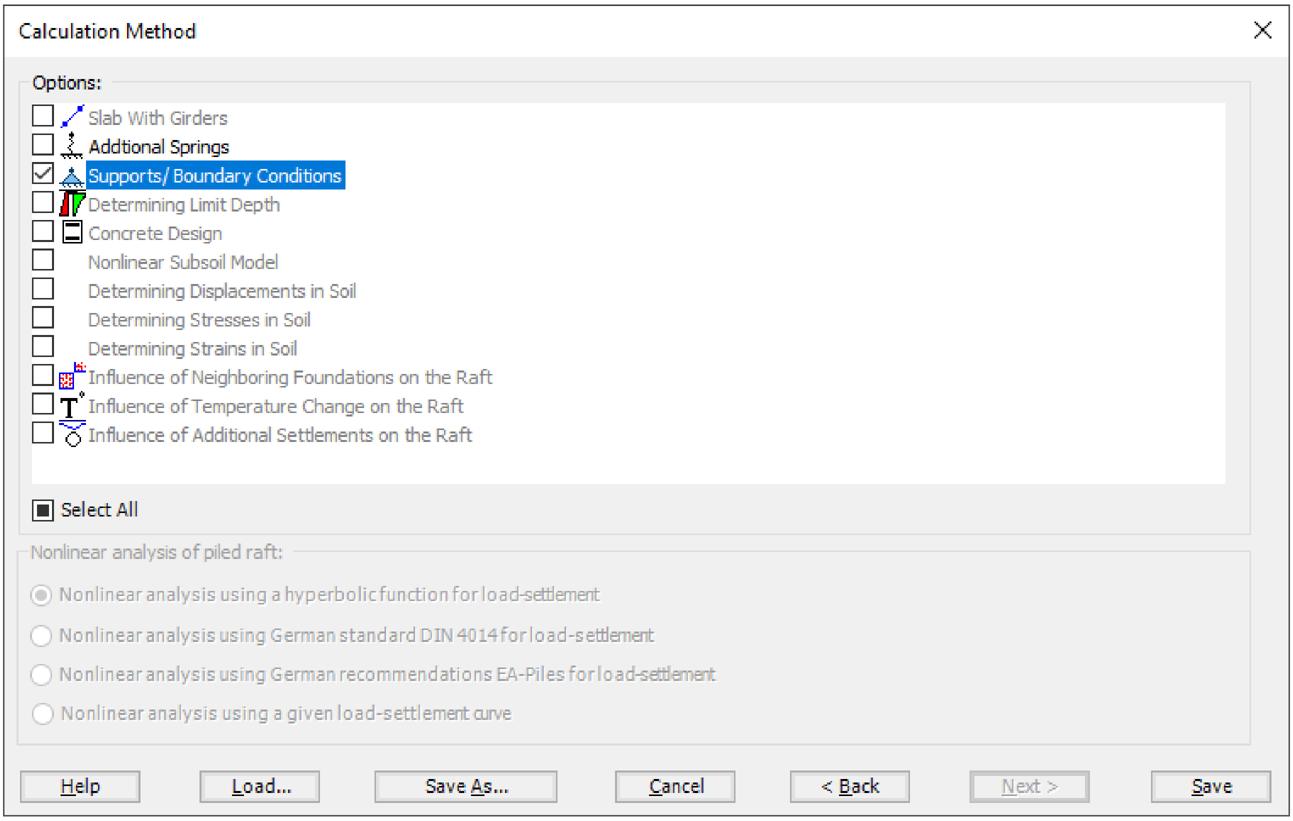


Figure 10.4 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 10.5. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Water container". *ELPLA* will use automatically this file name in all reading and writing processes.

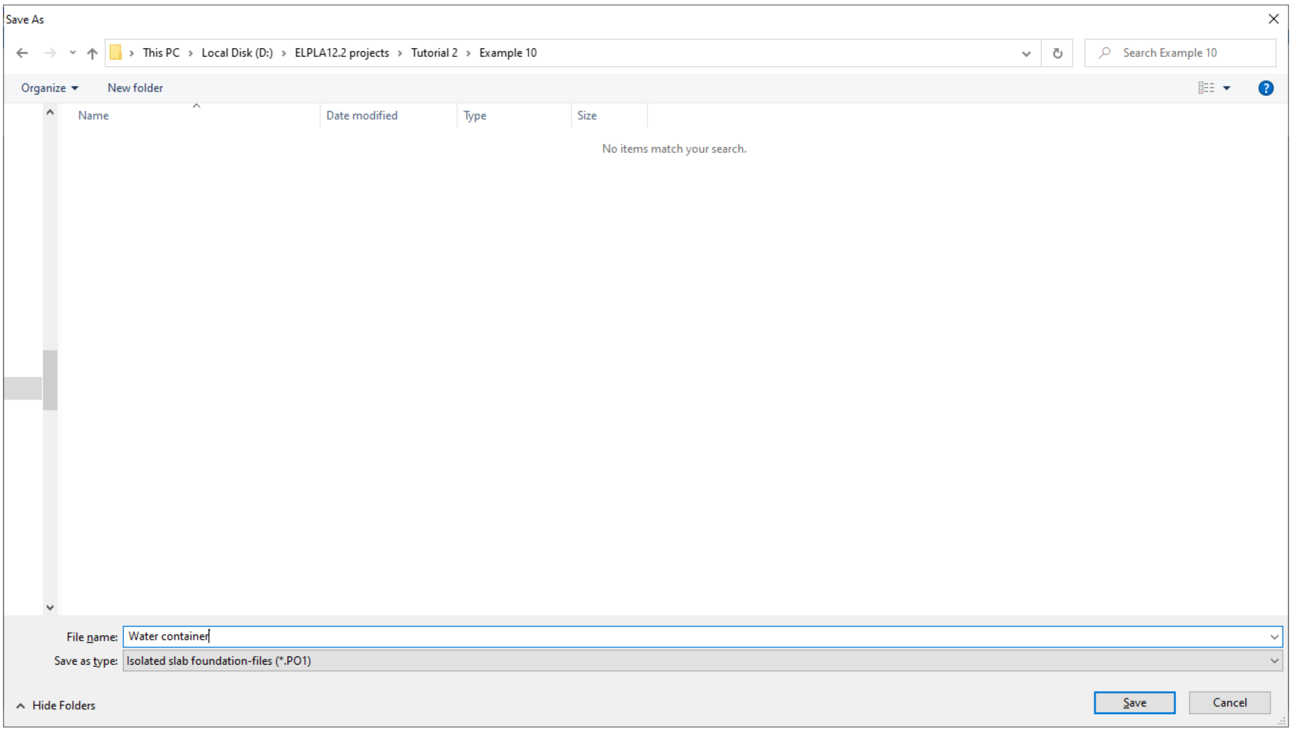


Figure 10.5 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Water container] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 10.6 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a cylindrical water container"
- Type the date of the project in the "Date" edit box
- Type "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

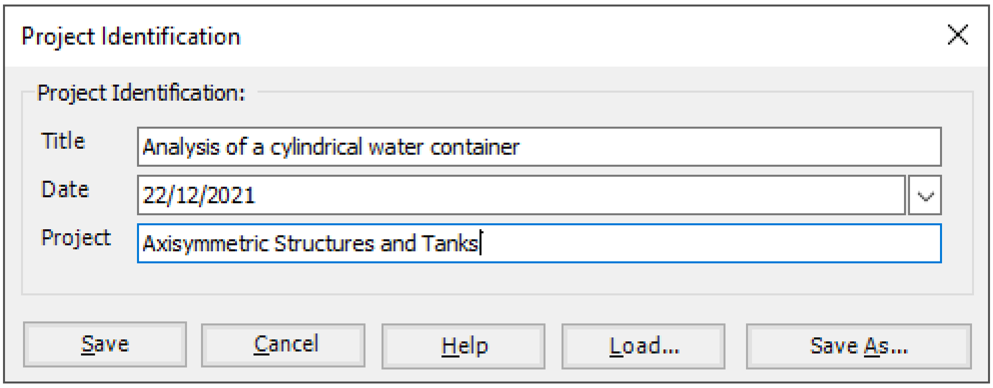


Figure 10.6 "Project Identification" dialog box

4.3 FE-Net data

For the given problem, a cylindrical water container with a conical base of a radius of a = 3.0 [m] and a height of H = 12.0 [m] is considered, the height of the tank is divided into two main segments, the first one is divided into 30 subsegments (30×0.3 [m]), while the second is divided into 20 subsegments (20×0.15 [m]). To define the FE-Net for this tank, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 10.7. This wizard will guide you through the steps required to generate a FE-Net, the first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

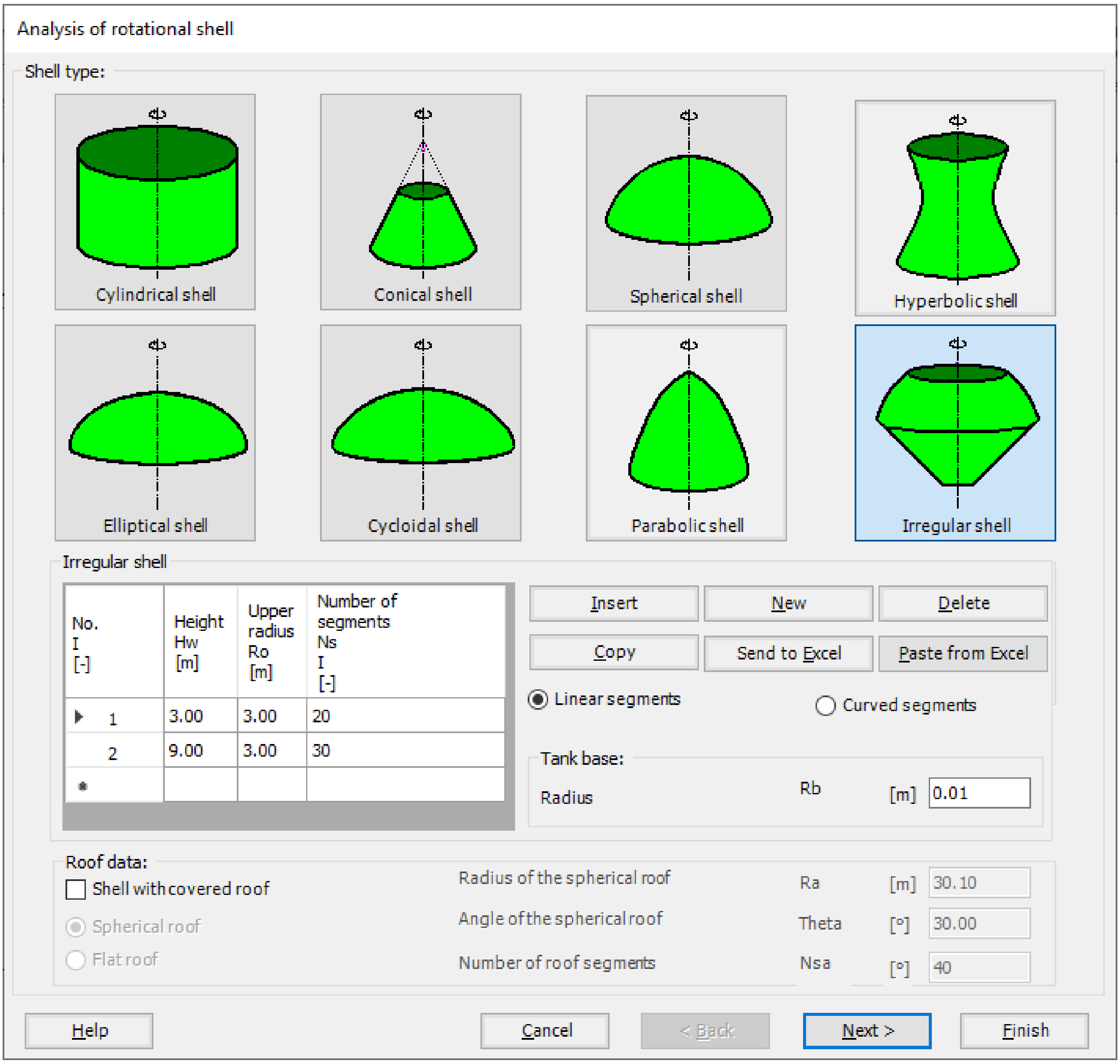


Figure 10.7 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Irregular shell" button
- Type 0.01 in the "Tank base radius *Rb*" edit box, which is approximately zero
  (Note that: from the principles of axis-symmetrical shell elements, radius cannot be exactly zero)

To identify the first main segment

- Type 3 in the "Height *Hw*" edit box
- Type 3 in the "Upper radius *Ro*" edit box
- Type 20 in the "Number of segments *Ns*" edit box

To identify the second main segment

- Type 9 in the "Height *Hw*" edit box
- Type 3 in the "Upper radius *Ro*" edit box
- Type 30 in the "Number of segments *Ns*" edit box

- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Irregular shell" Form containing the data of the segments appears in Figure 10.8, The user can edit the data of each segment individually or all of them by using "In Table" button, If it is necessary.

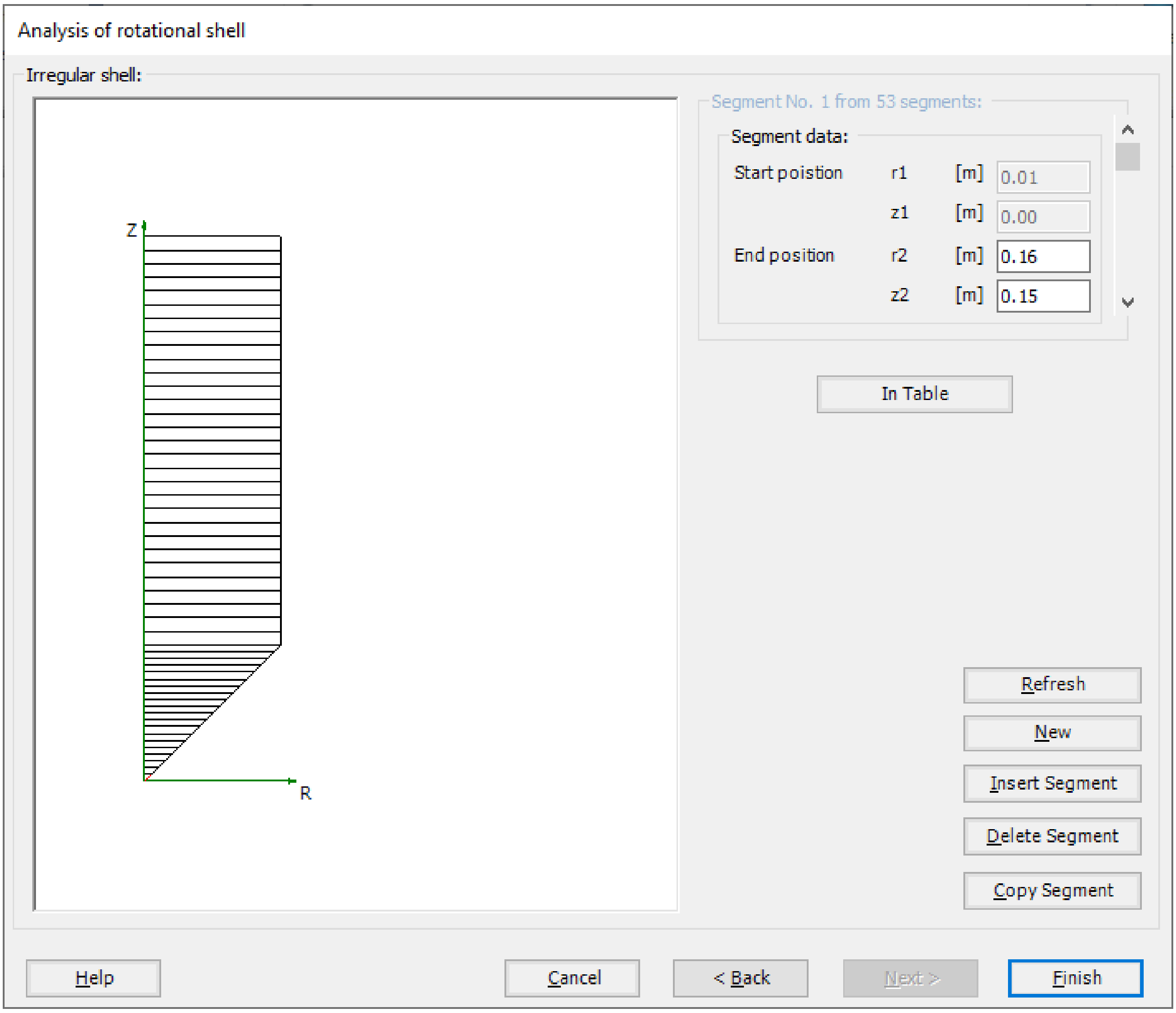


Figure 10.8 "Irregular shell" Form

Click "Finish" in "Analysis of rotational shell" wizard, the generated FE-Net appears Figure 10.9.

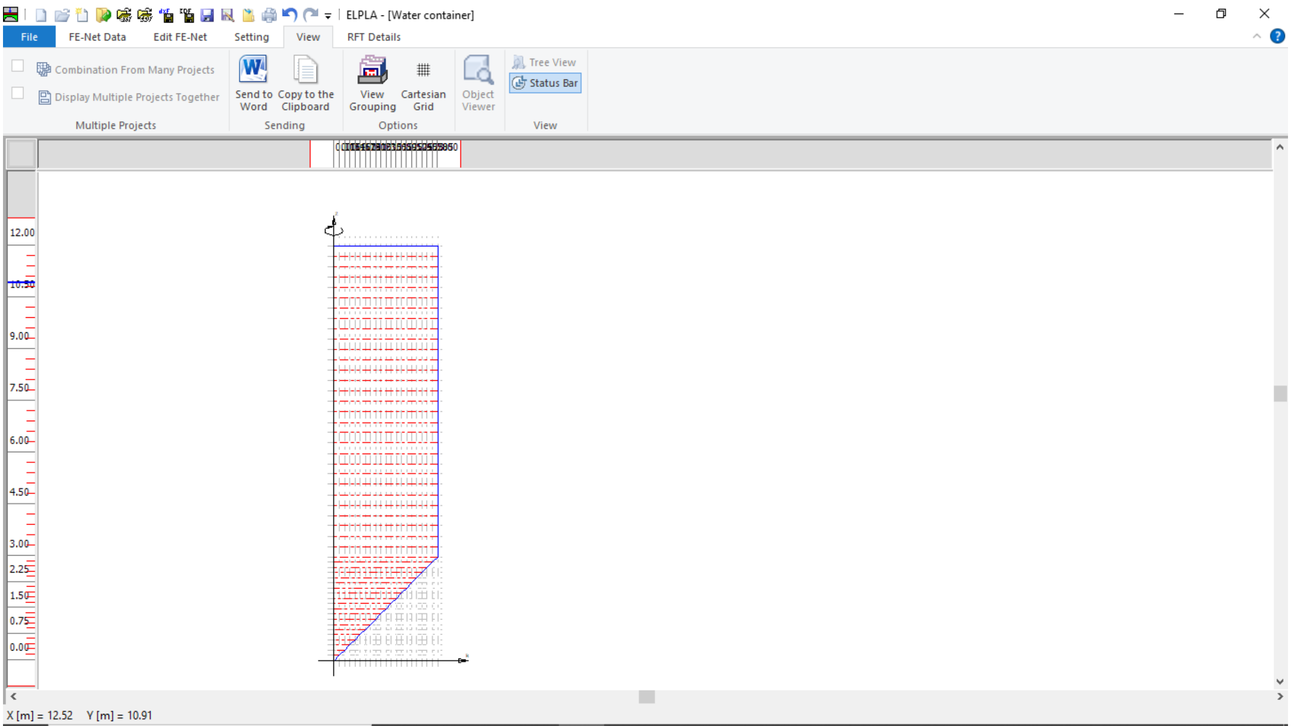


Figure 10.9 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 10.9 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 10.9 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Shell properties

To define the tank properties, choose "Shell Properties" command from "Data" Tab. The following window in Figure 10.10 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, group regions, unit weight of the tank, and the filled material properties.

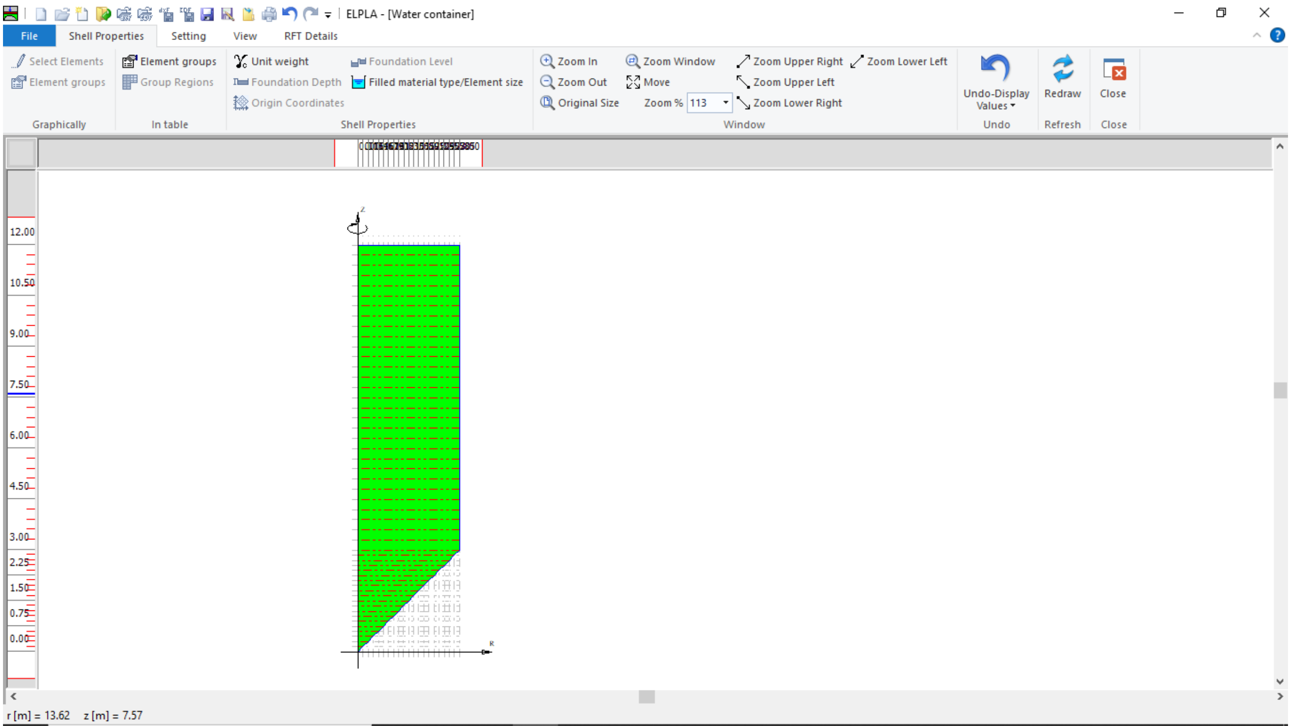


Figure 10.10 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 10.11 appears. In this list box, define E-Modulus, *Poisson's* ratio and slab thickness for both the wall and the base of the tank as they differ in thickness. Then click "OK" button.

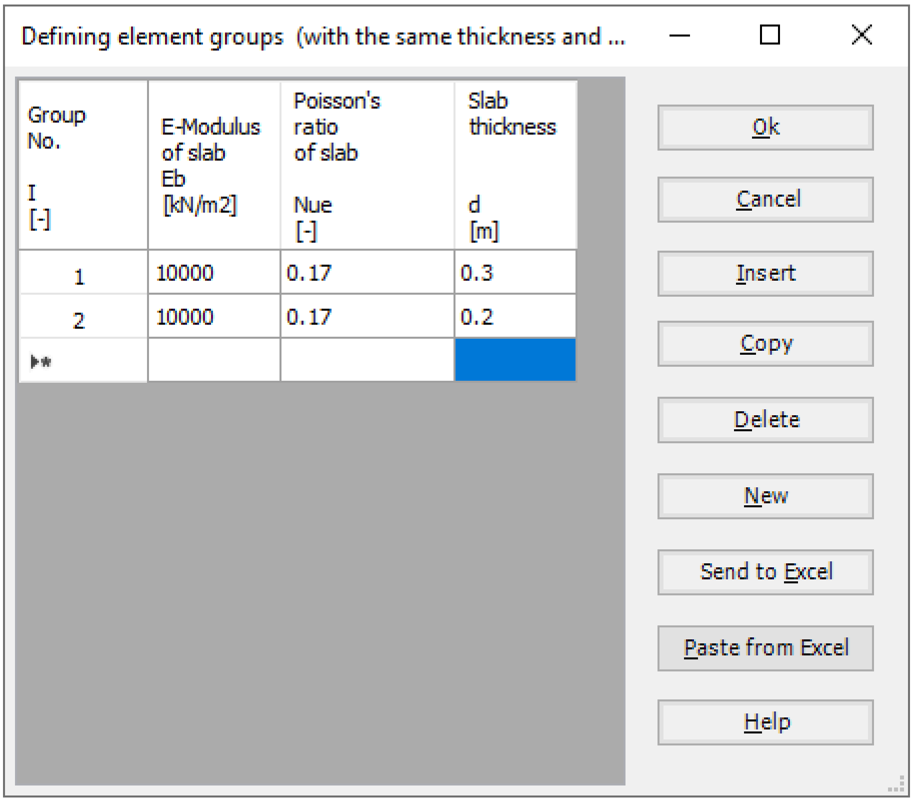


Figure 10.11 "Defining element groups" list box

Defining the slab thickness for materials on the net may be carried out either graphically or numerically (in a table). In the current example, the user will define the slab thickness on the net graphically.

To define the slab thickness for the tank base

- Choose "Select Elements" command from "Graphically" menu in the window of Figure 10.10
- When "Select Elements" command is chosen, the cursor will change from an arrow to a cross hair. A group of elements can be selected by holding the left mouse button down at the corner of the region. Then, drag the mouse until a rectangle encompasses the required group of elements. When the left mouse button is released, all elements in the rectangle are selected
- Select the elements that include the tank base as Figure 10.12
- Choose "Elements Groups" command from "Graphically" menu in the window of Figure 10.10, "Group Regions" dialog box Figure 10.13 appears
- Define the "Group No." of the base elements as type "2", while "Group No." of the wall elements will be as type "1", where type "1" is the default "Group No.", then click "OK" button

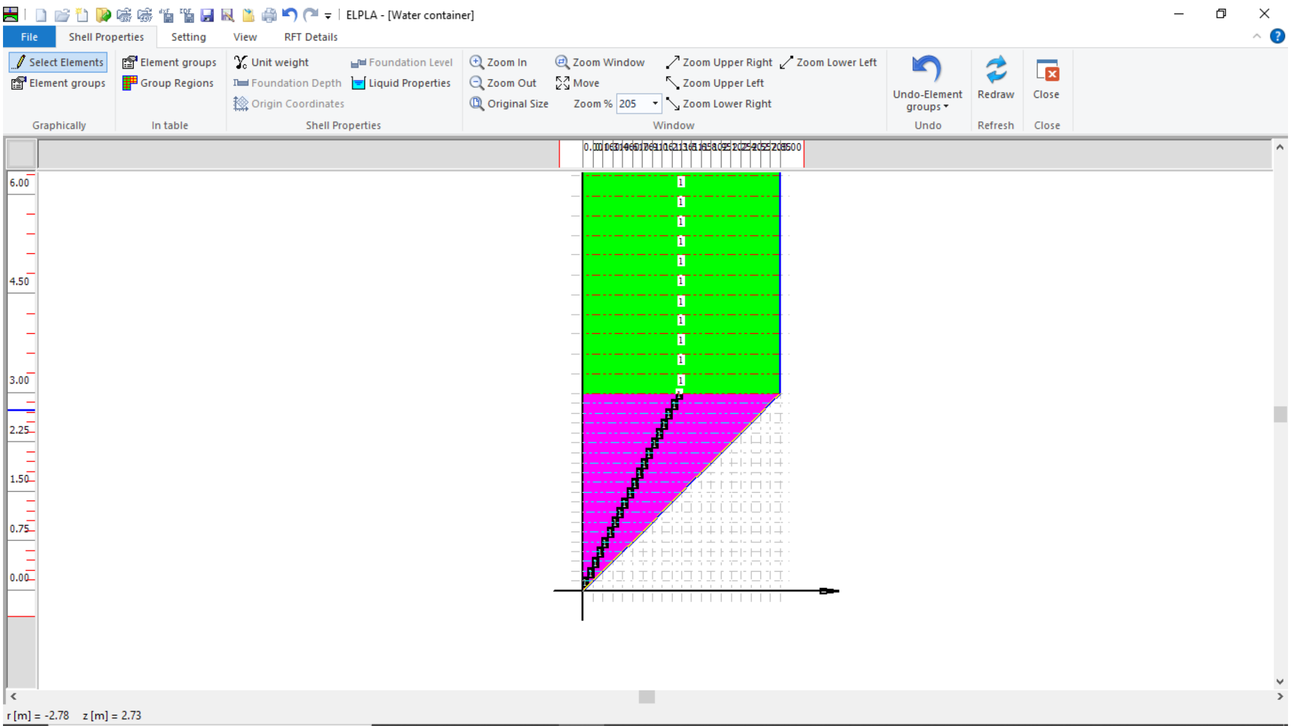


Figure 10.12 Selecting the elements that include the tank base

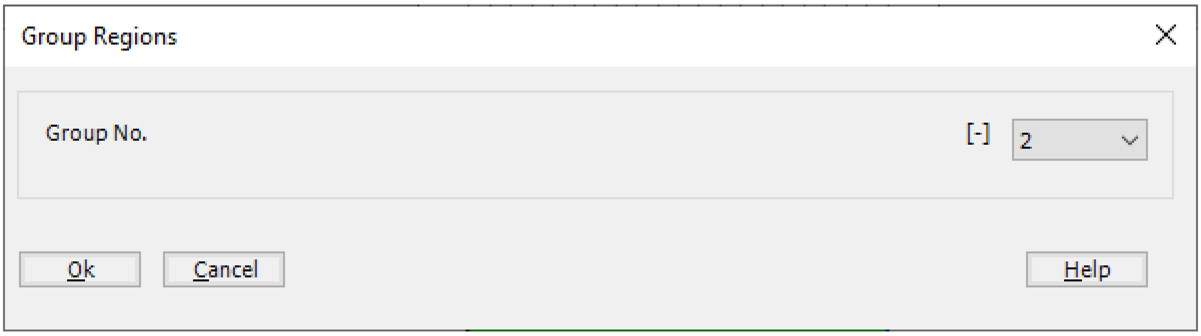


Figure 10.13 "Group Regions" dialog box

To enter the unit weight of the container, choose "Unit weight" command from "Shell Properties" menu in Figure 10.10. The following dialog box in Figure 10.14 with a default unit weight of 25 [kN/m^3] appears, type 0 in the "Unit weight" edit box to neglect the self-weight of the tank, then click "OK" button.

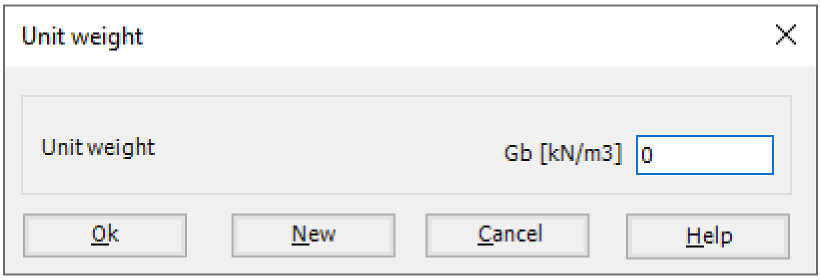


Figure 10.14 "Unit weight" dialog box

To define the filled material properties of the container, choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 10.10. The following form in Figure 10.15 appears.

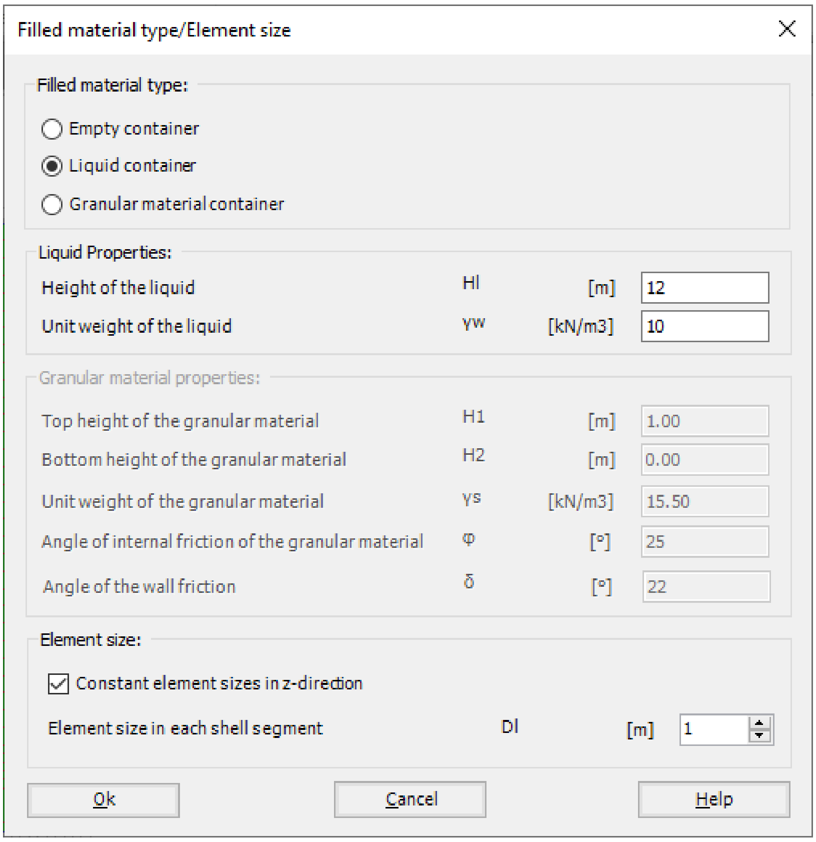


Figure 10.15 "Filled material type/Element size" Form

To define the filled material properties of the container:

- Select "Liquid container" option
- Type 12 in the "Height of the liquid" edit box
- Type 10 in the "Unit weight of the liquid" edit box

To define the element size of the container:

- Check the "Constant element sizes in z-direction" check box
- Type 1 in the "Element size in each shell segment" edit box. The element size is chosen to be 1 [m] larger than the segment size in order to ignore further subdivision of the segments into smaller elements. In some cases, it is necessary to divide the segment into smaller elements in order to make the analysis more precise. Nevertheless, the final results of the internal forces appear only at the nodes of segments
- Click "OK" button

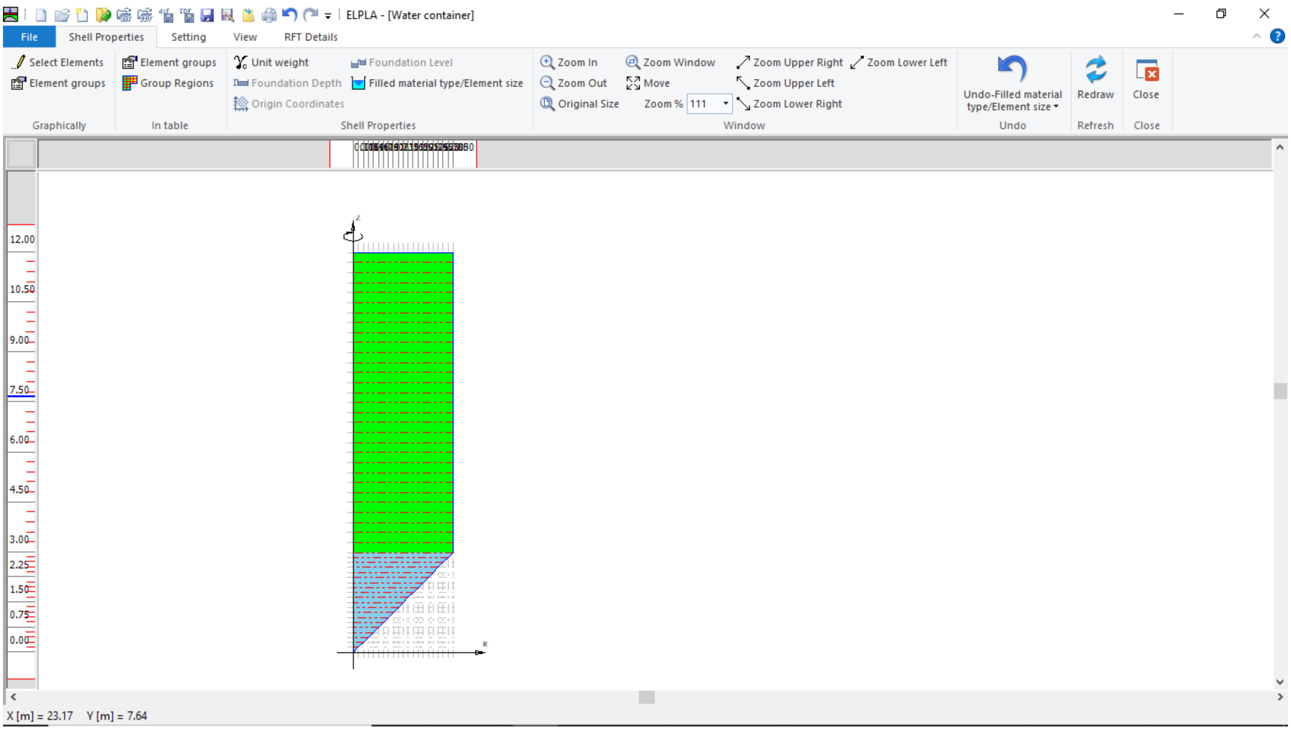


Figure 10.16 "Shell Properties" window after defining the container properties

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

- Choose "Save" command from "File" menu in Figure 10.16 to save the shell properties
- Choose "Close" command from "File" menu in Figure 10.16 to close the "Shell properties" window and return to *ELPLA* main window

4.5 Supports/ boundary conditions

To define the support, choose "Supports/ Boundary Conditions" command from “Data” Tab. The following window in Figure 10.17 appears.

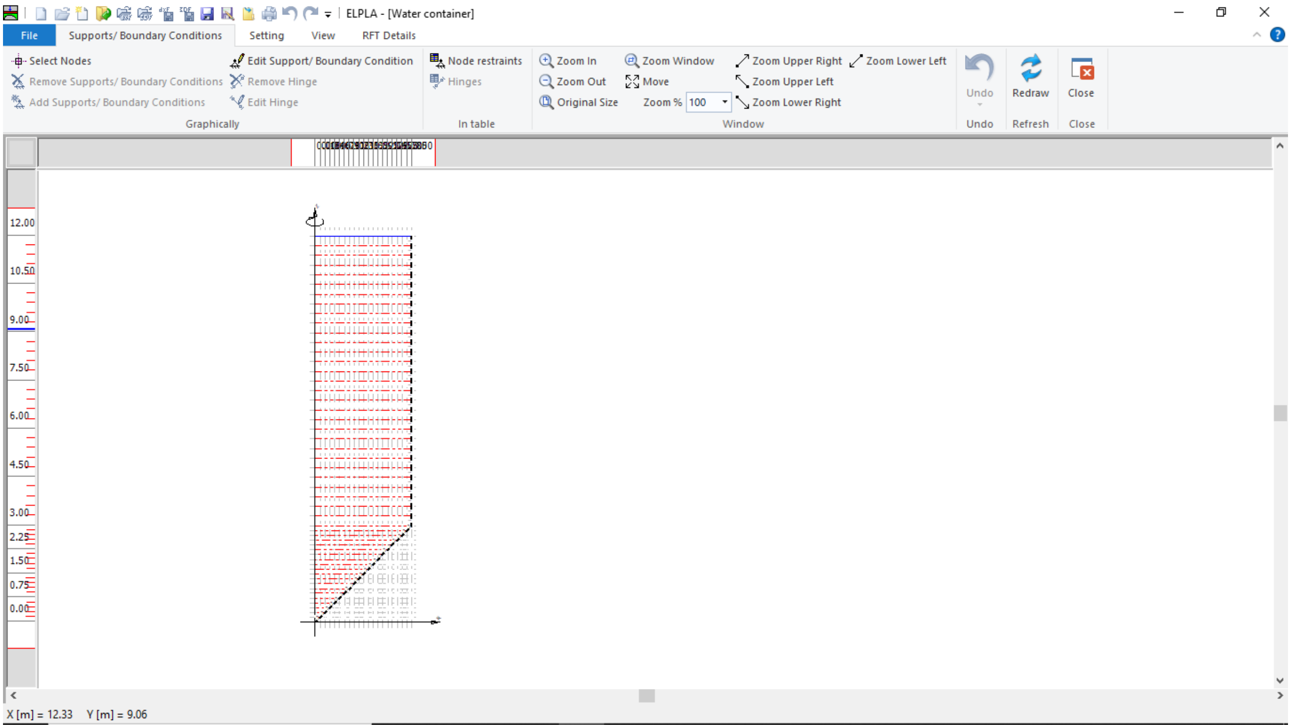


Figure 10.17 "Supports/ Boundary Conditions" Window

To define the support on the net:

- Choose "Select Nodes" command from "Graphically" menu in Figure 10.17. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the node that has the support as shown in Figure 10.18
- After selecting the node, choose "Add Supports/ Boundary Conditions" command from "Graphically" menu (Figure 10.17). The "Supports/ Boundary Conditions" dialog box in Figure 10.19 appears.

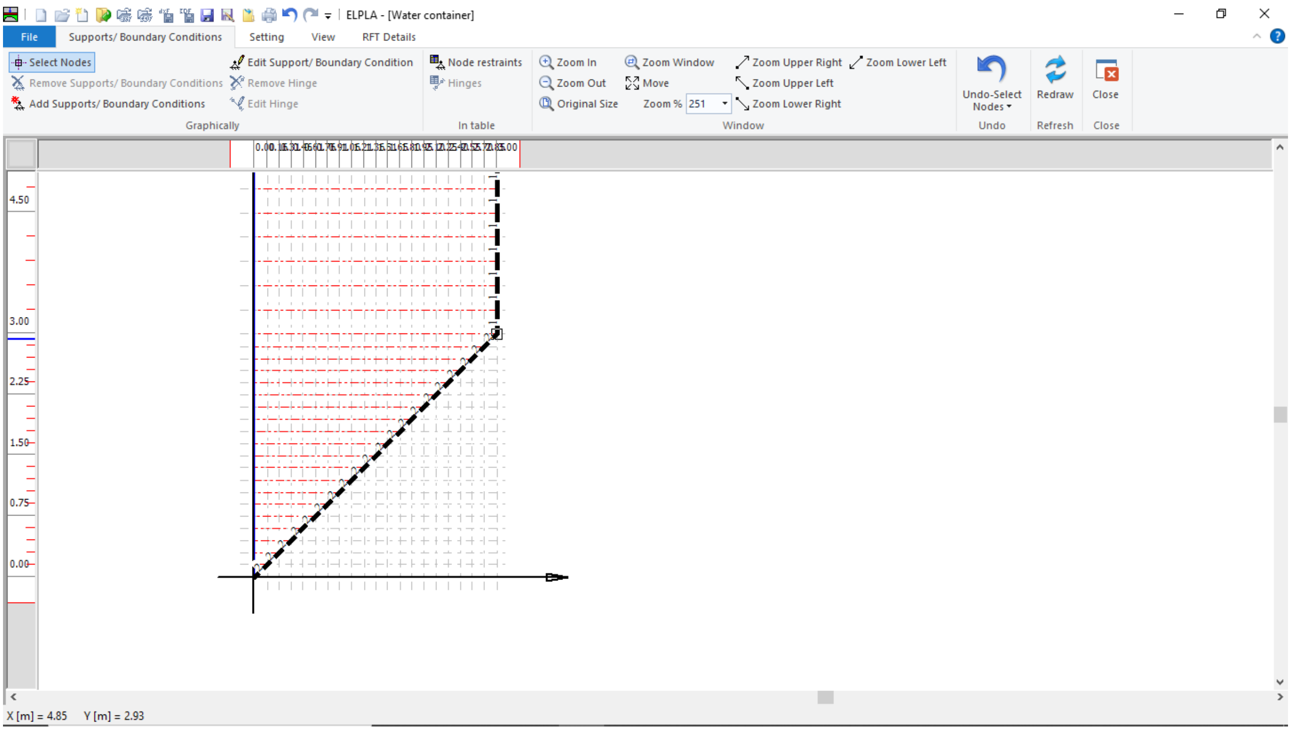


Figure 10.18 Selection of the node that has the support

In this dialog box

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

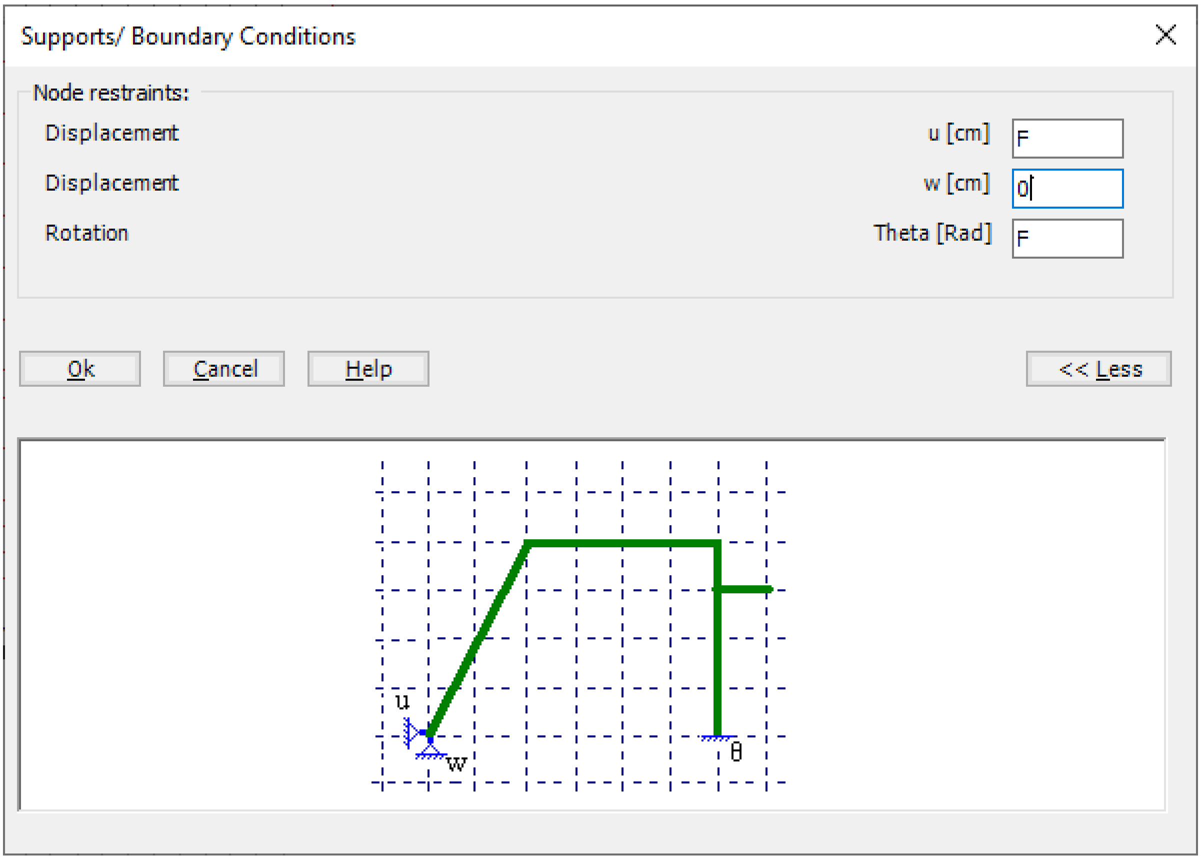


Figure 10.19 "Supports/ Boundary Conditions" dialog box

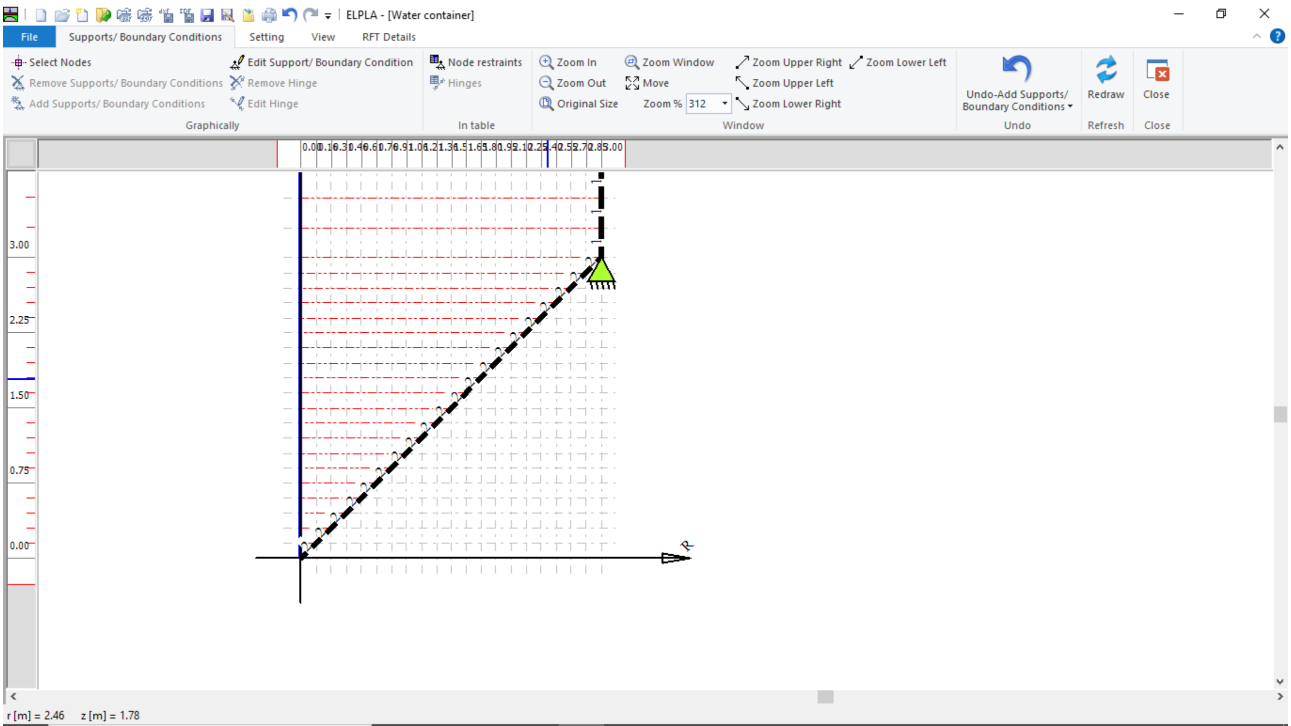


Figure 10.20 "Supports/ Boundary Conditions" window after defining the support

After defining the supports, do the following two steps

- Choose "Save " command from "File" menu in Figure 10.20 to save the data of supports
- Choose "Close" command from "File" menu in Figure 10.20 to close the "Supports/ Boundary conditions" window and return to the main window

4.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 10.21 appears.

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 10.21. In this example, there is not applied load, as the vertical load has been already defined by the unit weight of the tank, while the hydrostatic pressure on the tank wall is defined by the unit weight of water.

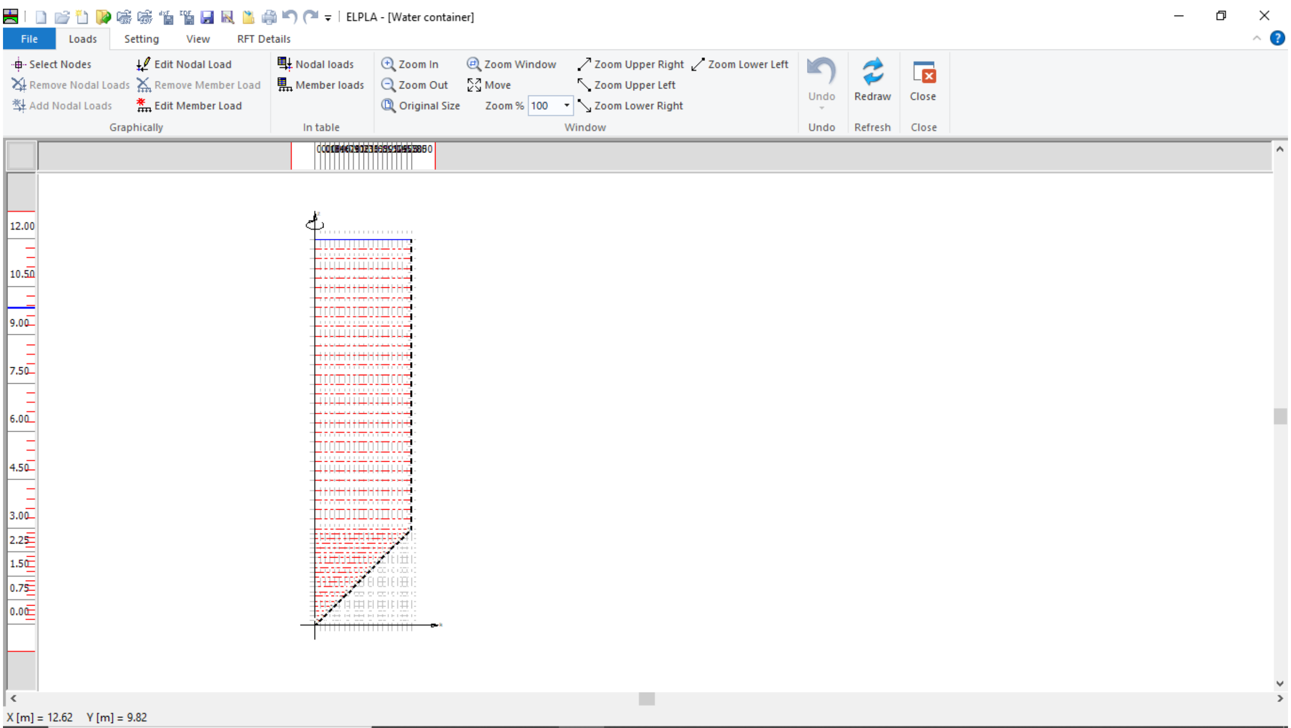


Figure 10.21 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 10.21 to save the load data
- Choose "Close" command from "File" menu in Figure 10.21 to close the "Loads" window and return to *ELPLA* main window

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

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 10.22.

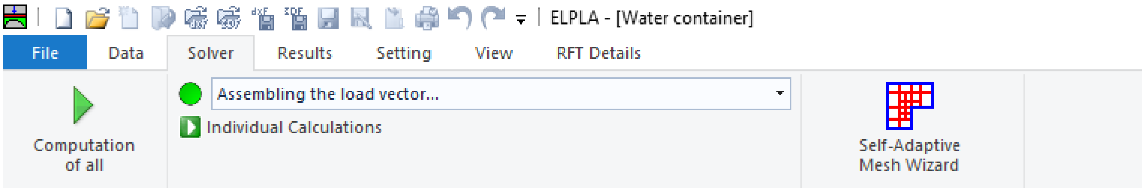


Figure 10.22 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

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

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

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab window.

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

Analysis progress

Analysis progress menu in Figure 10.23 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

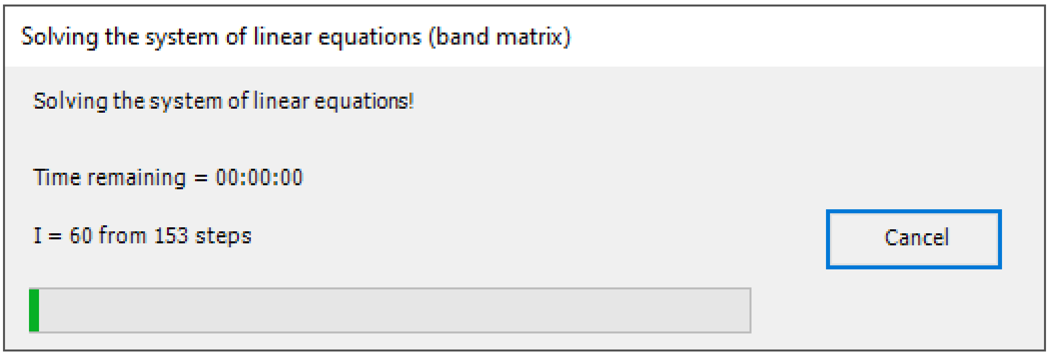


Figure 10.23 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 10.24. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

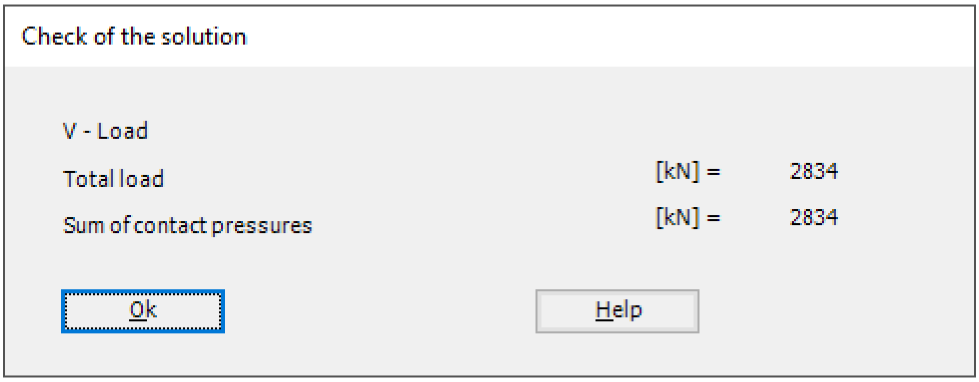


Figure 10.24 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab. To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 10.25).

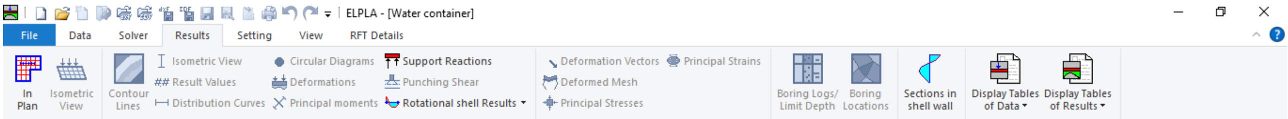


Figure 10.25 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Support Reactions
- Sections in shell wall
- Display tables of data
- Display tables of results

To view the radial forces on the shell wall

- From "Rotational shell results" command in the "Results" menu, choose "In Plan" command, the following option box in Figure 10.26 appears
- In the "Distribution of Internal Forces" option box, select "Radial forces Nr" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 10.27.

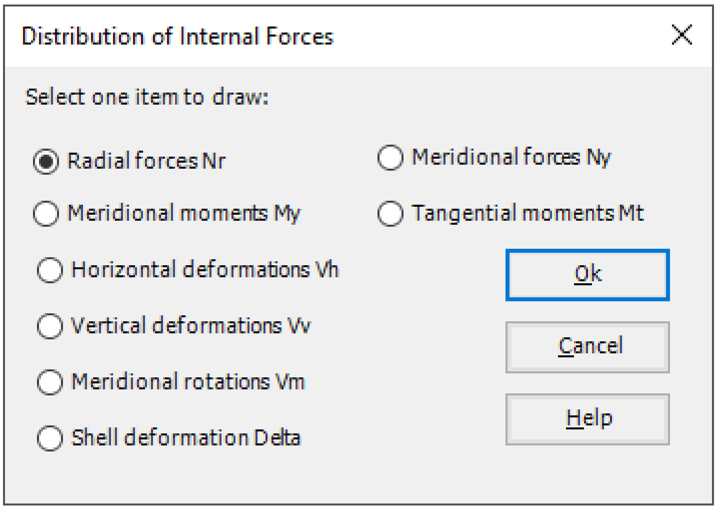


Figure 10.26 "Distribution of Internal Forces" option box

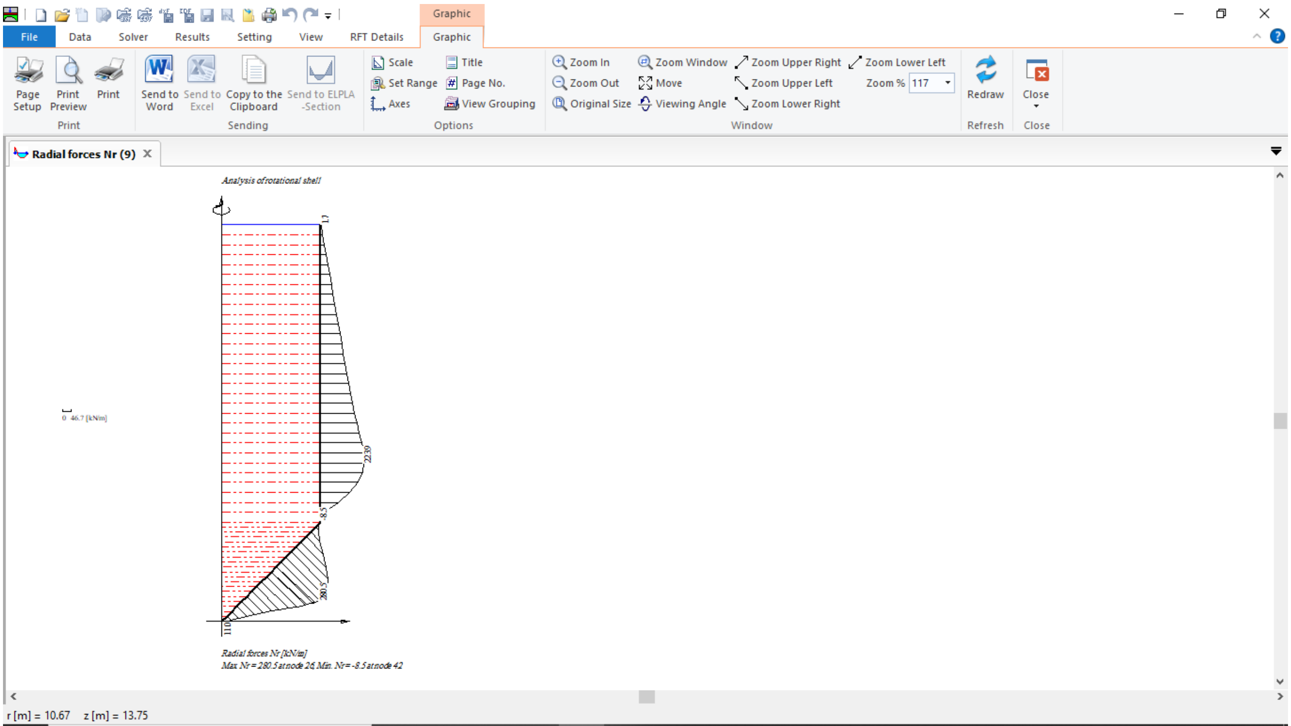


Figure 10.27 Radial forces on the shell wall

To view the meridional moments on the shell wall

- From "Rotational shell results" command in the "Results" menu, choose "In Plan" command. The following option box in Figure 10.28 appears
- In the "Distribution of Internal Forces" option box, select "Meridional moments *My*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 10.29.

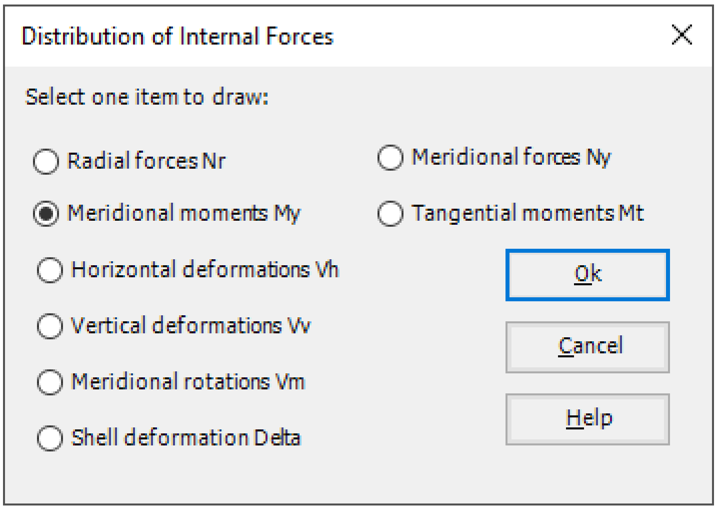


Figure 10.28 "Distribution of Internal Forces" option box

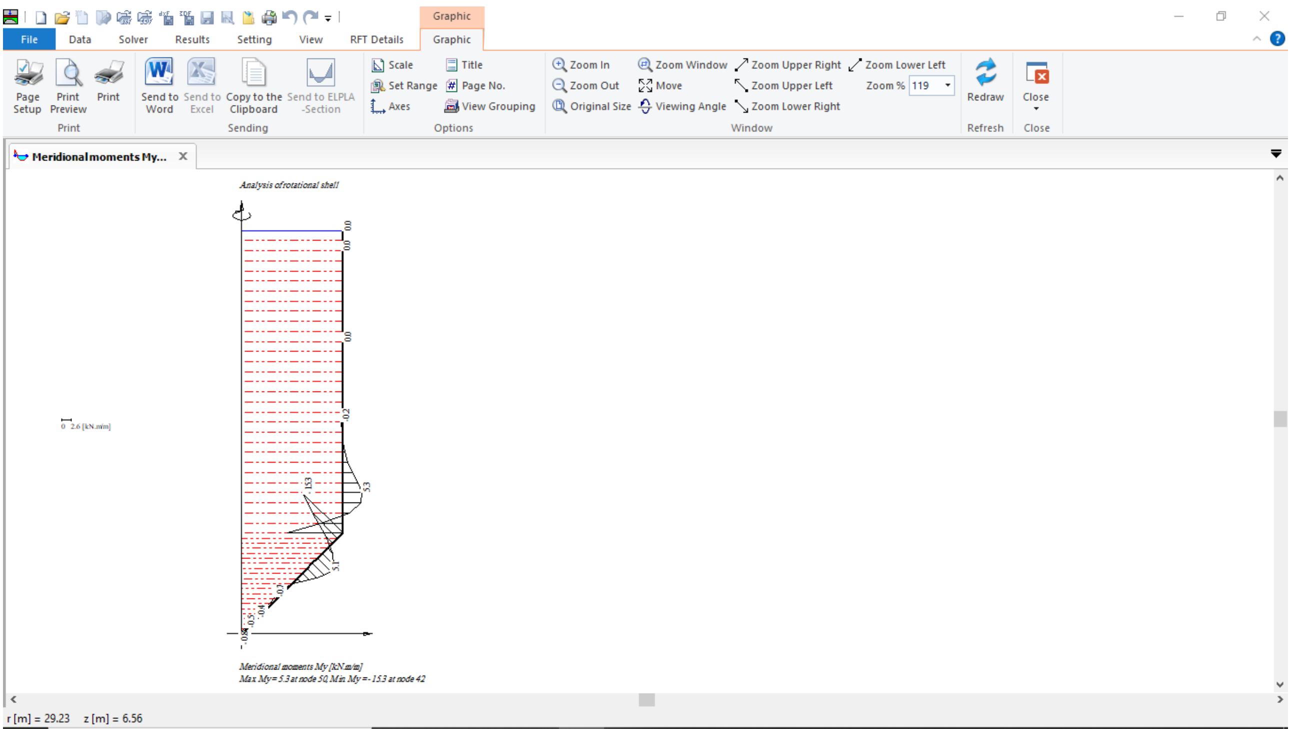


Figure 10.29 Meridional moments on the shell wall

To view element groups of the tank

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 10.30 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

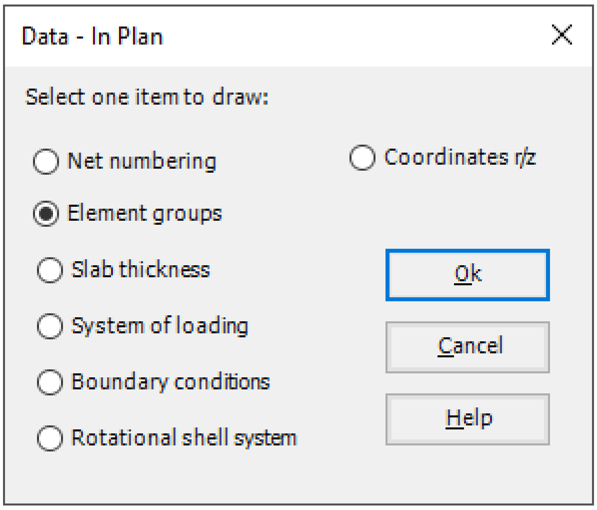


Figure 10.30 "Data – In Plan" option box

To view the supports / boundary conditions on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 10.31 appears
- In this check group box, check both "Supports Reactions *RV*" and "Supports /Boundary Conditions" check box
- The user can choose any other data to be viewed
- Click "OK" button

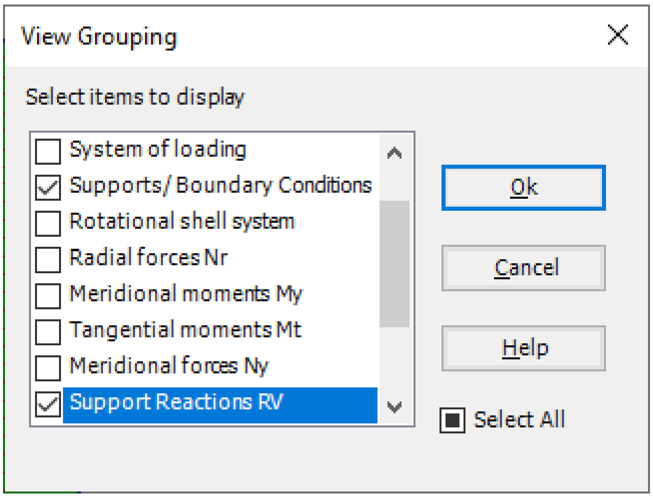


Figure 10.31 "View Grouping" check group box

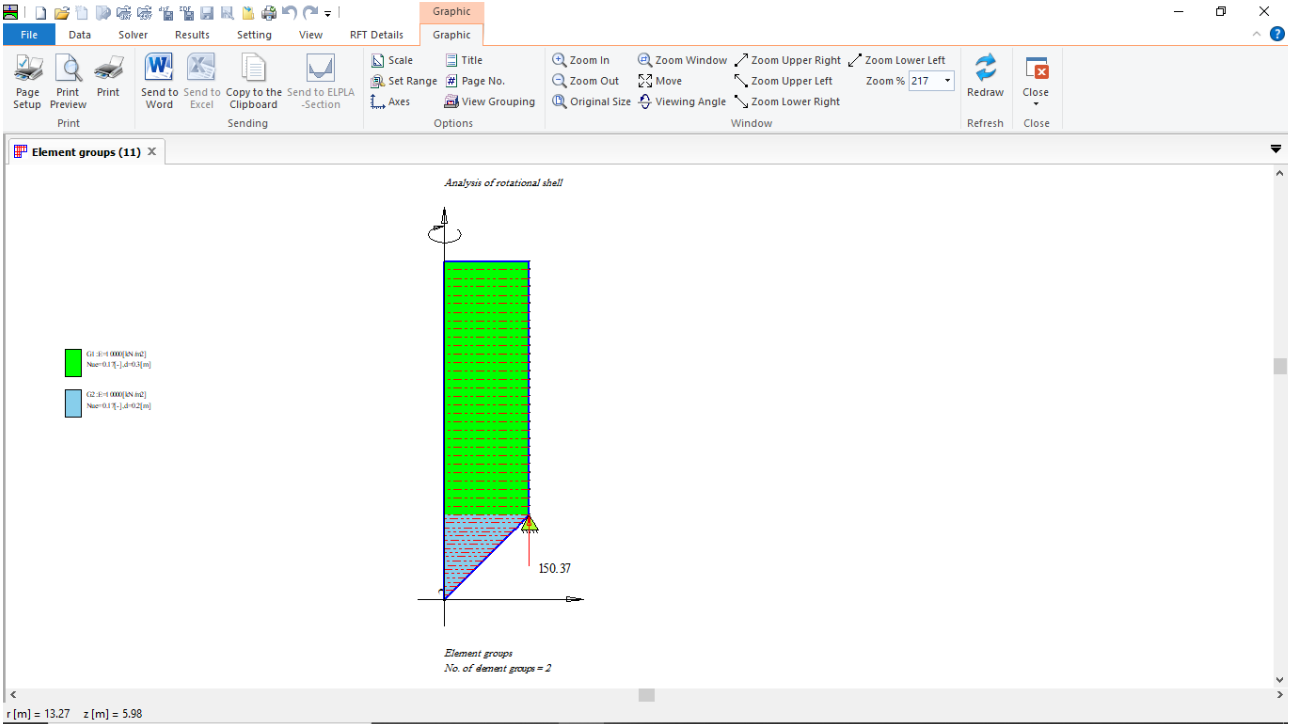


Figure 10.32 Element groups of the tank

Example 11

Analysis of a chimney
with a hyperbolic shell wall

Contents Page

1 Description of the problem

An example of an axi-symmetrically chimney with hyperbolic shell wall under a uniform external pressure is selected to illustrate some features of *ELPLA* for analyzing shell elements.

2 Chimney geometry and properties

Consider a hyperbolic shell of revolution with the following geometry:

Throat radius	R_o	= 18	[m]
Throat height	H_l	= 45	[m]
Lower radius	R_u	= 36	[m]
Total height	H	= 72	[m]
Thickness of the wall	t	=0.24	[m]

Meridian equation of the hyperbolic shell of revolution is given by:

$$r^2(\xi) = \frac{R_u^2 - R_o^2}{H_l^2}(\xi - H_l)^2 + R_o^2$$

$$r^2(\xi) = \frac{36^2 - 18^2}{45^2}(\xi - 45)^2 + 18^2$$

$$r^2(\xi) = 0.48(\xi - 45)^2 + 324$$

where r [m] is the radius at height ξ [m].

Figure 11.1 shows the geometry the chimney with its hyperbolic shell wall with dimensions and supports, while the shell material are listed in Table 11.1.

Table 11.1 hyperboloid shell material

Modulus of Elasticity of the chimney material	E_c	= 3×10^7	[kN/m^2]
Poisson's ratio of the chimney material	ν_c	= 0.3	[-]

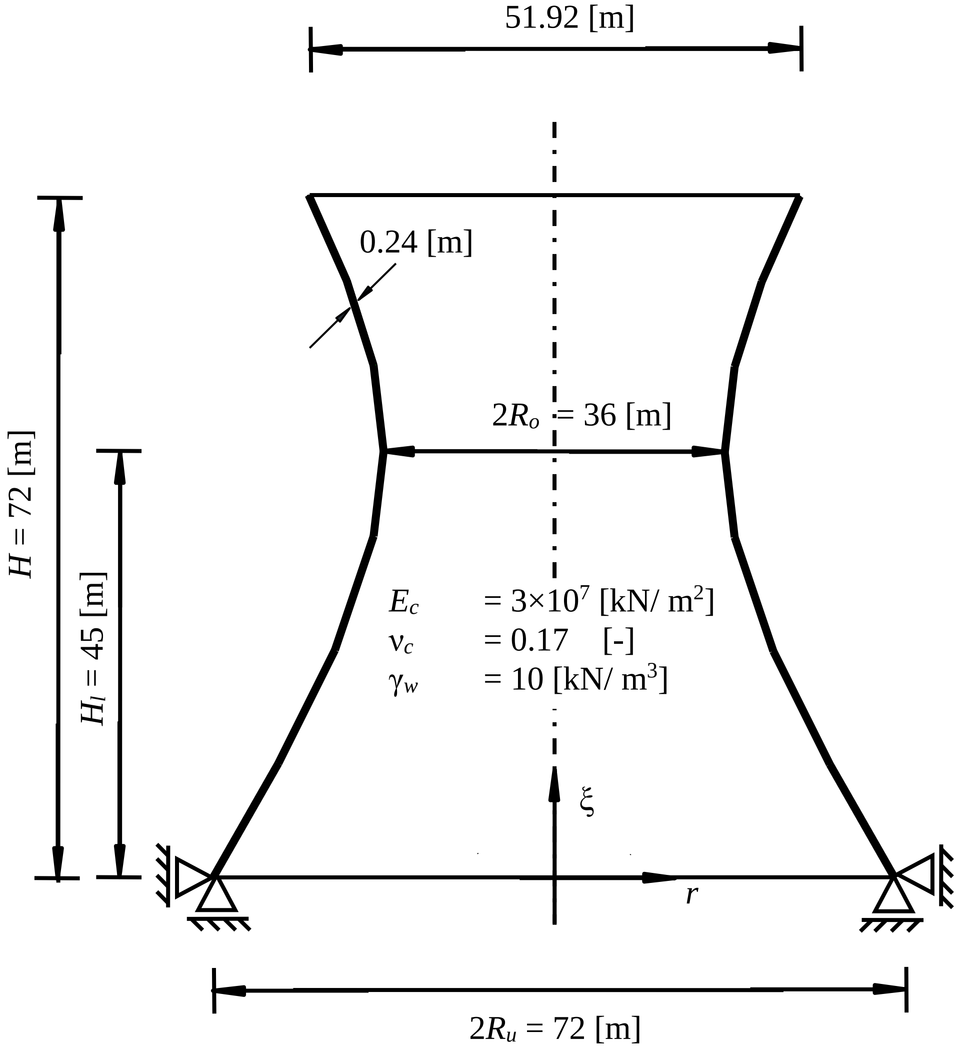


Figure 11.1 Geometry of the chimney with dimensions and supports

3 Numerical Analysis

In the analysis, the height of the chimney is divided into 7 main segments; each segment is divided into a number of elements. Segment dimensions and number of elements of each segment are shown in Figure 11.2.

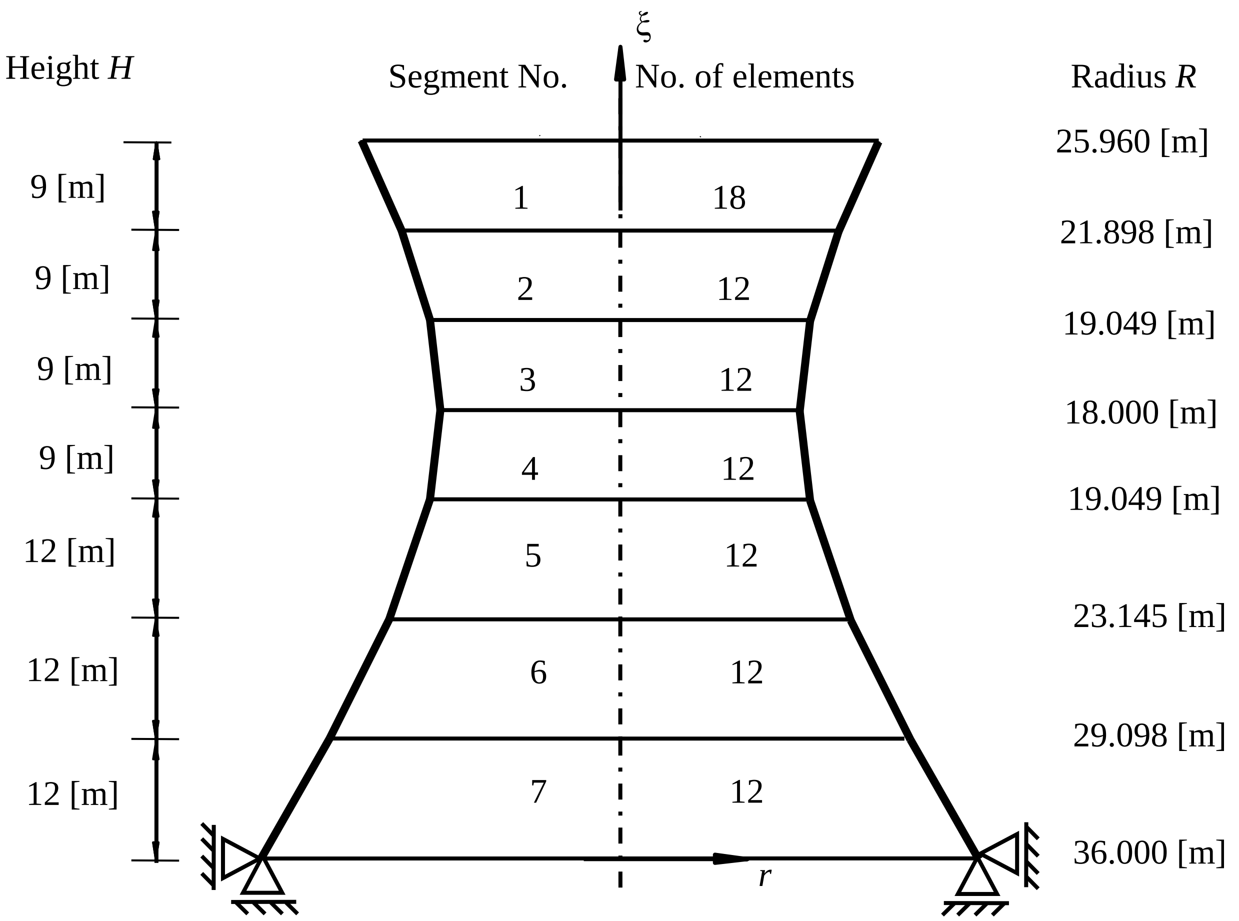


Figure 11.2 Segment dimensions and no. of elements in each segment

The chimney is exposed to a uniform external pressure of p_s = -10 [kN/m^2] as shown in Figure 11.3

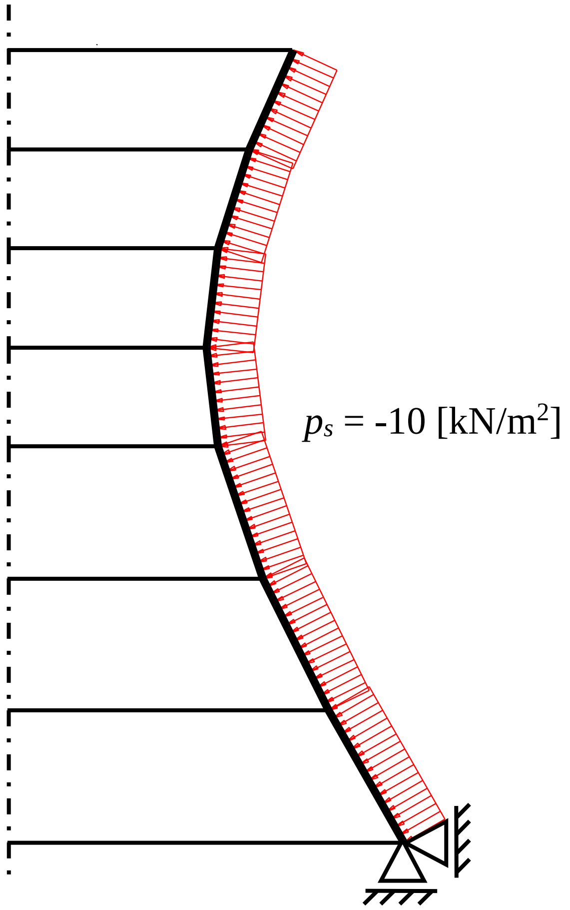


Figure 11.3 chimney with uniform external pressure of p_s = -10 [kN/m^2]

4 Creating the project

In this section, the user will learn how to create a project for analyzing a chimney with hyperbolic shell wall under a uniform external pressure. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

4.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 11.4. 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 11.4).

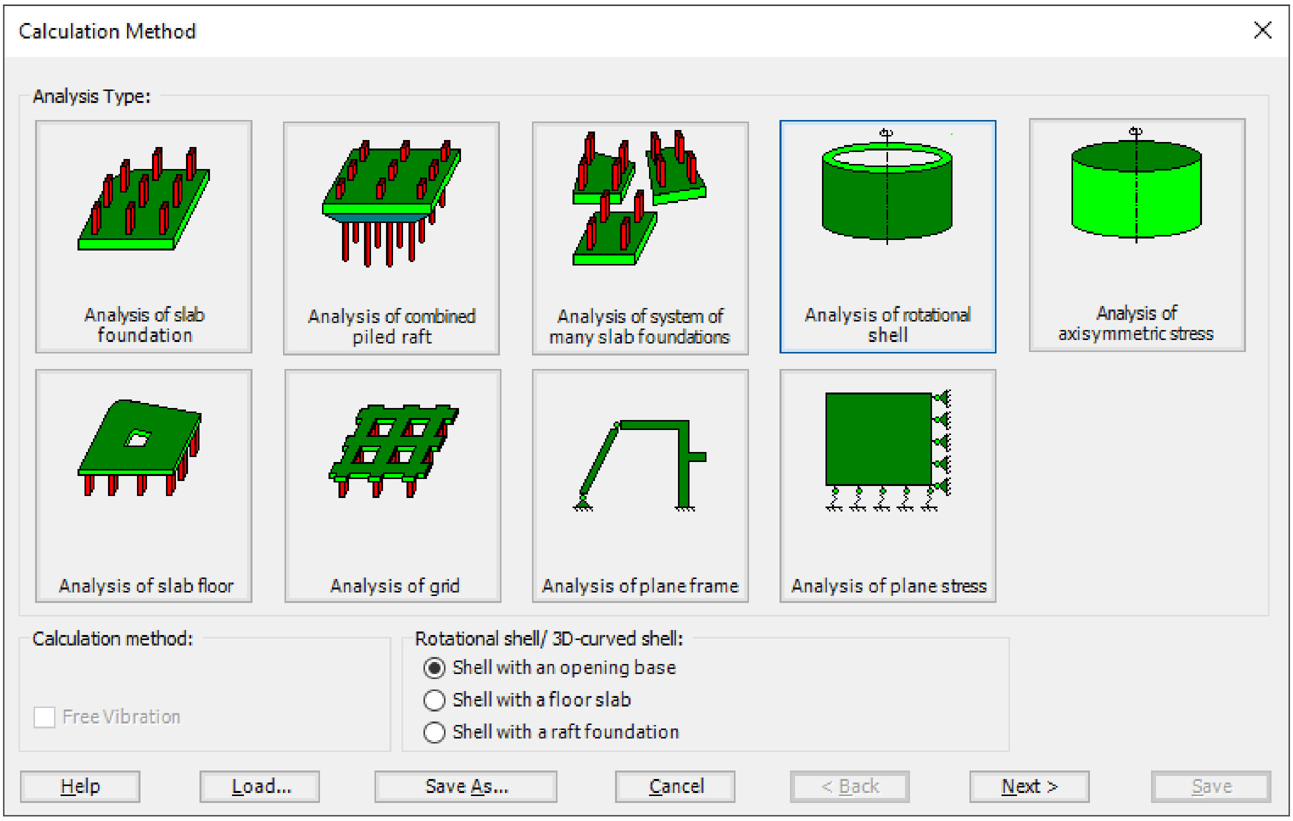


Figure 11.4 "Analysis Type" Form

In the "Analysis Type" Form in Figure 11.4, define the analysis type of the problem. As the analysis type is a hyperbolic shell problem, select "Analysis of rotational Shell" button, and check "Shell with an opening base" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Options" Form appears, Figure 11.5.

The last Form in the wizard is the "Options" Form, Figure 11.5. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Select "Supports/ Boundary Conditions", then click the "Save" button.

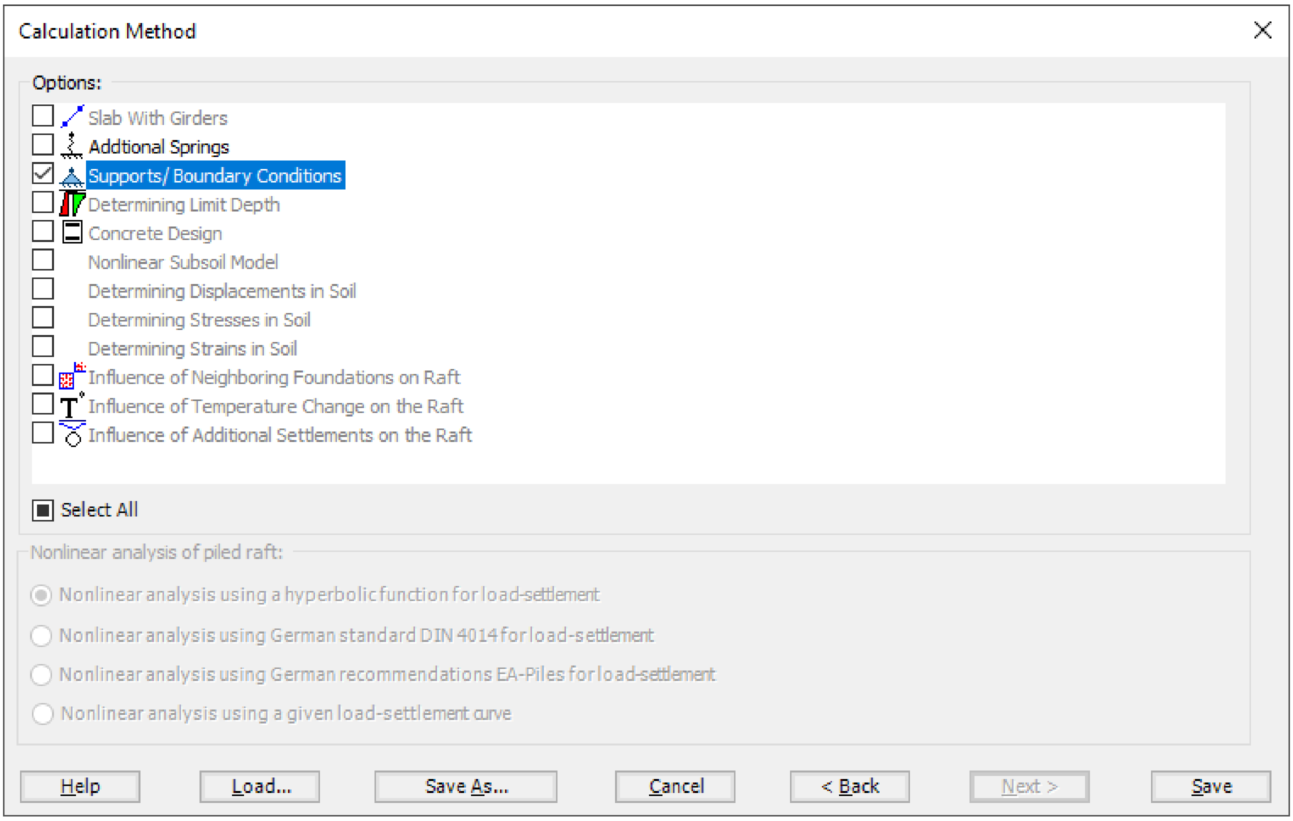


Figure 11.5 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 11.6. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Hyperbolic Chimney". *ELPLA* will use automatically this file name in all reading and writing processes.

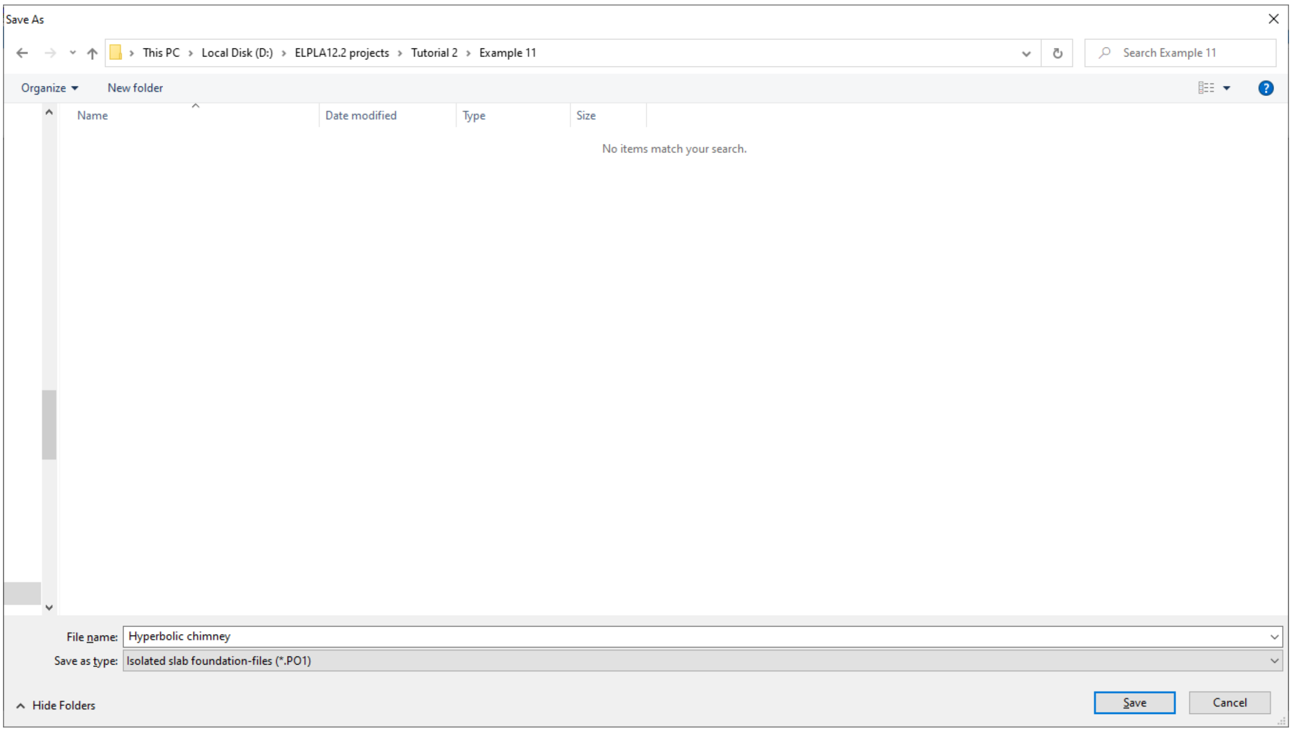


Figure 11.6 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Hyperbolic Chimney] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

4.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 11.7 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a chimney with hyperbolic shell wall"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

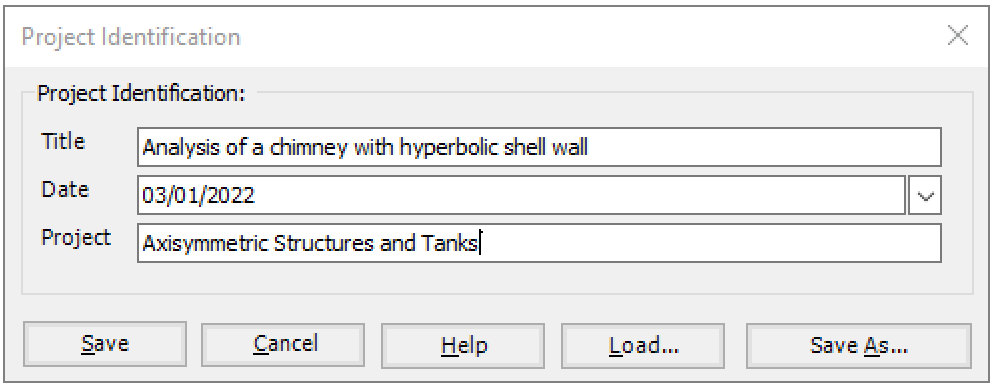


Figure 11.7 "Project Identification" dialog box

4.3 FE-Net data

For the given problem, the chimney has a lower radius of R_u = 36 [m] and a total height of H = 72 [m]. the wall height is divided into 7 main segments, each segment is divided into a number of elements. To define the FE-Net for this tank, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 11.8. This wizard will guide you through the steps required to generate a FE-Net. As shown in Figure 11.8, the first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

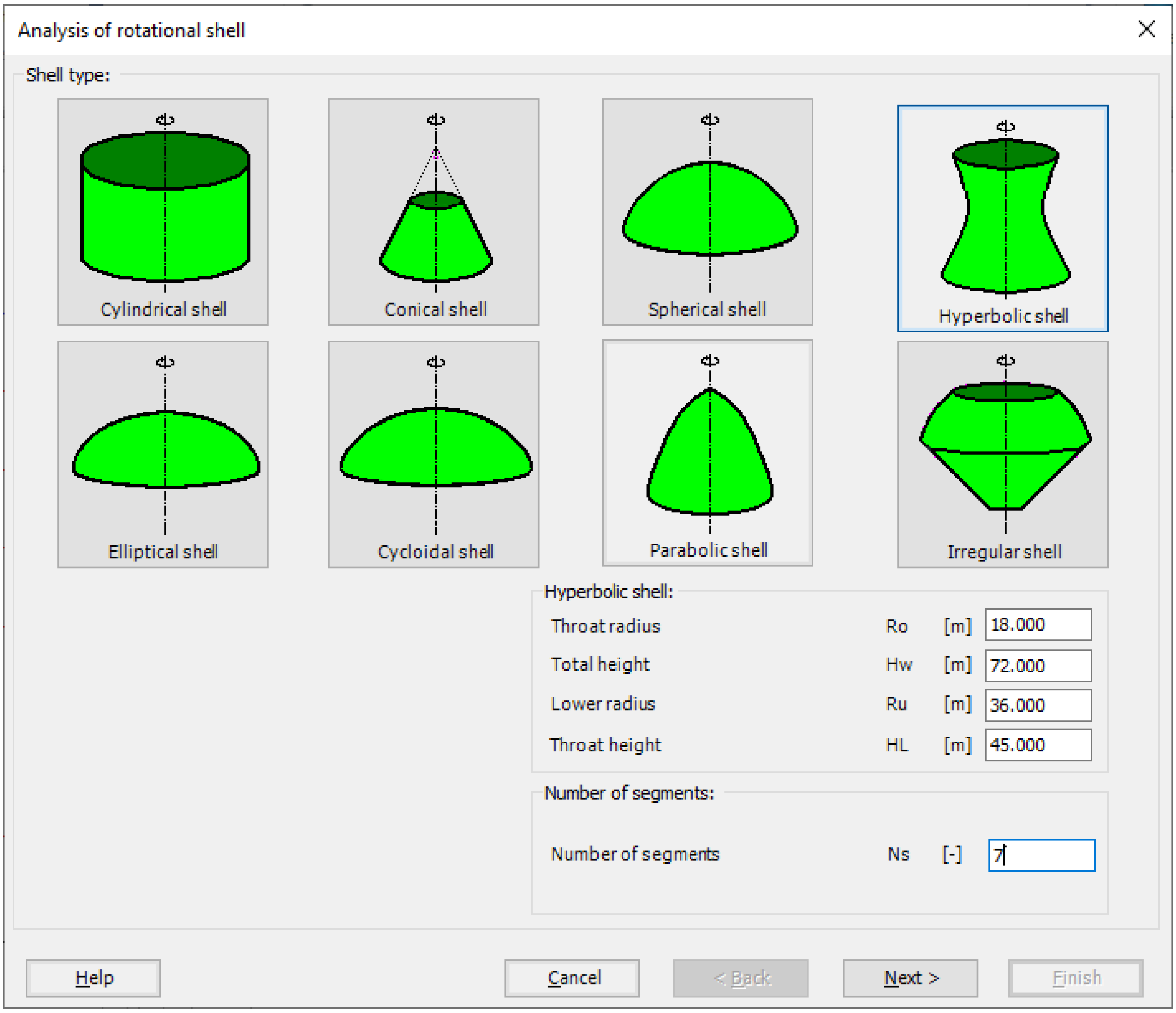


Figure 11.8 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Hyperbolic shell" button
- Type 18 in the "Throat radius" edit box
- Type 72 in the "Total height" edit box
- Type 36 in the "Lower radius" edit box
- Type 45 in the "Throat height" edit box
- Type 7 in the "Number of segments" edit box
- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Hyperbolic shell" Form appears Figure 11.9, *ELPLA* divides the height of the chimney wall into 7 equal segments, The user can edit the data of the segments individually by using "Modify" button, or all of them using "In Table" button.

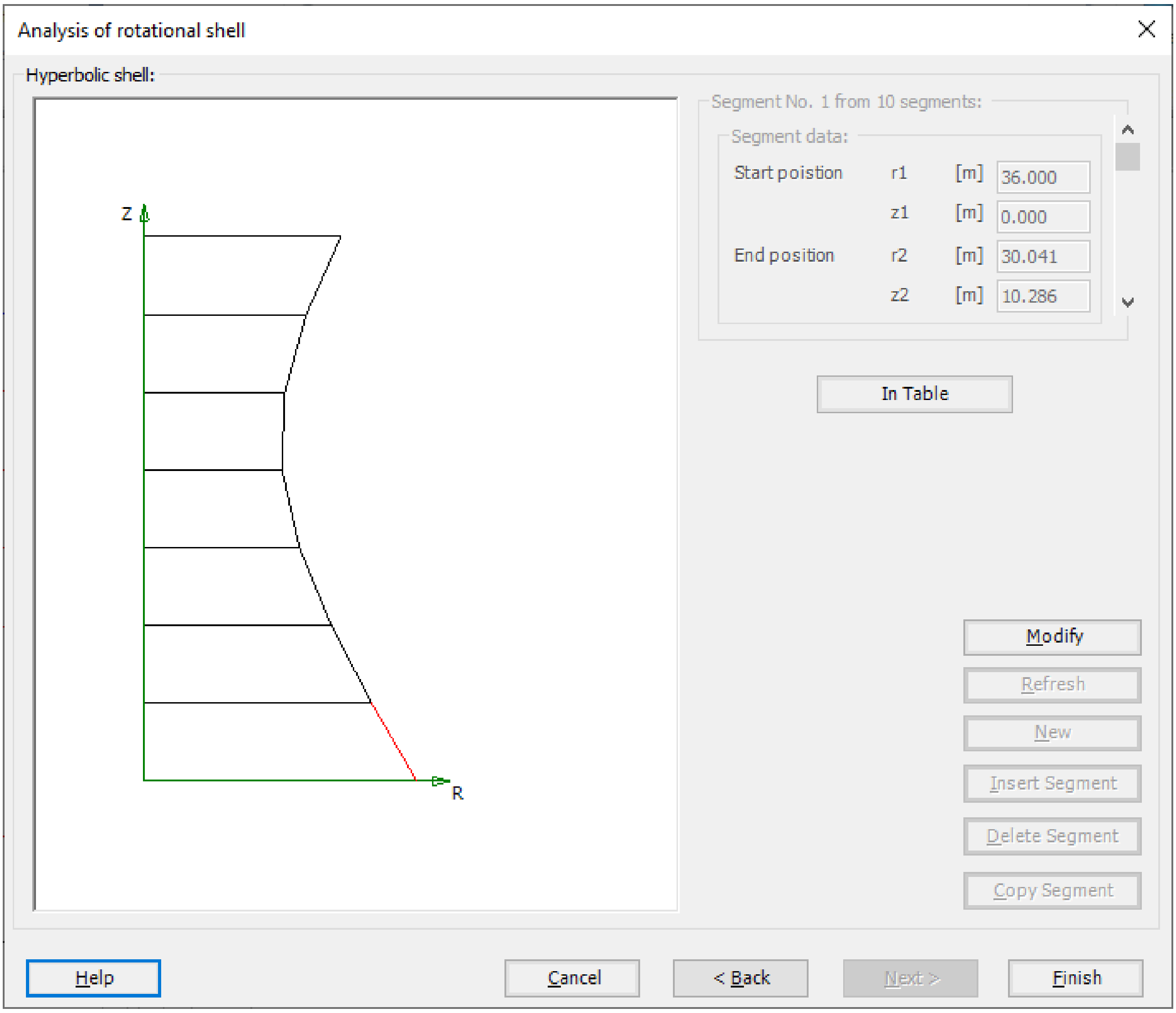
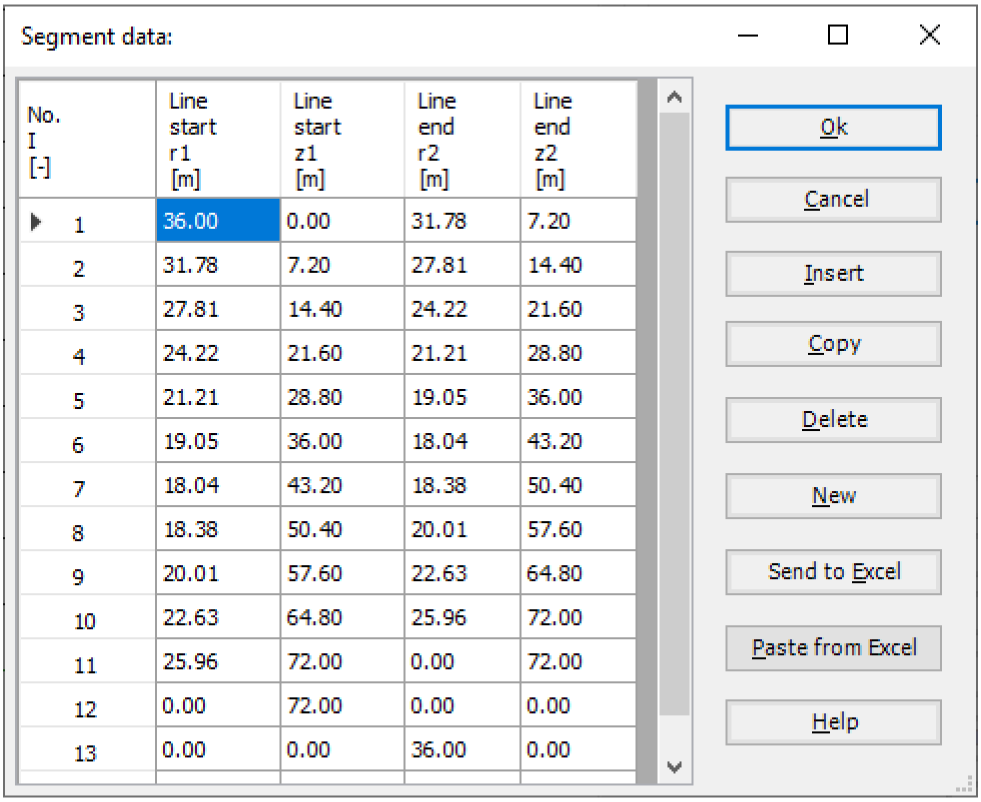


Figure 11.9 "Hyperbolic shell" Form



No. I [-]	Line start r1 [m]	Line start z1 [m]	Line end r2 [m]	Line end z2 [m]
1	36.00	0.00	31.78	7.20
2	31.78	7.20	27.81	14.40
3	27.81	14.40	24.22	21.60
4	24.22	21.60	21.21	28.80
5	21.21	28.80	19.05	36.00
6	19.05	36.00	18.04	43.20
7	18.04	43.20	18.38	50.40
8	18.38	50.40	20.01	57.60
9	20.01	57.60	22.63	64.80
10	22.63	64.80	25.96	72.00
11	25.96	72.00	0.00	72.00
12	0.00	72.00	0.00	0.00
13	0.00	0.00	36.00	0.00

Figure 11.10 "Segment data" Table

After modifying the coordinates of the segments, click "Finish" button, the FE-Net of the chimney appears in Figure 11.11

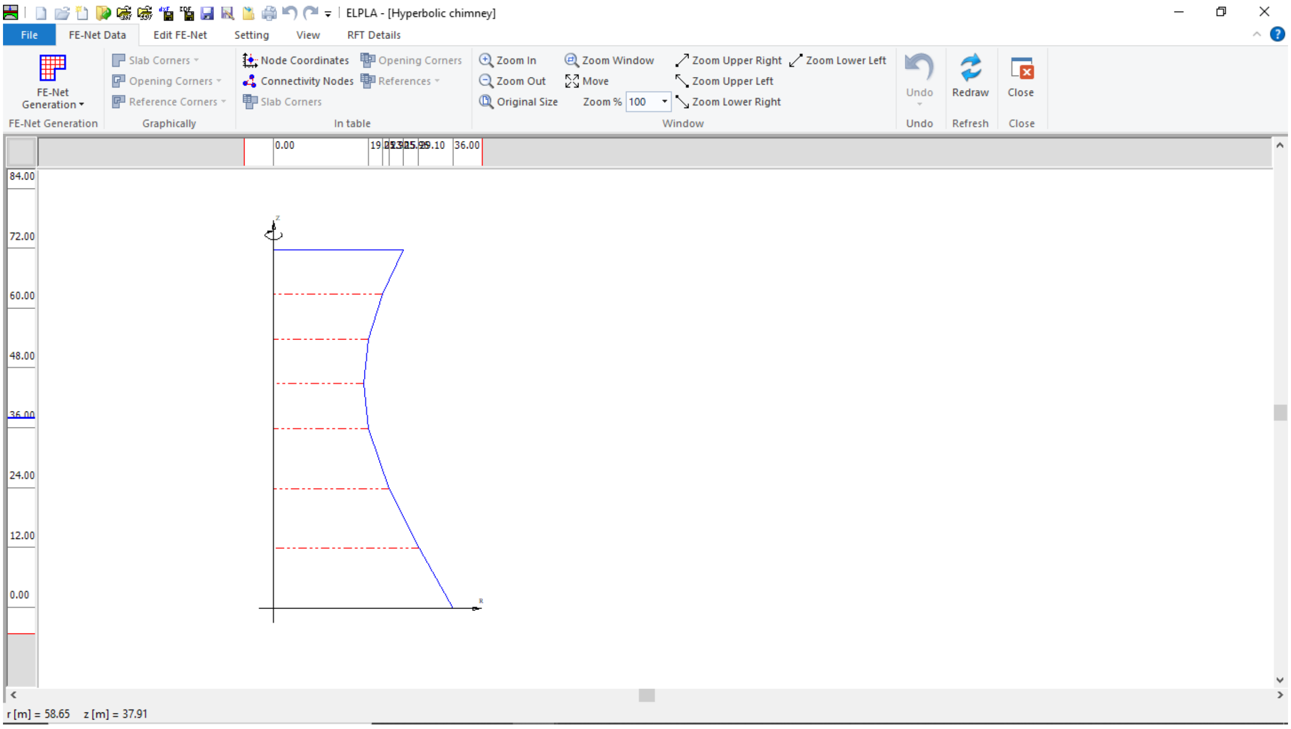


Figure 11.11 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 11.11 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 11.11 to close the "FE-Net" window and return to *ELPLA* main window

4.4 Shell properties

To define the chimney properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 11.12 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, unit weight of the chimney, and element size.

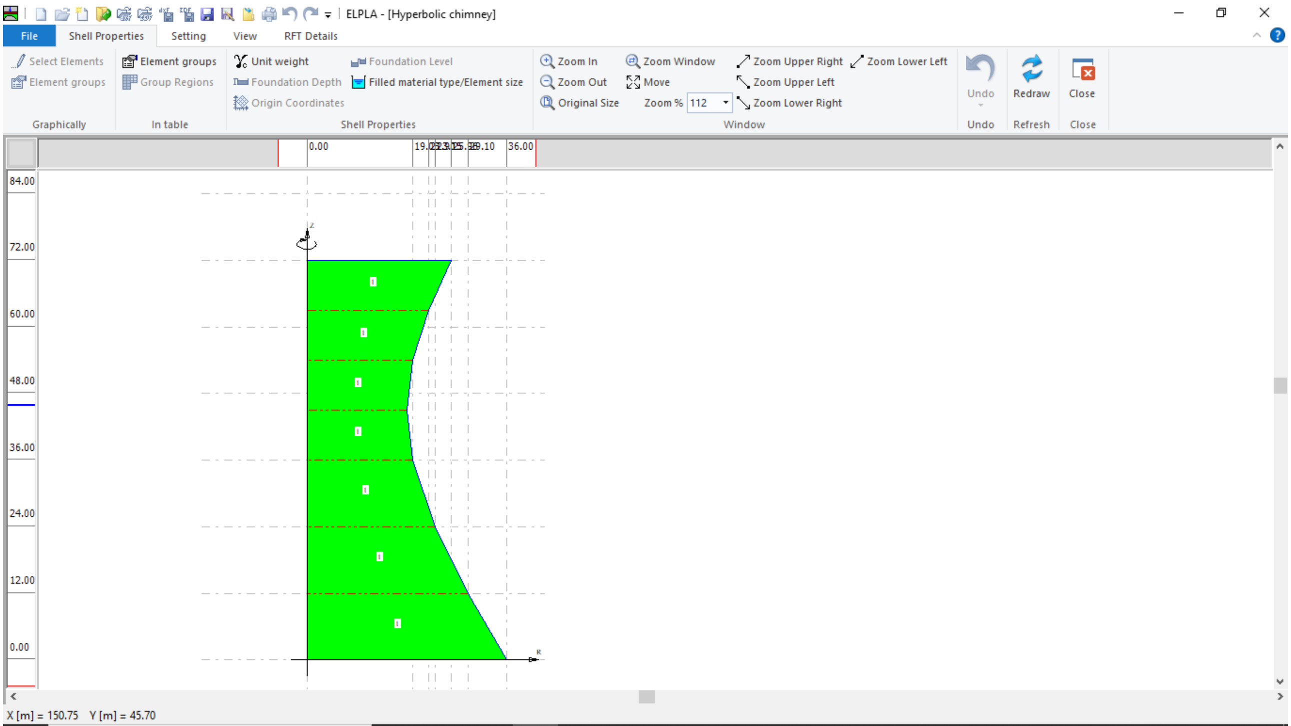


Figure 11.12 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 11.13 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness. Then click "OK" button.

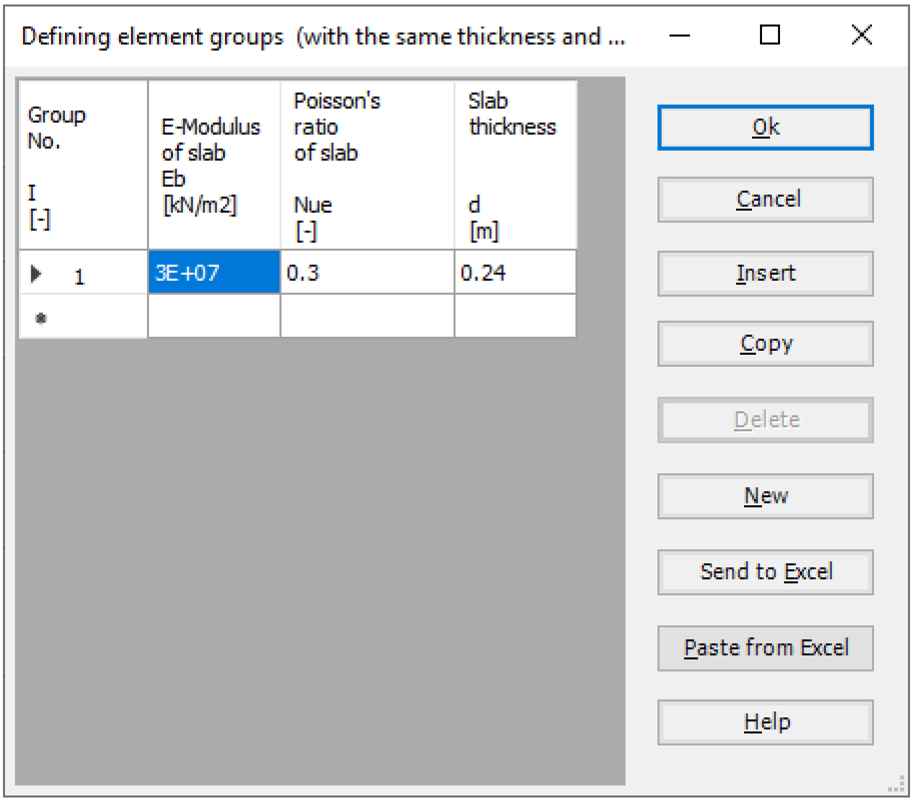


Figure 11.13 "Defining element groups" list box

To enter the unit weight of the chimney, choose "Unit weight" command from "Shell Properties" menu in Figure 11.12. The following dialog box in Figure 11.14 with a default unit weight of 25 [kN/m^3] appears, type 0 in the "Unit weight" edit box to neglect the self-weight of the chimney, then click "OK" button.

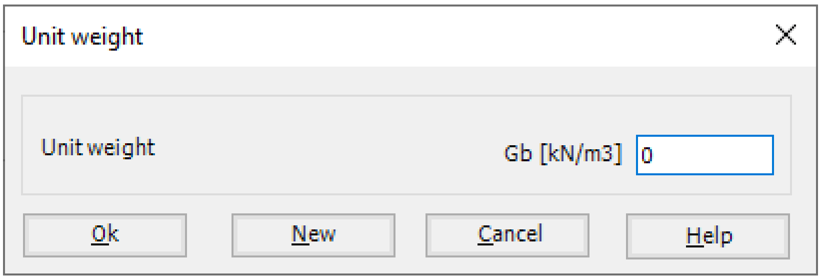


Figure 11.14 "Unit weight" dialog box

To define the element size of the chimney, choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 11.12. The following form in Figure 11.15 appears, then:

- Select "Empty container" option
- uncheck the "Constant element sizes in z-direction" check box
- Click "Element size in each segment" button, the following list box in Figure 11.16 appears
- Define the size of the element in each segment as in the following list box in Figure 11.16, to keep the same number of elements in each segment
- Click "OK" button

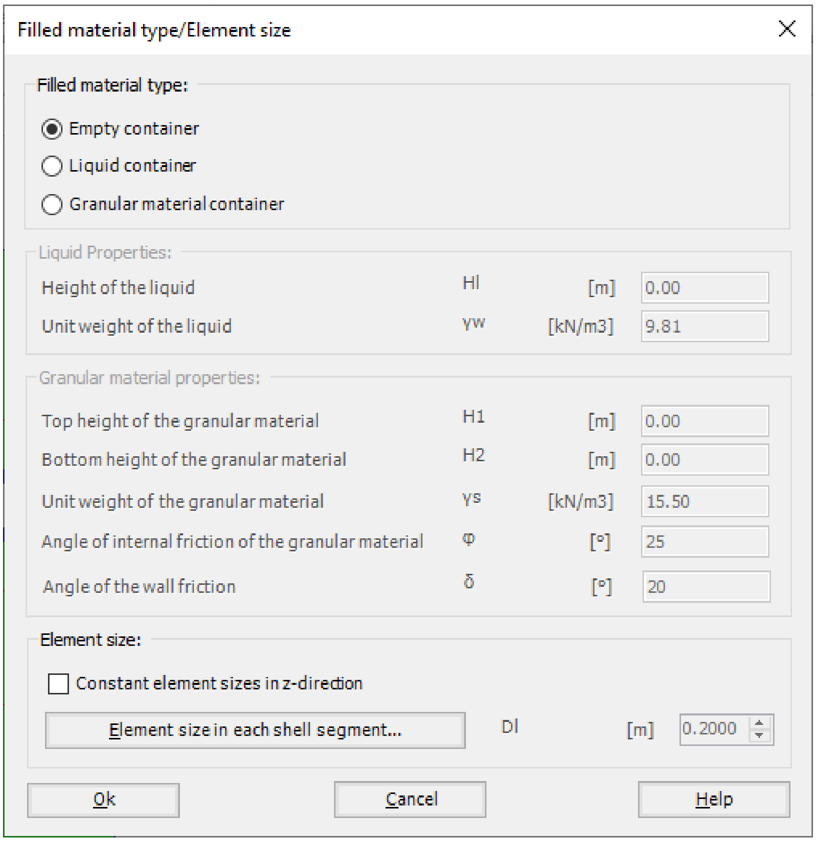


Figure 11.15 "Filled material type/Element size" Form

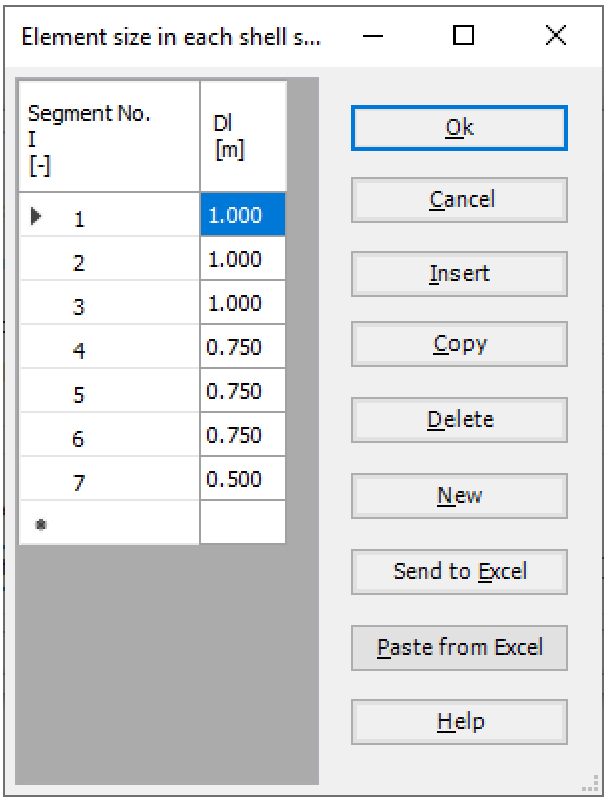


Figure 11.16 "Element size in each segment" list box

After defining the chimney properties, do the following two steps:

- Choose "Save" command from "File" menu in Figure 11.12 to save the shell properties
- Choose "Close" command from "File" menu in Figure 11.12 to close the "Shell properties" window and return to *ELPLA* main window

4.5 Supports/ Boundary Conditions

To define the hinged support, choose "Supports/ Boundary Conditions" command from “Data” Tab. The following window in Figure 11.17 appears.

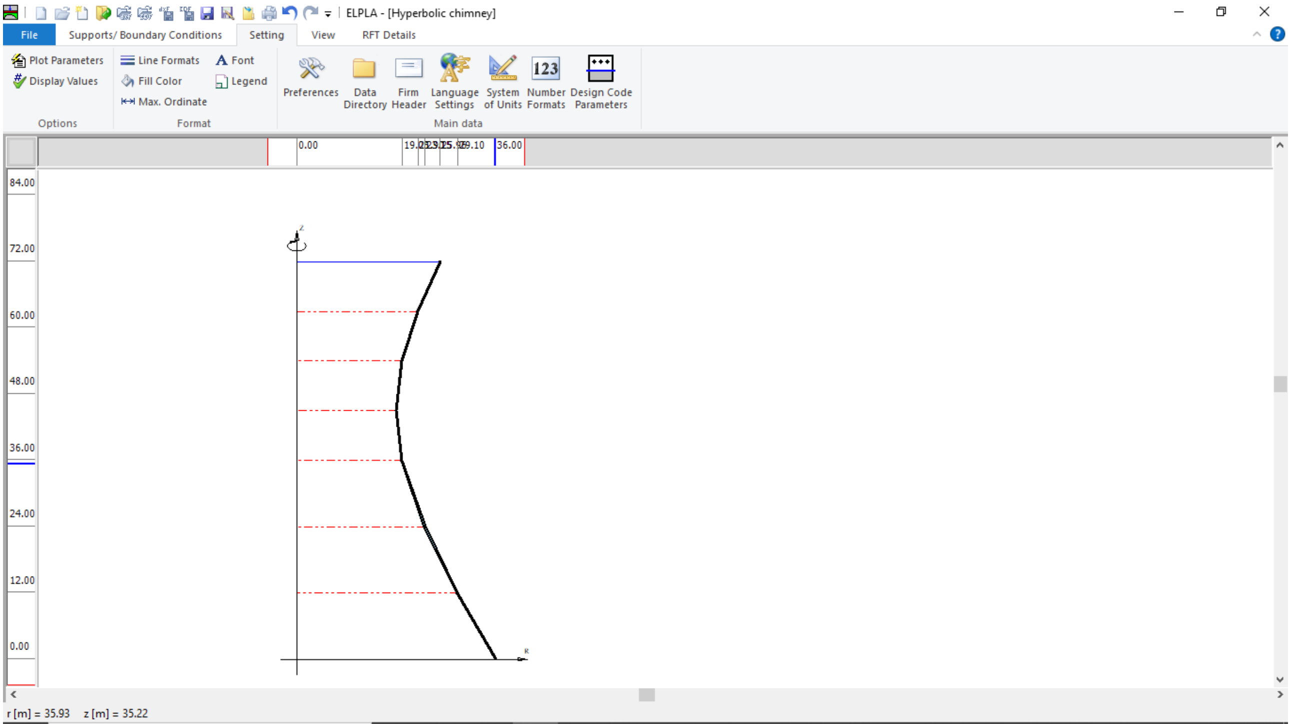


Figure 11.17 "Supports/ Boundary Conditions" Window

To define the support on the net:

- Choose "Select Nodes" command from "Graphically" menu in Figure 11.17. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the node that has the support as shown in Figure 11.18
- After selecting the node, choose "Add Supports/ Boundary Conditions" command from "Graphically" menu (Figure 11.17). The "Supports/ Boundary Conditions" dialog box in Figure 11.19 appears.

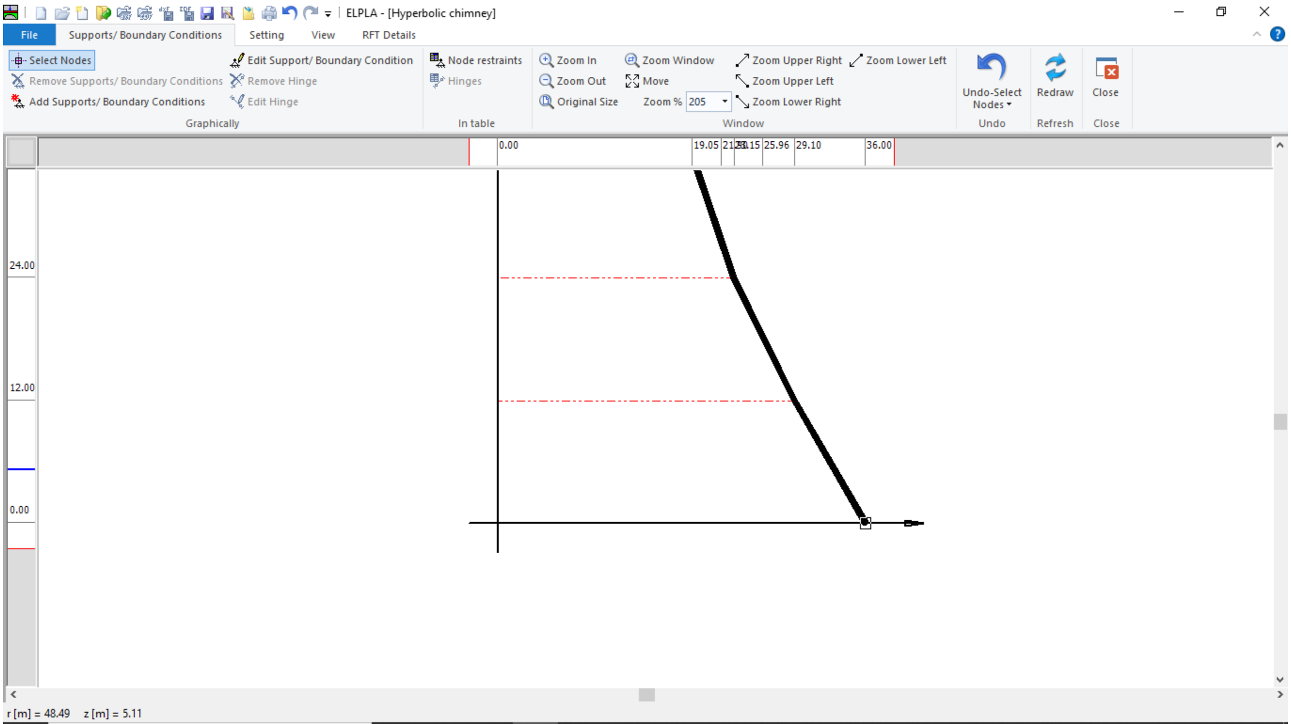


Figure 11.18 Selection of the node that has a hinged support

In this dialog box

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

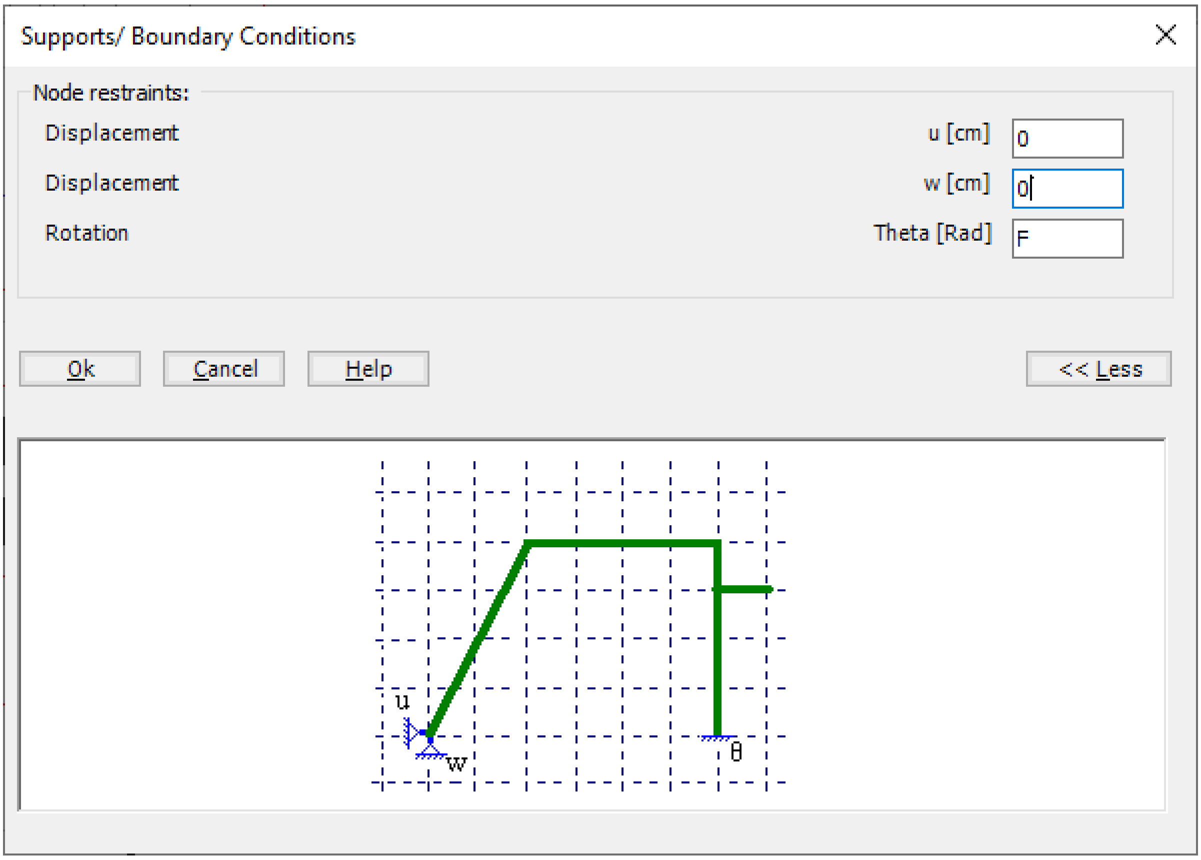


Figure 11.19 "Supports/ Boundary Conditions" dialog box

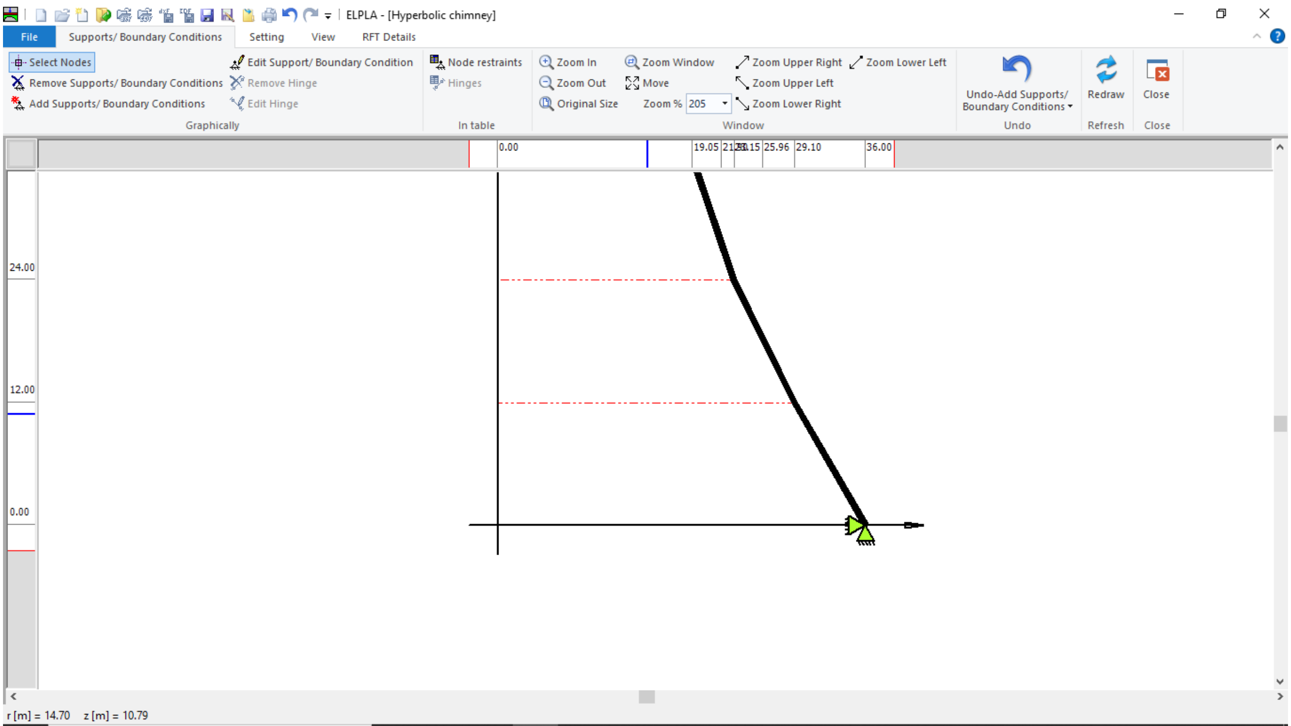


Figure 11.20 Supports on the screen

After defining the supports, do the following two steps

- Choose "Save " command from "File" menu in Figure 11.17 to save the data of supports
- Choose "Close" command from "File" menu in Figure 11.17 to close the "Supports/ Boundary conditions" window and return to the main window.

4.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 11.21 appears.

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 11.21. In this example, the hyperbolic chimney is exposed to a uniform external pressure of p_s = -10 [kN/m^2].

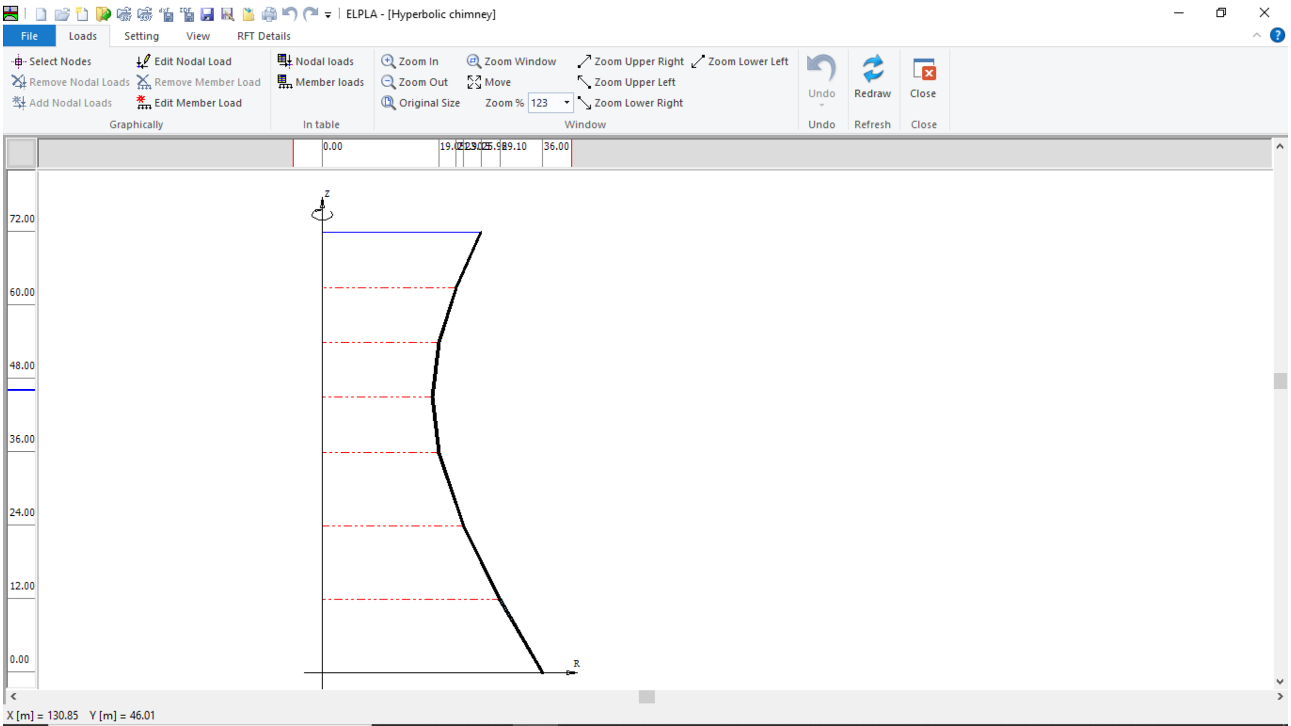


Figure 11.21 "Loads" Window

To define the uniform distributed load:

- choose "Edit Member Load" command from "Graphically" menu in Figure 11.21. When "Edit Member Load" command is chosen, the cursor will change from an arrow to a cross hair
- Double click on the element which is exposed to the uniform distributed load
- "Edit Member Load" dialog box in Figure 11.22 appears, type "-10" in "Member loads" edit box
- Click "OK" button

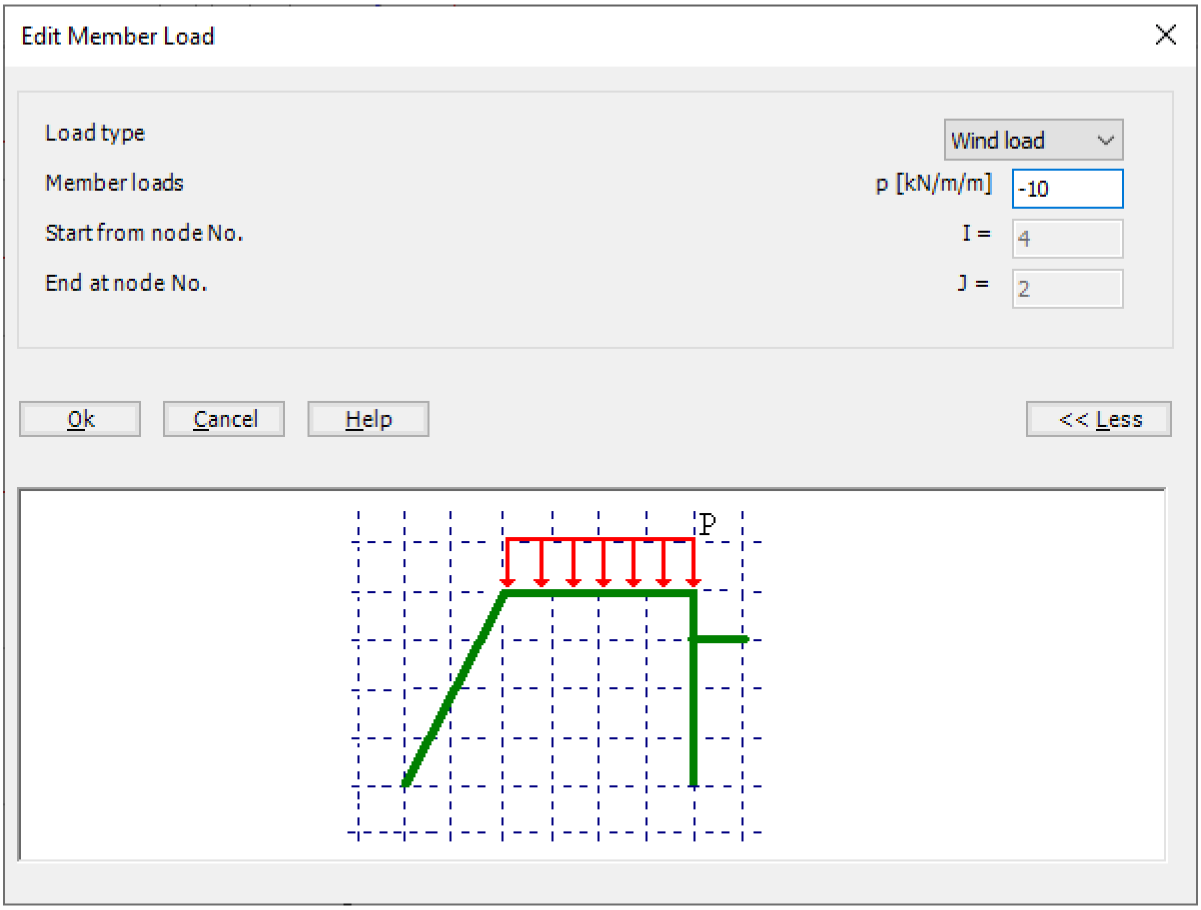


Figure 11.22 "Edit Member Load" dialog box

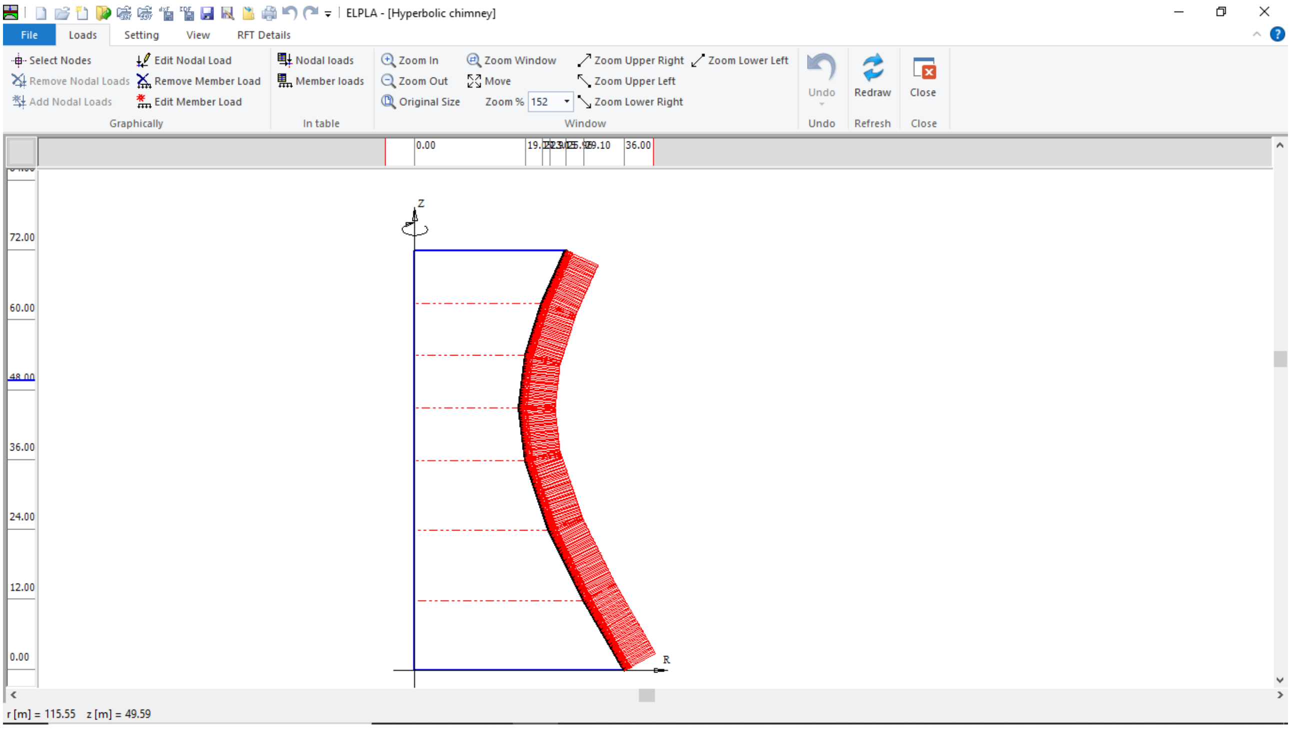


Figure 11.23 "Loads" window after defining the uniform distributed load

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

- Choose "Save" command from "File" menu in Figure 11.21 to save the load data
- Choose "Close" command from "File" menu in Figure 11.21 to close the "Loads" window and return to *ELPLA* main window.

Creating the project of the chimney with a hyperbolic shell wall is now complete. It is time to analyze this project. In the next section, you will learn how to use *ELPLA* for analyzing projects.

5 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 11.24.

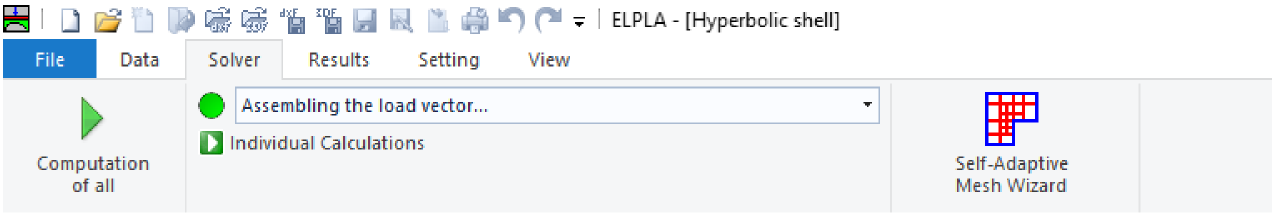


Figure 11.24 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

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

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

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab window.

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

Analysis progress

Analysis progress menu in Figure 11.25 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

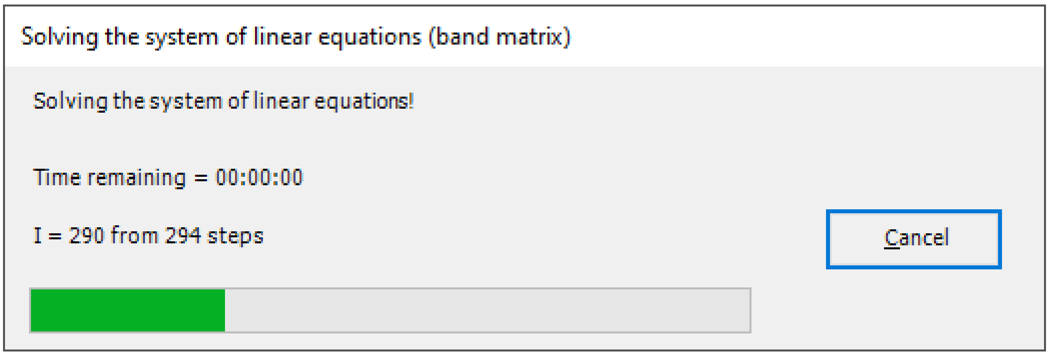


Figure 11.25 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 11.26. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

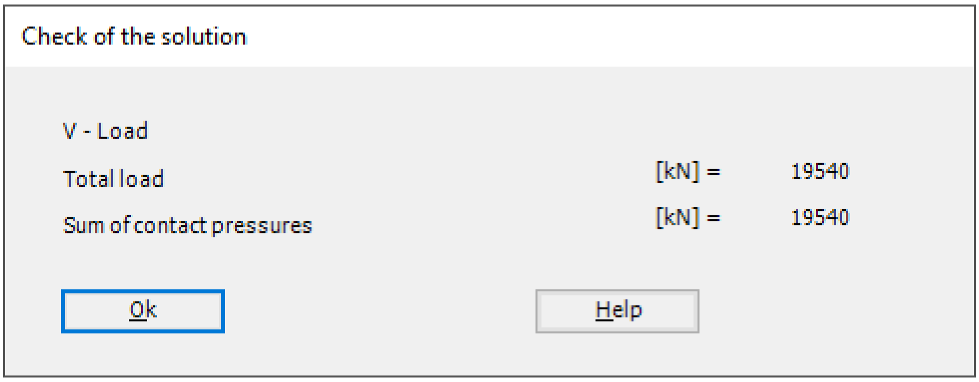


Figure 11.26 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab. To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 11.27).

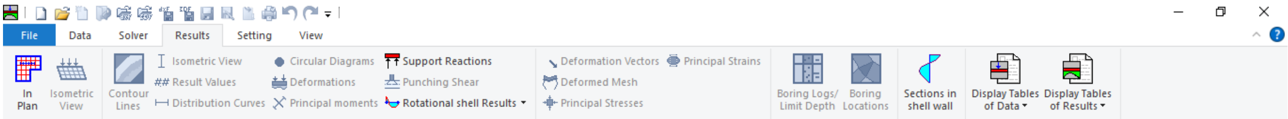


Figure 11.27 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Supports reactions
- Sections in shell wall
- Display tables of data
- Display tables of results

To view the radial forces in the shell wall

- From "Rotational shell results" command in the "Results" menu, choose "In Plan" command. The following option box in Figure 11.28 appears
- In the "Distribution of Internal Forces" option box, select "Radial forces Nr" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 11.29.

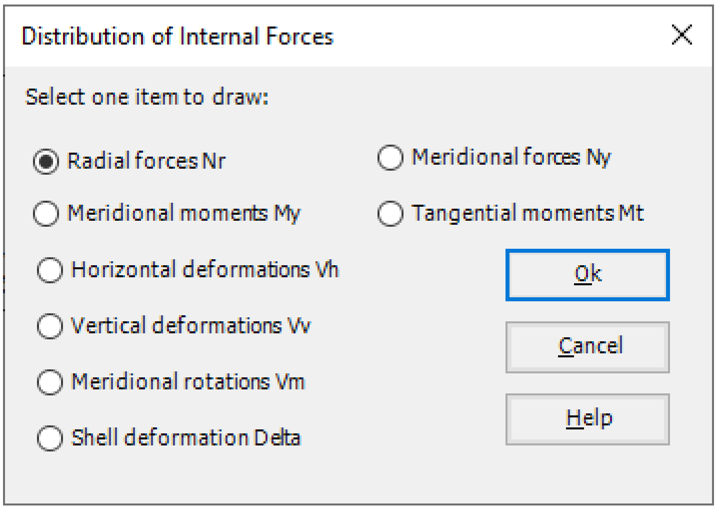


Figure 11.28 "Distribution of internal forces" option box

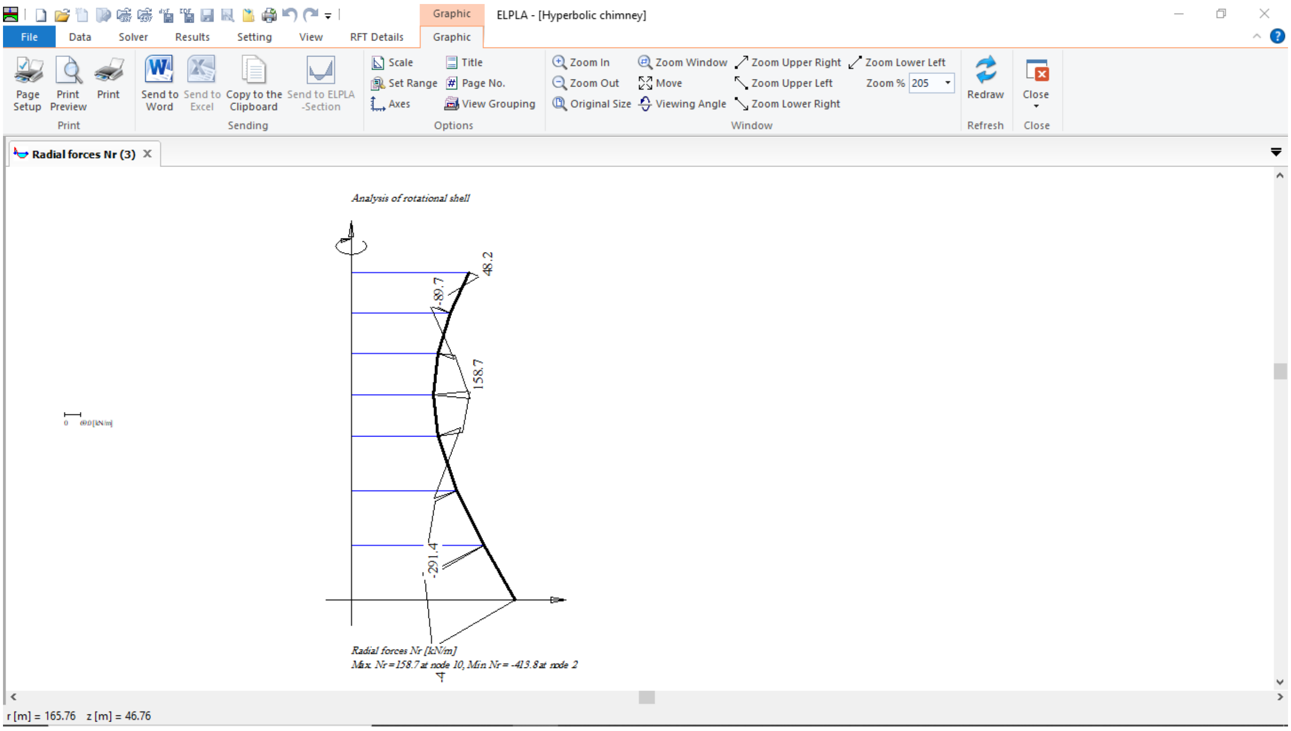


Figure 11.29 Radial forces in shell wall

To view the vertical deformations of the chimney wall

- From "Rotational shell results" command in the "Results" menu, choose "In Plan" command. The following option box in Figure 11.30 appears
- In the "Distribution of Internal Forces" option box, select "Vertical deformations *Vv*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 11.31.

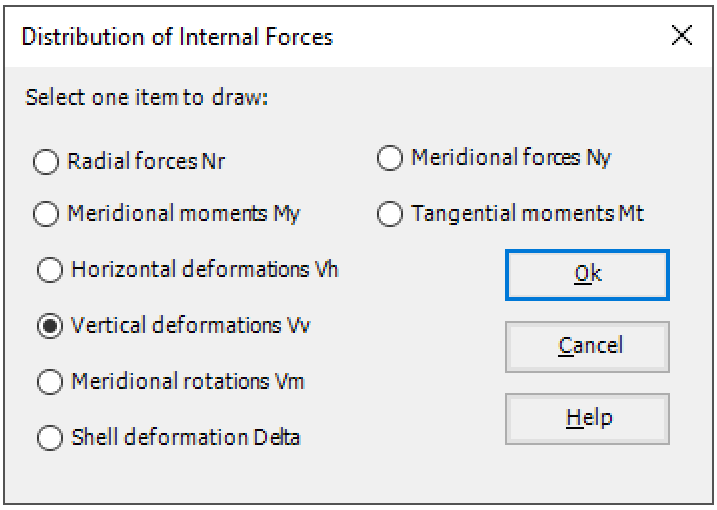


Figure 11.30 "Distribution of internal forces" option box

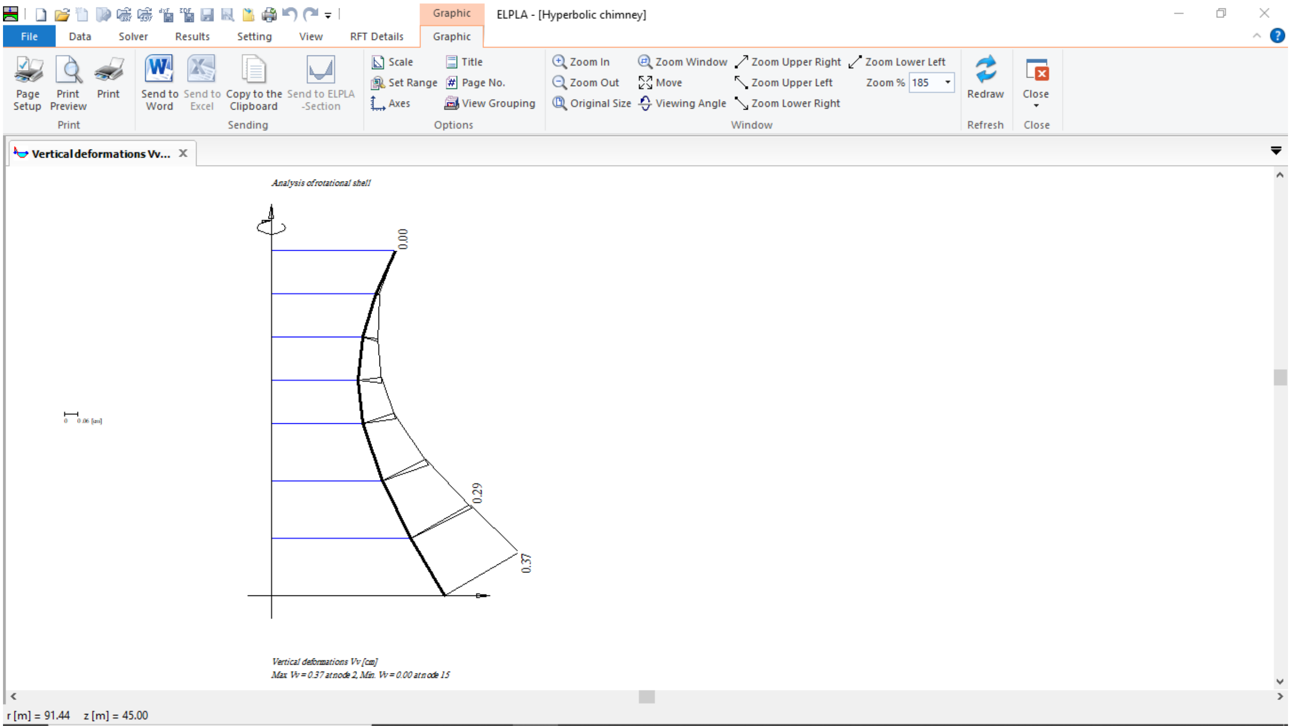


Figure 11.31 Vertical deformations of the chimney wall

To view the slab thickness of the chimney

- Choose "Slab thickness" from "In Plan" command in "Data" menu. The following option box in Figure 11.32 appears
- Click "OK" button

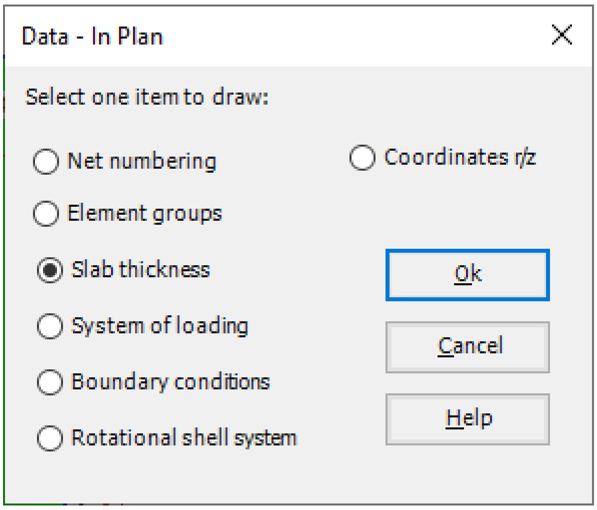


Figure 11.32 "Data – In Plan" option box

To view the supports / boundary conditions on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 11.33 appears
- In this check group box, check "Supports /Boundary Conditions" check box
- The user can choose any other data to be viewed
- Click "OK" button

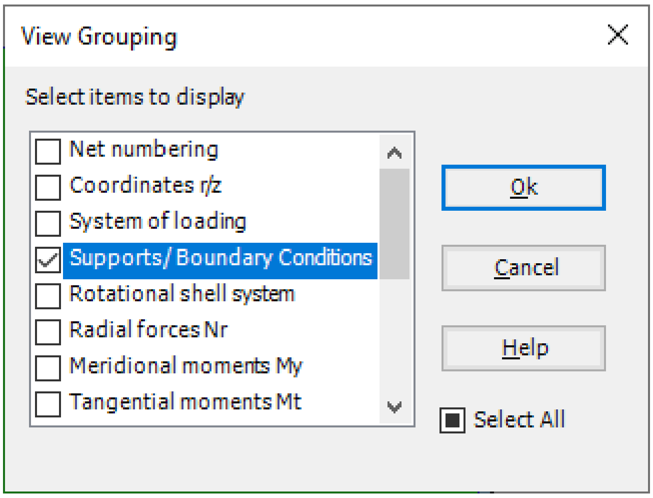


Figure 11.33 "View Grouping" check group box

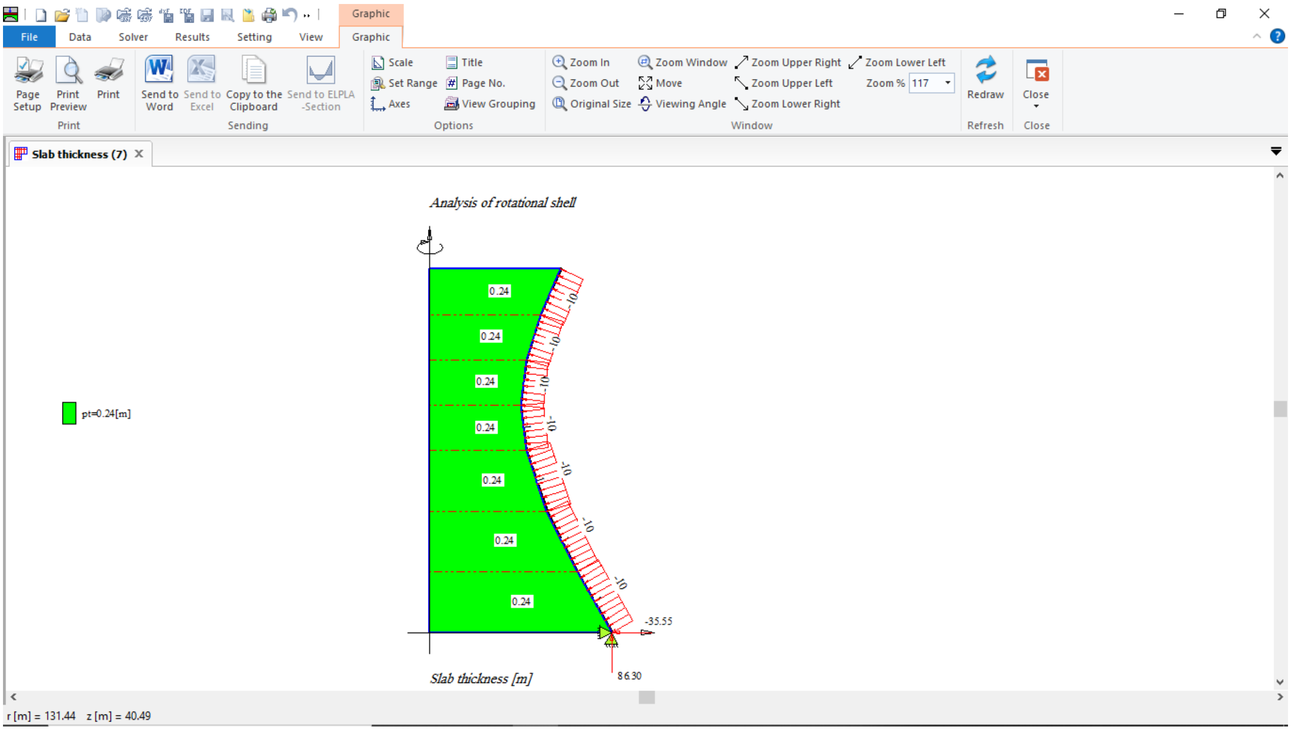


Figure 11.34 Slab thickness with the load system

Example 12

Analysis of an irregular container resting on a layered soil

Contents Page

1 Description of the problem

An example of an axi-symmetrically container with irregular shape resting on a layered soil is selected to illustrate some features of *ELPLA* for analyzing shell elements.

2 Container geometry and properties

A container having a base radius of a = 5 [m], middle wall radius of Rm = 7 [m] and a height of H = 14 [m] is considered as shown in Figure 12.1. Thicknesses of the container base, wall and roof are t_b = 0.4 [m], t_w = 0.3 [m], t_r = 0.2 [m], respectively. The container is filled with water and rests on layered soil. The container material and unit weight of the water are listed in Table 12.1.

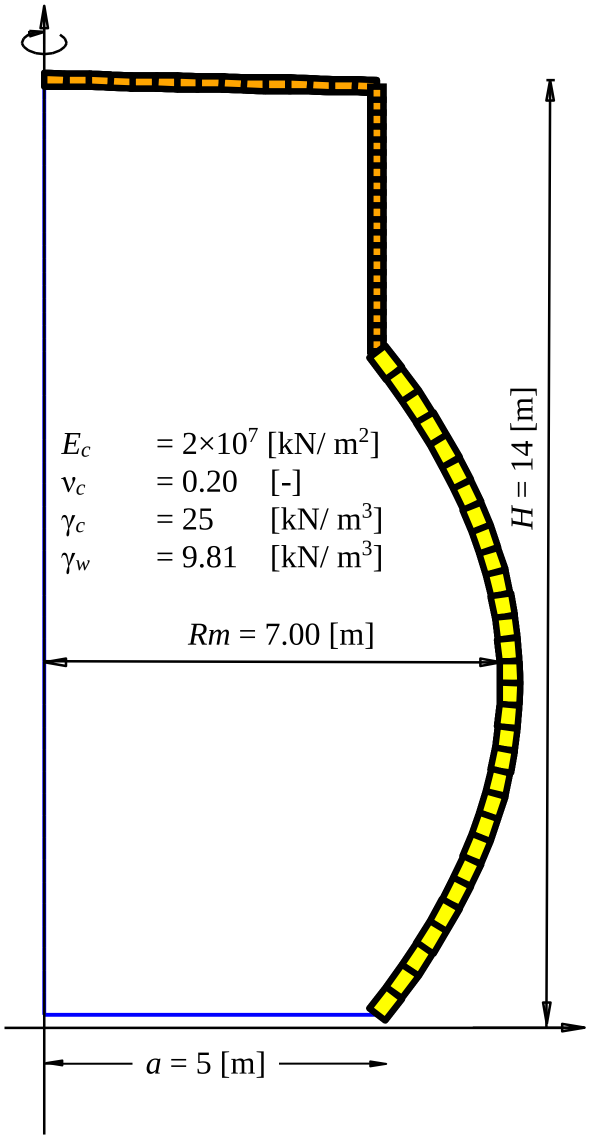


Figure 12.1 Irregular container with dimensions

Table 12.1 Container material and water unit weight

Modulus of Elasticity	E_c	$= 2\times10^7$	[kN/ m²]
Poisson's ratio	ν_c	= 0.2	[-]
Unit weight of the container material	γ_c	= 25	[kN/ m³]
Unit weight of the water	γ_w	= 10	[kN/ m³]

The geometry and parameters of the container are:

Base:

- Base radius *Rb* = 5 [m]
- Base thickness *tb* = 0.4 [m]

Wall:

- Wall thickness *tw* = 0.3 [m]
- The first main segment parameters are:
  - Height *Hw* = 10 [m]
  - Middle radius *Rm* = 7 [m]
  - Upper radius *Ro* = 5 [m]
  - Number of subsegments *Ns* = 30
- The second main segment parameters are:
  - Height *Hw* = 4 [m]
  - Middle radius *Rm* = 5 [m]
  - Upper radius *Ro* = 5 [m]
  - Number of subsegments *Ns* = 20

Roof:

- Flat roof type
- Number of subsegments *Ns* = 15
- Roof thickness *tr* = 0.2 [m]

2.1 Soil properties

The subsoil under the container consists of two layers, sandy clay and sand with gravel.

The soil properties of the sandy clay layer are:

Modulus of elasticity for loading	E	= 35000	[kN/m²]
Modulus of elasticity for reloading	W	= 105000	[kN/m²]
Poisson's ratio	ν_s	= 0.3	[-]

while those of the sand with gravel layer are:

Modulus of elasticity for loading	E	= 80000	[kN/m²]
Modulus of elasticity for reloading	W	= 240000	[kN/m²]
Poisson's ratio	ν_s	= 0.3	[-]

2.2 Numerical Analysis

In the analysis, the wall height of the container is divided into two main segments, the first one is divided into 30 subsegments (30×0.3 [m]), while the second is divided into 20 subsegments (20×0.2 [m]). The flat roof is divided into 15 subsegments. The raft foundation is divided into 10 circular intervals as shown in Figure 12.2.

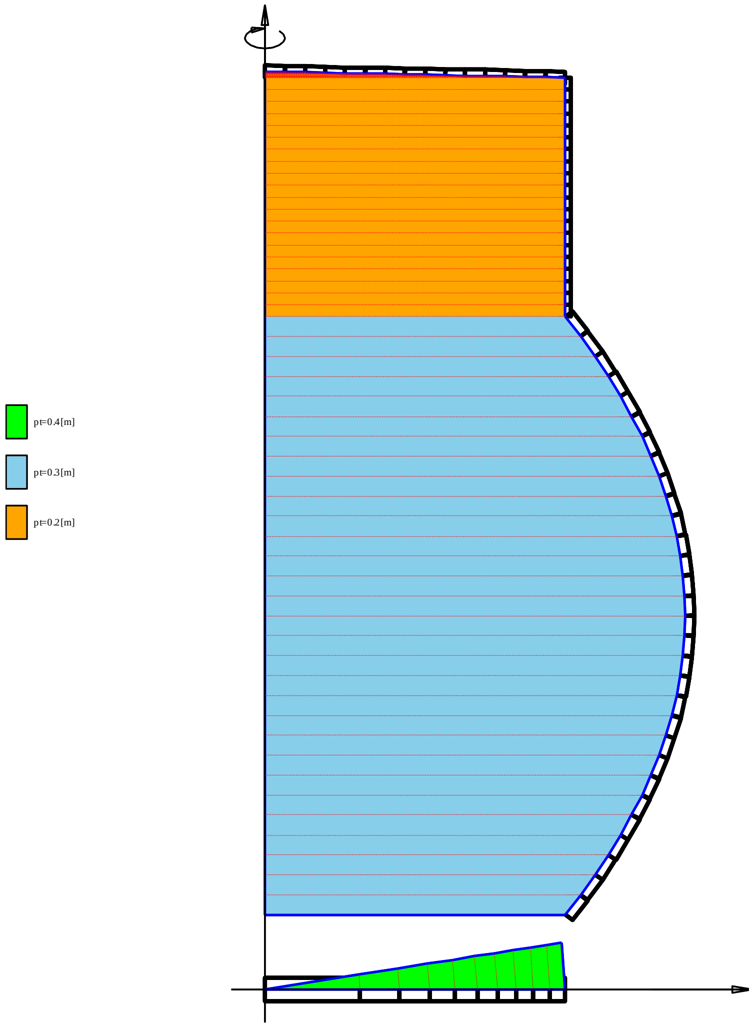


Figure 12.2 Finite element mesh of the container with a sector from the base

3 Creating the project

In this section, the user will learn how to create a project for analyzing a container resting on a layered soil. The project will be processed gradually to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

3.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 12.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of Forms. The first Form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 12.3).

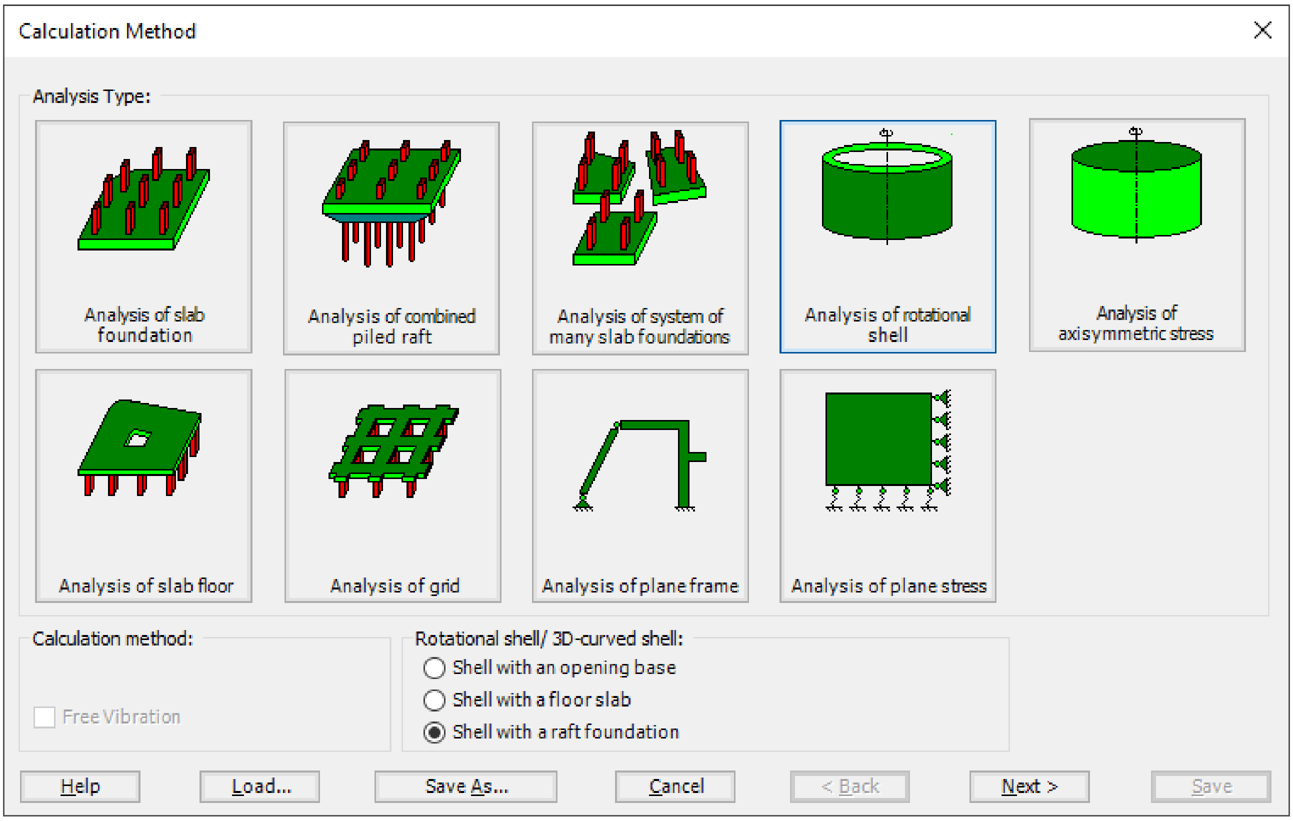


Figure 12.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 12.3, define the analysis type of the problem. As the analysis type is a container resting on layered soil problem, select "Analysis of rotational shell" button, and check "Shell with a raft foundation" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Calculation Methods" Form appears, Figure 12.4.

To define the calculation method:

- Select the calculation method "7-Modulus of Compressibility (Elimination)"
- Click "Next" button to go to the next Form

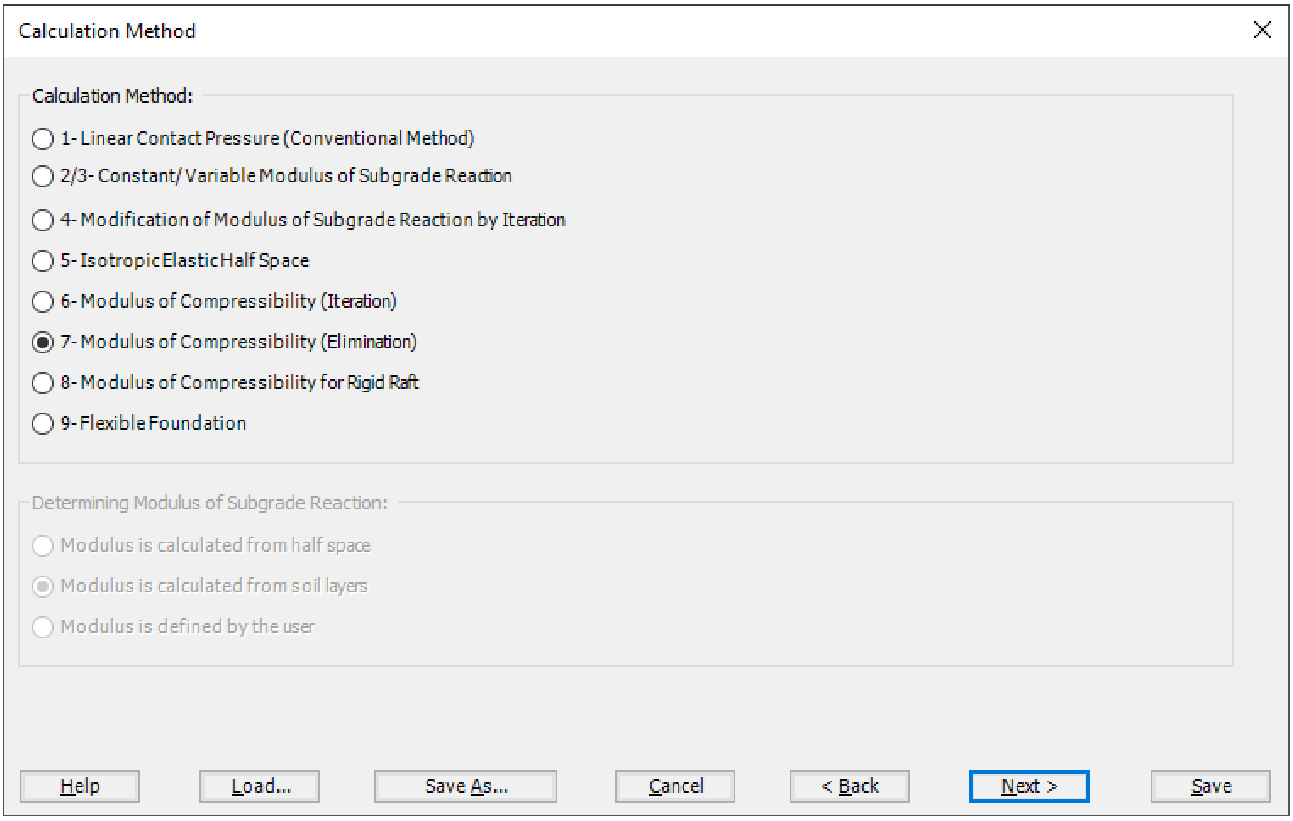


Figure 12.4 "Calculation Methods" Form

The last Form in the wizard is the "Options" Form, Figure 12.5. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Since no option will be considered in the analysis, click the "Save" button.

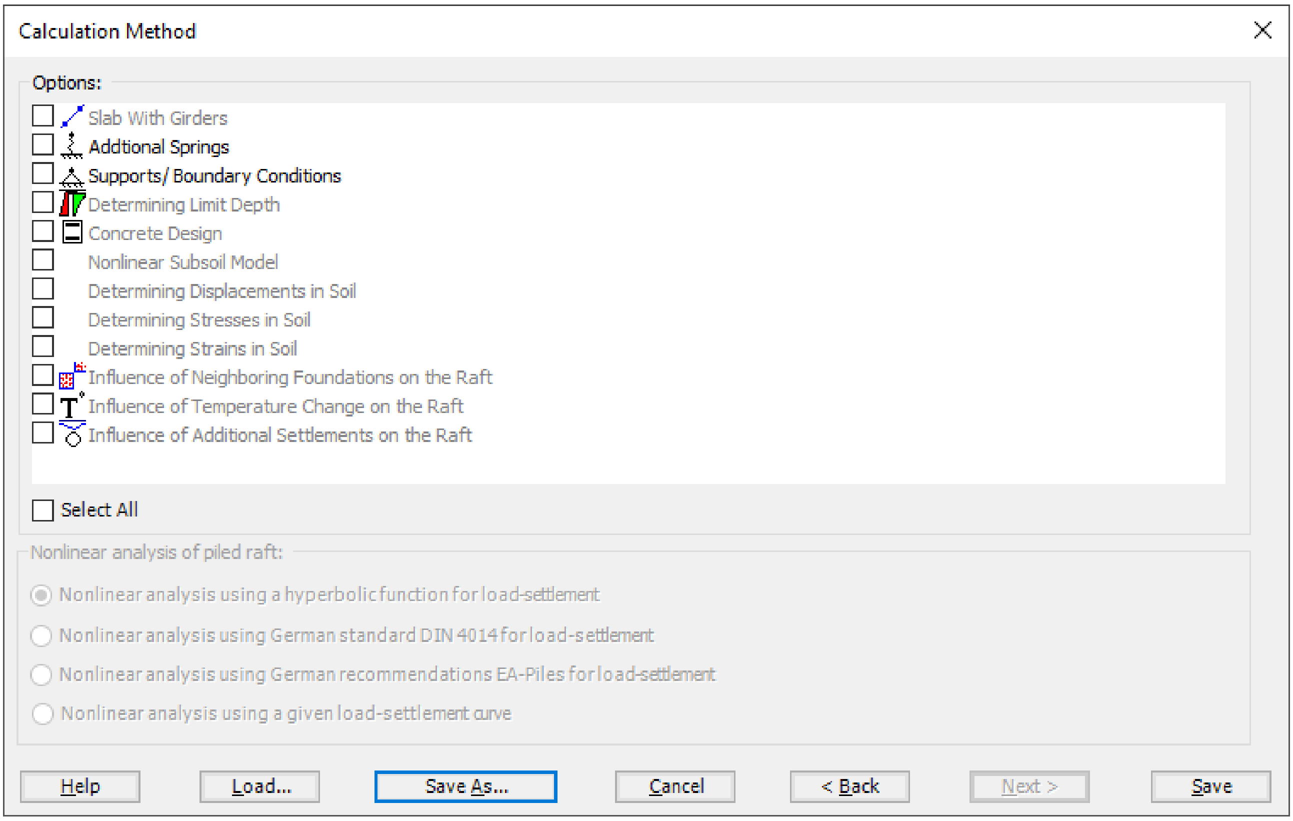


Figure 12.5 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 12.6. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Container on layered soil". *ELPLA* will use automatically this file name in all reading and writing processes.

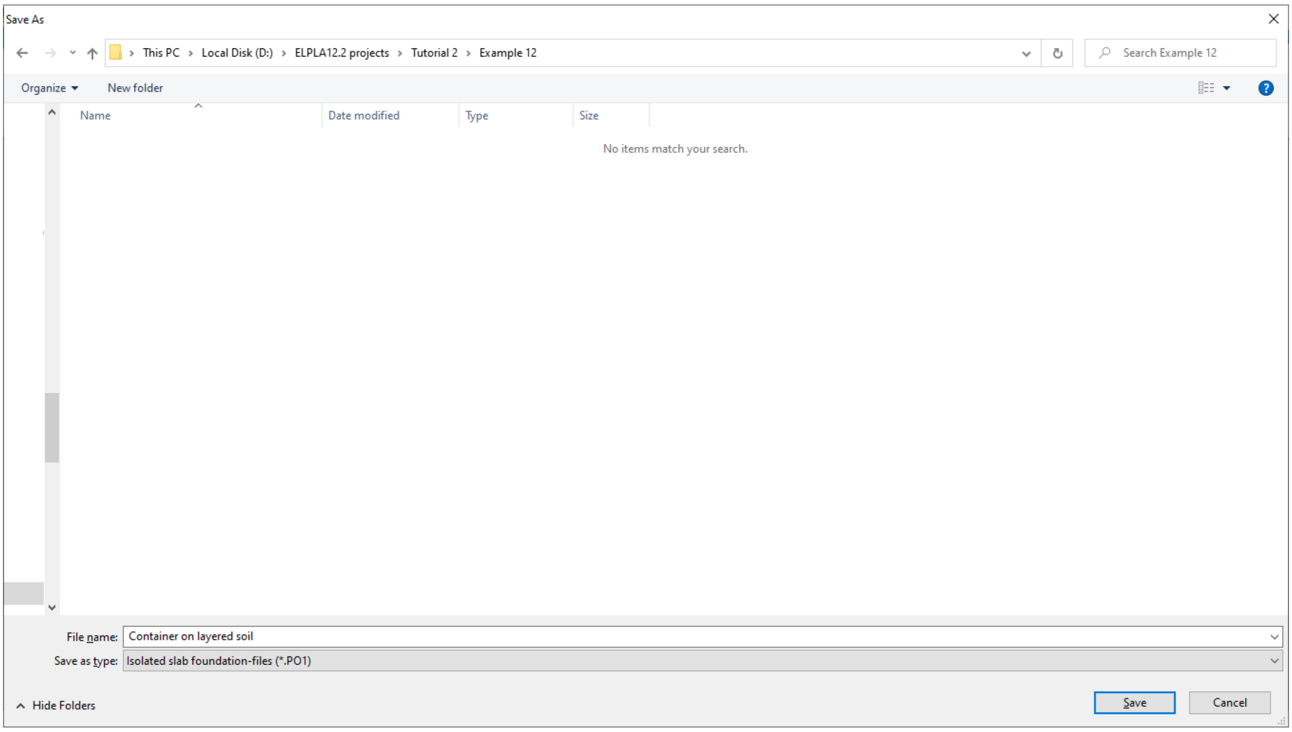


Figure 12.6 "Save as" dialog box

ELPLA will activate the “Data” Tab. In addition, the file name of the current project [Container on layered soil] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

3.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 12.7 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box:
  "Analysis of a container resting on different soil layers"
- Type the date of the project in the "Date" edit box
- Type the word "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

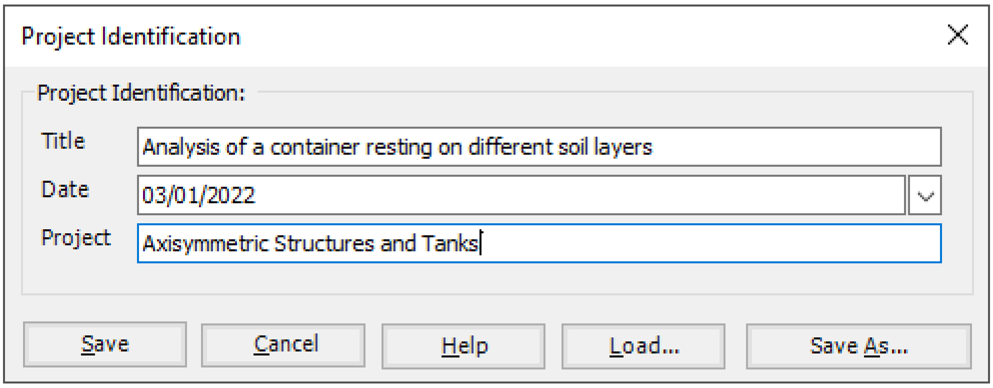


Figure 12.7 "Project Identification" dialog box

3.3 FE-Net data

For the given problem, the container has a base radius of a = 5 [m], and a height of H = 14 [m]. The height of the container is divided into two main segments, the first one is divided into 30 subsegments (30×0.3 [m]), while the second is divided into 20 subsegments (20×0.2 [m]), and the roof segment is divided into 15 subsegments. *ELPLA* has different procedures for defining the FE-Net. To define the FE-Net for this container, choose "FE-Net Data" command from the "Data" Tab. "Analysis of rotational shell" wizard appears as shown in Figure 12.8. This wizard will guide you through the steps required to generate a FE-Net. As shown in Figure 12.8, the first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of shells. These net templates are used to generate standard nets.

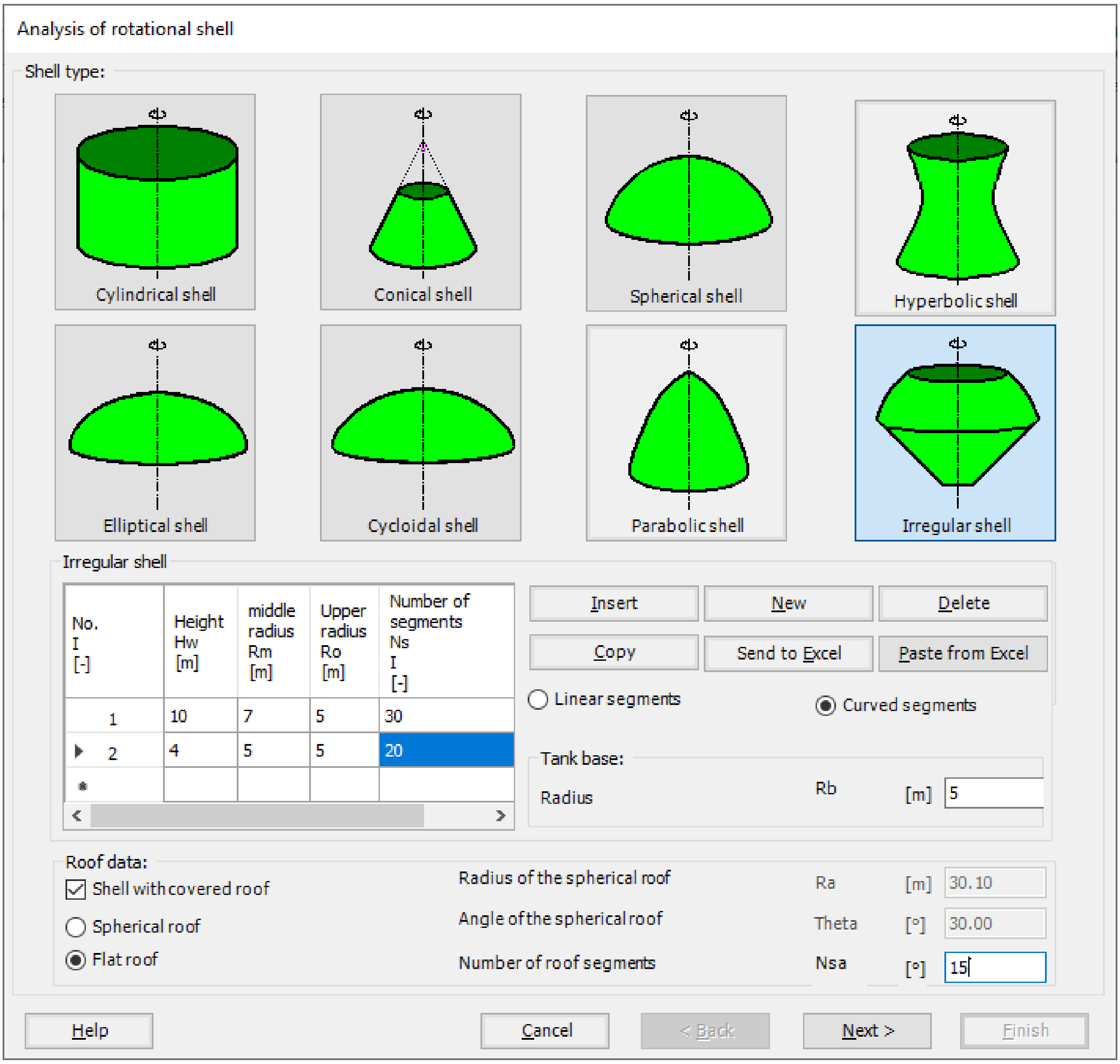


Figure 12.8 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net of the container which has an irregular wall shape:

- In the "Shell type" options choose "Irregular shell" button
  To define the base of the container:
  - Type 5 in the "Tank base radius *Rb*" edit box

  To define the first segment of the height of the container:
  - Choose "Curved segments" option
  - Type 10 in the "height *Hw*" edit box
  - Type 7 in the "middle radius *Rm*" edit box
  - Type 15 in the "Upper radius *Ro*" edit box, as the upper radius is the same as the base radius
  - Type 30 in the "Number of segments *Ns*" edit box

  To define the second segment of the height of the container:
  - Type 4 in the "height *Hw*" edit box
  - Type 5 in the "middle radius *Rm*" edit box
  - Type 5 in the "Upper radius *Ro*" edit box, as the upper radius is the same as the base radius
  - Type 20 in the "Number of segments *Ns*" edit box

  To define the roof of the container:
  - Check the "Shell with covered roof" check box
  - Choose "Flat roof" option
  - Type 15 in the "Number of roof segments" edit box
- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Irregular shell" Form containing the data of the segments appears in Figure 12.9, the user can edit the data of each segment individually or all of them by using "In Table" button, if it is necessary. Click "Next" button to go to the next Form.

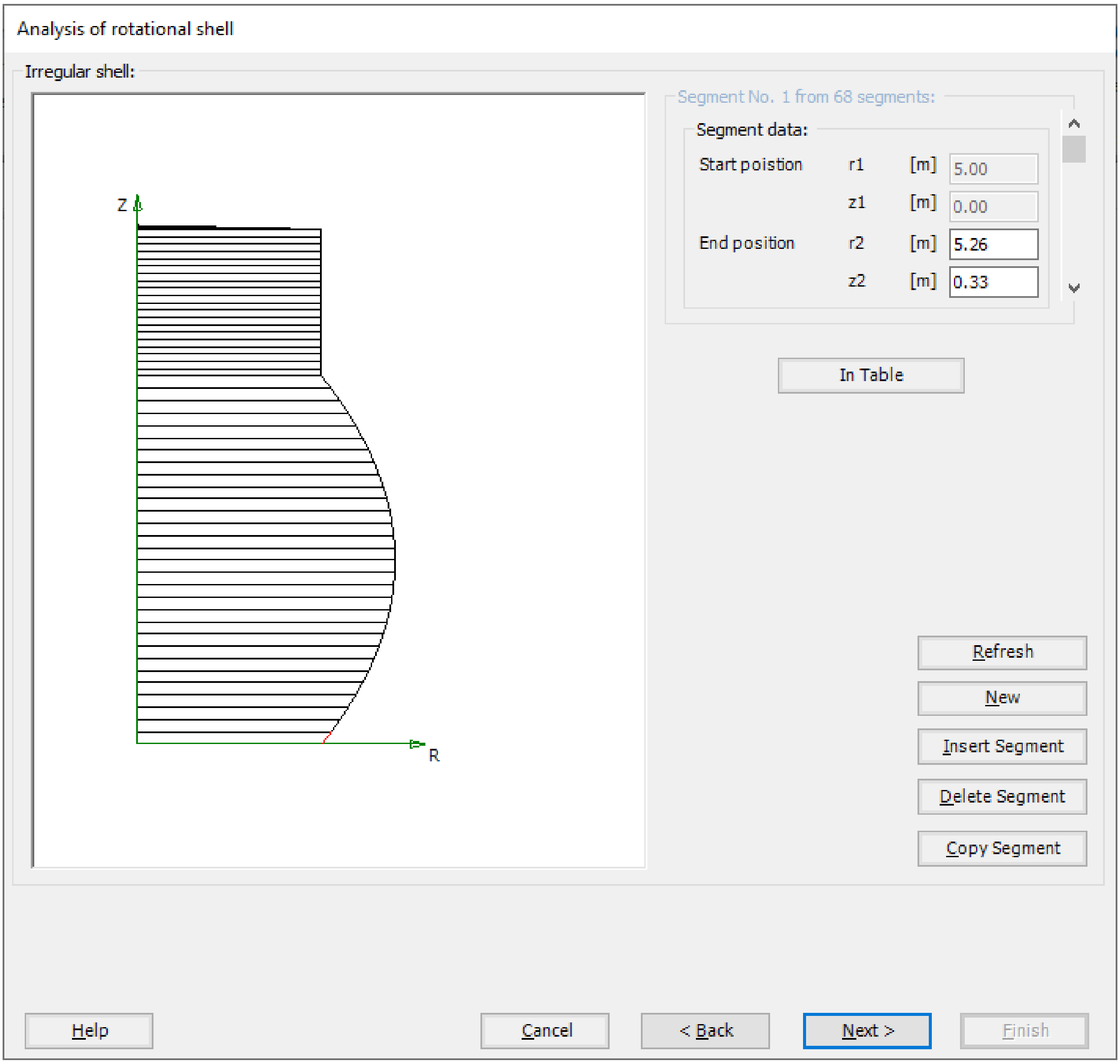


Figure 12.9 "Irregular shell" Form

The next Form of the "Analysis of rotational shell" wizard is the "Net of Base" Form Figure 12.10. In this Form, the default values of the grid intervals appear, to edit the grid spacing in *x*-direction of the base, do the following steps in "Grid in *x*-direction" frame:

- Choose "Constant ring area" check box
- Type 10 in the "No. of grid intervals"
- Click "Finish" button to return to "FE-Net Data" window

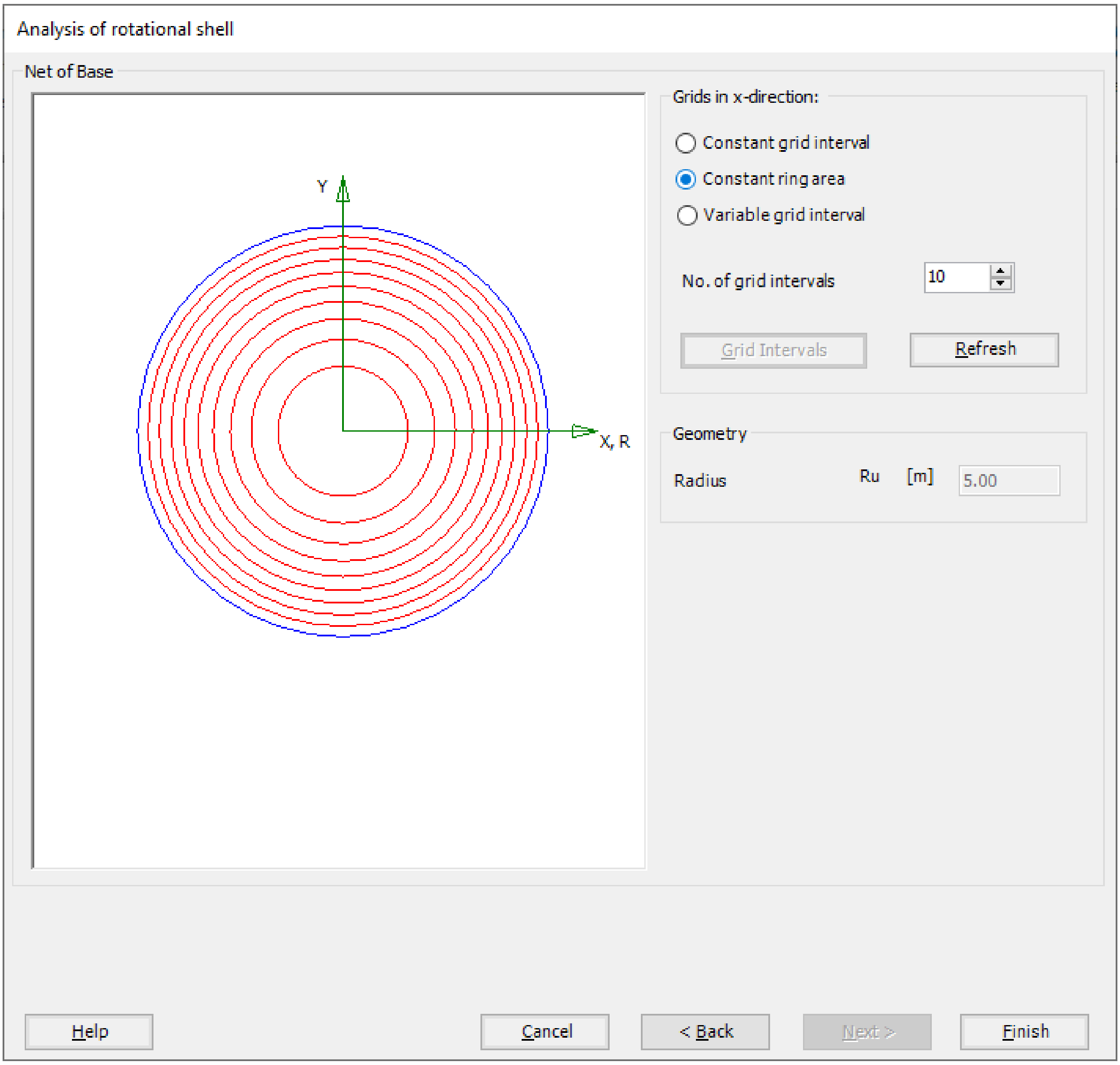


Figure 12.10 "Net of Base" Form

ELPLA will generate the FE-Net of the container with a sector from the base with 10 circular intervals. The following window in Figure 12.11 appears with the generated net.

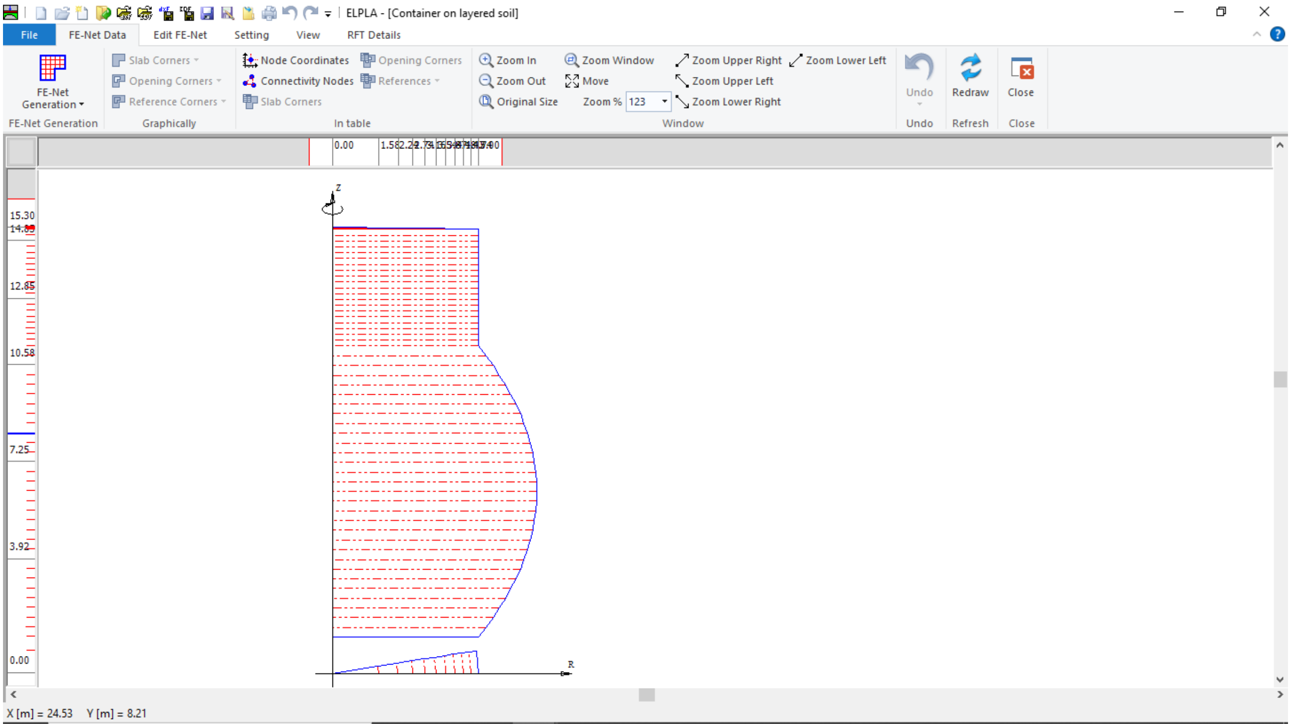


Figure 12.11 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 12.11 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 12.11 to close the "FE-Net" window and return to *ELPLA* main window

3.4 Soil Properties

In *ELPLA*, there are three different soil models with several calculation methods. Therefore, the soil properties for each method are required to be defined according to the used soil model. In the current example, the soil model, which is used in the analysis, is "Layered Soil Model". This model requires that the subsoil have to be defined by boring logs. In the example, the boring log has multi-layers with different soil materials. The geotechnical data for each layer is unit weight of the soil γ_s and modulus of Elasticity for loading E and reloading W and *Poisson's* ratio vs.

To define the soil properties, choose "Soil Properties" command from “Data” Tab. The following "Soil Properties" Window in Figure 12.12 appears with a default-boring log.

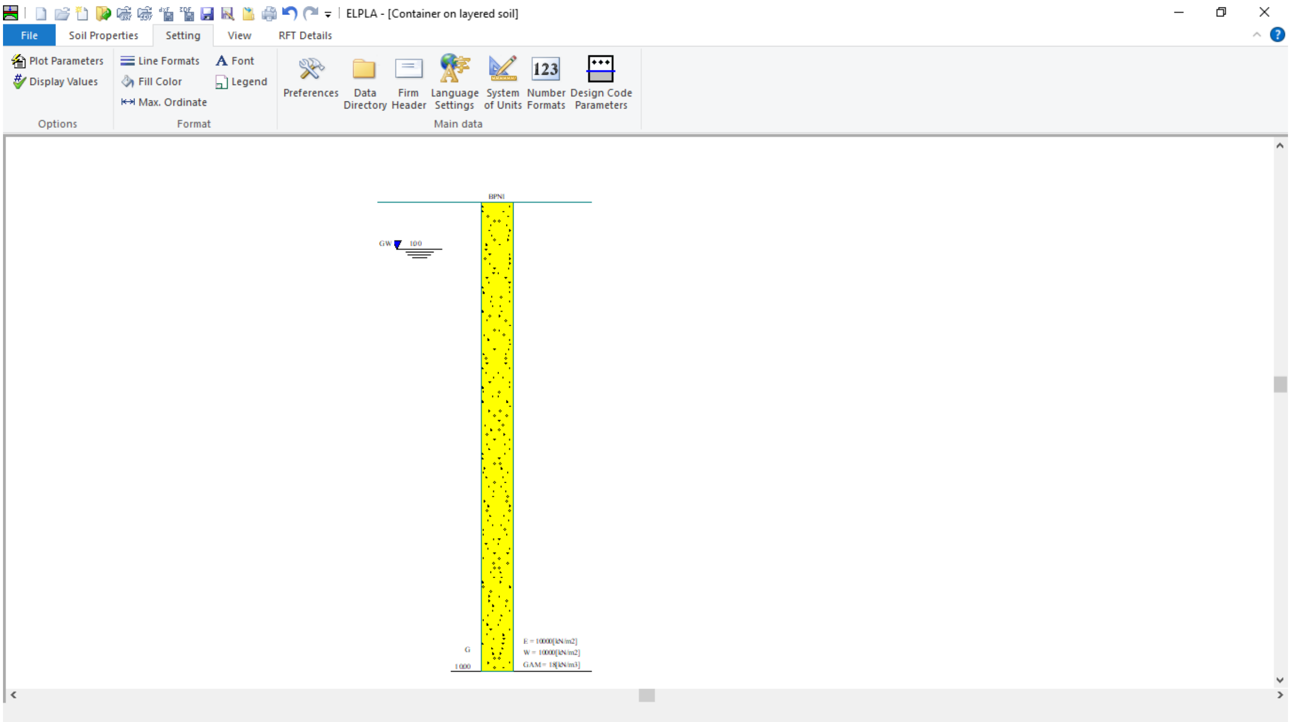


Figure 12.12 "Soil Properties" window with a default-boring log

In Figure 12.12, soil properties are defined through the "Data" menu, which contains the following two commands:

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

To enter the soil properties for the boring log of the current example

- Choose "Soil Data" command from "Data" menu in the window of Figure 12.12
  The following dialog box in Figure 12.13 with default-boring log data appears, Define the data of the boring log as the following

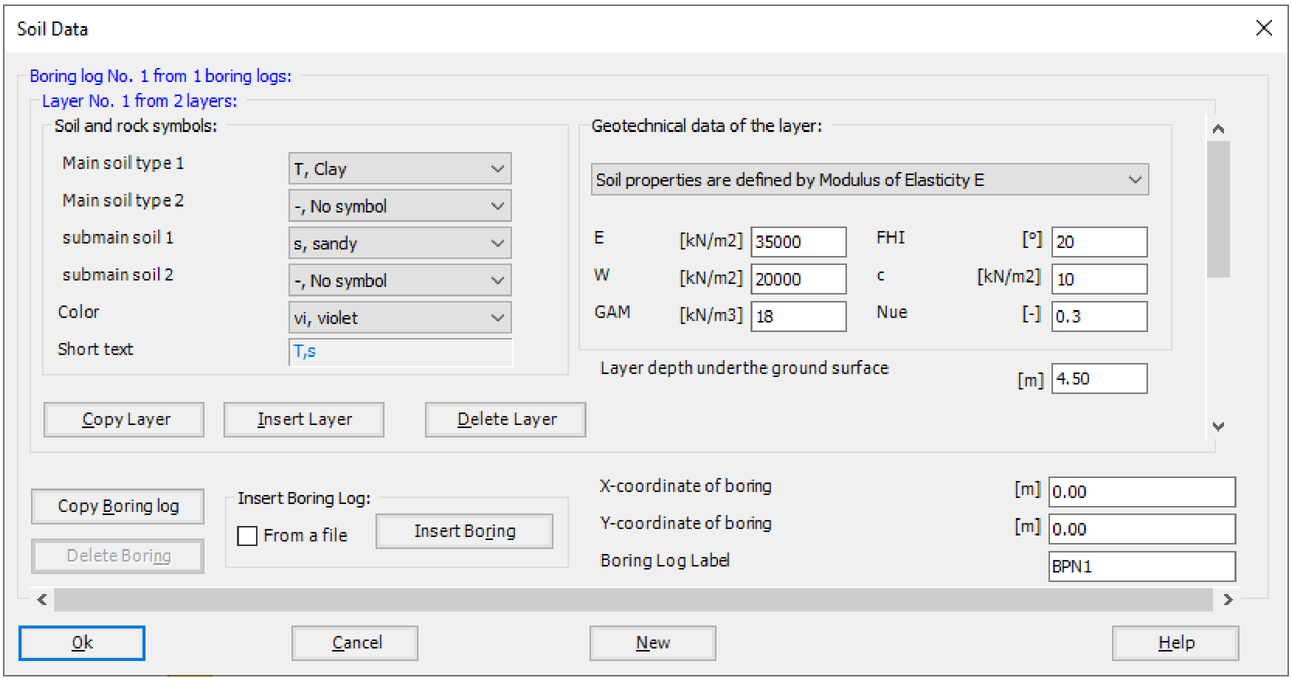


Figure 12.13 "Soil data" dialog box

In the "Geotechnical data of the layer" dialog group box in Figure 12.13, Soil properties are defined by Modulus of elasticity, define the geotechnical data of the first soil layer of the boring log as follows:

E	= 35000	[kN/m^2]
W	= 105000	[kN/m^2]
Gam	= 18	[kN/m^3]
C	= 10	[kN/m^2]
φ	= 20	[°]

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

To define the soil type and color for the first layer, select "T, Clay" as the soil type in "Main soil type 1" combo box, and "S, Sand" as the submain soil type in "Submain soil 1" combo box in "Soil and rock symbols" dialog group box in Figure 12.13. The color of the layer according to the German Standard DIN 4023 will be automatically created. The user can change this color.

To enter the second layer of the boring log

- Click "Copy Layer" button in Figure 12.13. A layer with the same properties of the first layer will be copied
- Use the vertical scrollbar to move to the second soil layer. Layer No. will be typed automatically at the upper-left corner of the main dialog box of soil layers as a head title
- In "Geotechnical data of the layer" dialog group box in Figure 12.13, define the geotechnical data of the sand with gravel layer as follows:
    - *E* = 80000 [kN/m^2]
    - *W* = 240000 [kN/m^2]
    - Gam = 18 [kN/m^3]
    - *C* = 0 [kN/m^2]
    - φ = 35 [º]
- Change the value of the layer depth under the ground surface from 4.5 [m] to 19.8 [m]
- From "Soil and rock symbols" dialog group box, choose "S, Sand" and "G, Gravel" in "Main soil type 1" and "Main soil type 2" combo boxes, as the main soil type of the second layer is sand with gravel

Note that: as the analysis is nonlinear, the unit weight of the soil, cohesion and angle of internal friction play no roles in the analysis.

Now all data and parameters for the boring log (Figure 12.14) have been entered.

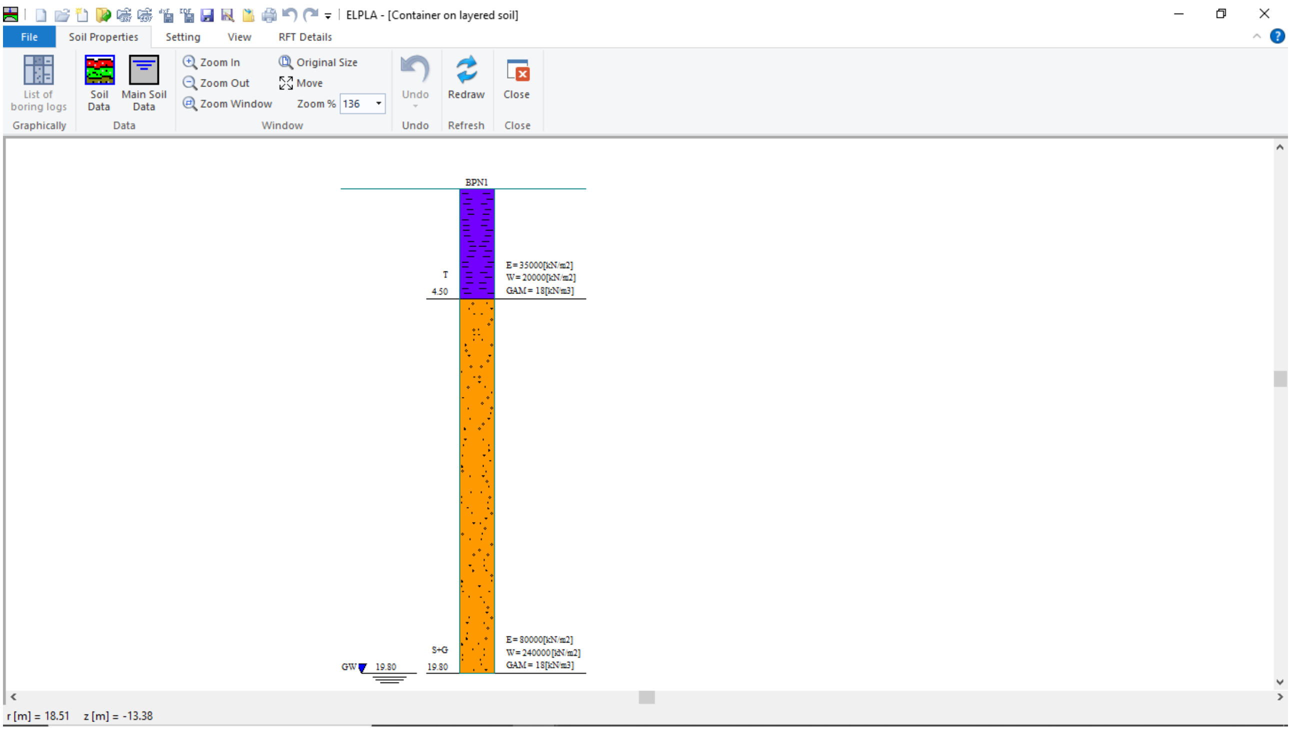


Figure 12.14 boring log

To enter the main soil data for all layers, choose "Main Soil Data" command from "Data" menu in Figure 12.12. The following dialog box in Figure 12.15 appears with default main soil data. The main soil data for the current example, which are required to be defined, are the settlement reduction factor α [-] and the groundwater depth under the ground surface G_w [m]. Any other data corresponding to main soil data are not required in this example. Therefore, the user can take these data from the default soil properties.

In the dialog box of Figure 12.15, enter the settlement reduction factor α [-] and the groundwater depth under the ground surface G_w [m]. Then click "OK" button.

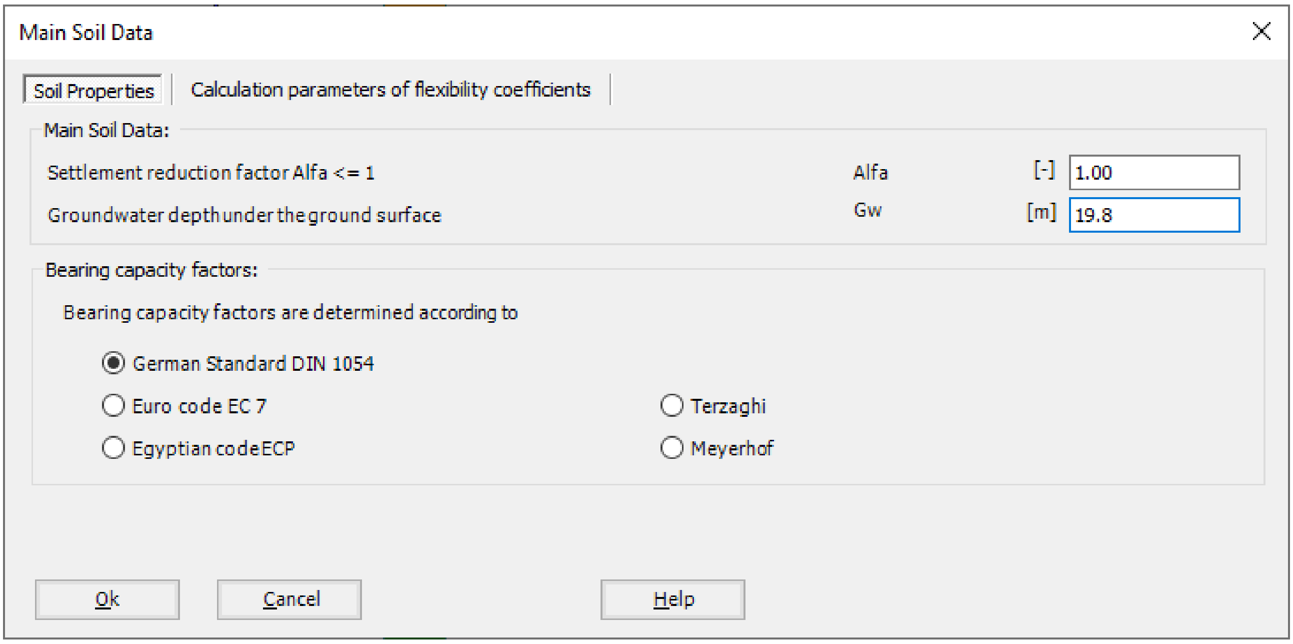


Figure 12.15 "Main Soil Data" dialog box

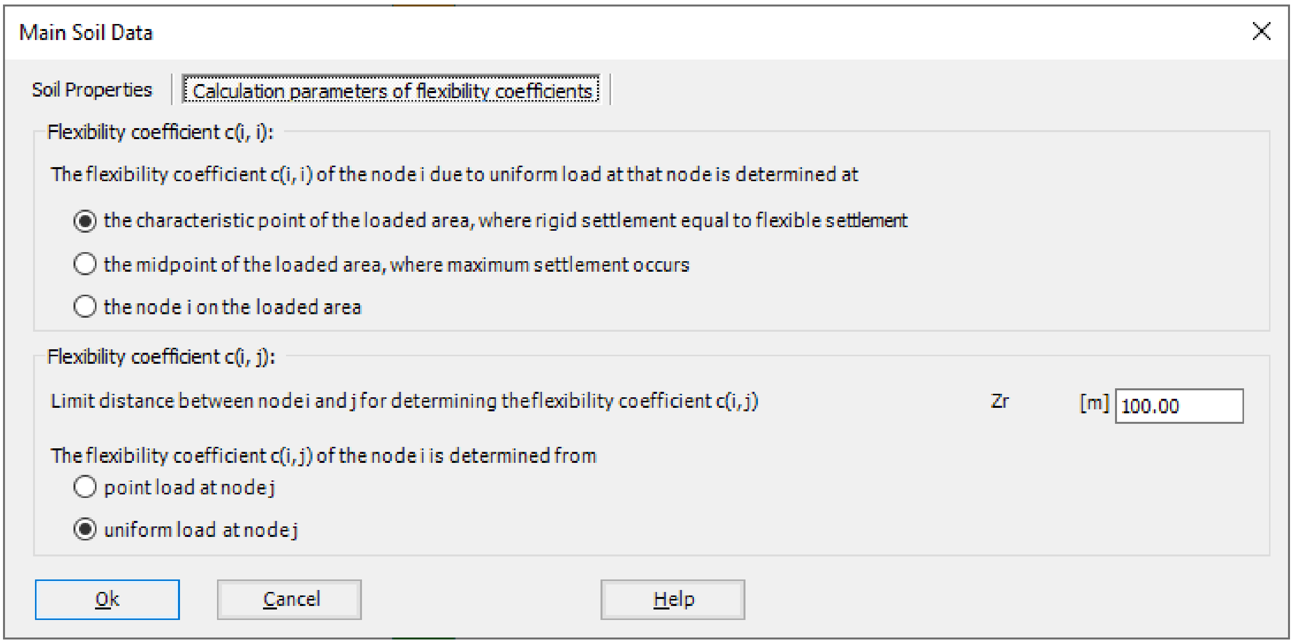


Figure 12.16 "Calculation parameters of flexibility coefficients" dialog box

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

- Choose "Save" command from "File" menu in Figure 12.12 to save the data of boring log
- Choose "Close" command from "File" menu in Figure 12.12 to close "Soil properties window and return to *ELPLA* main window.

3.5 Shell properties

To define the container properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 12.17 appears with default shell properties. The data of shell properties for the current example, which is required to be defined, are element groups, group regions, the unit weight of the container, and the filled material properties. Any other data corresponding to shell properties in the program menus are not required for this example.

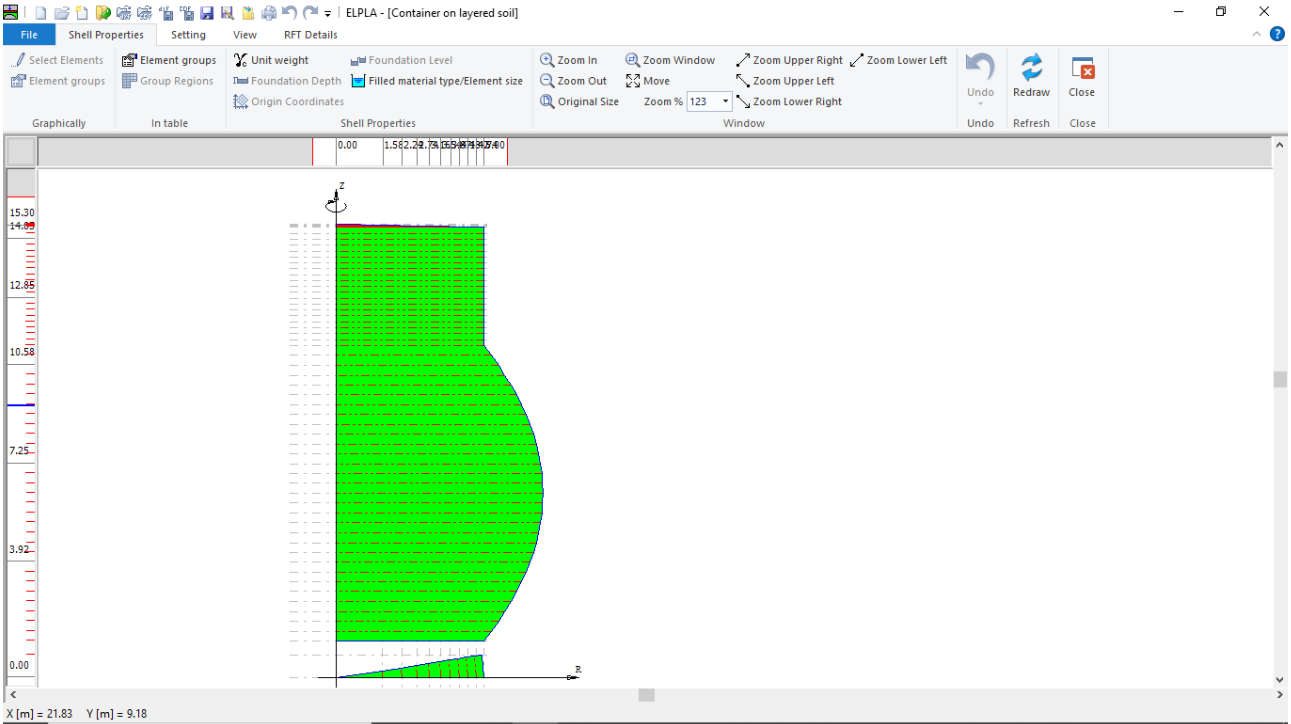


Figure 12.17 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 12.18 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness for the container base, wall and roof as they differ in thickness. Then click "OK" button to go to the next step.

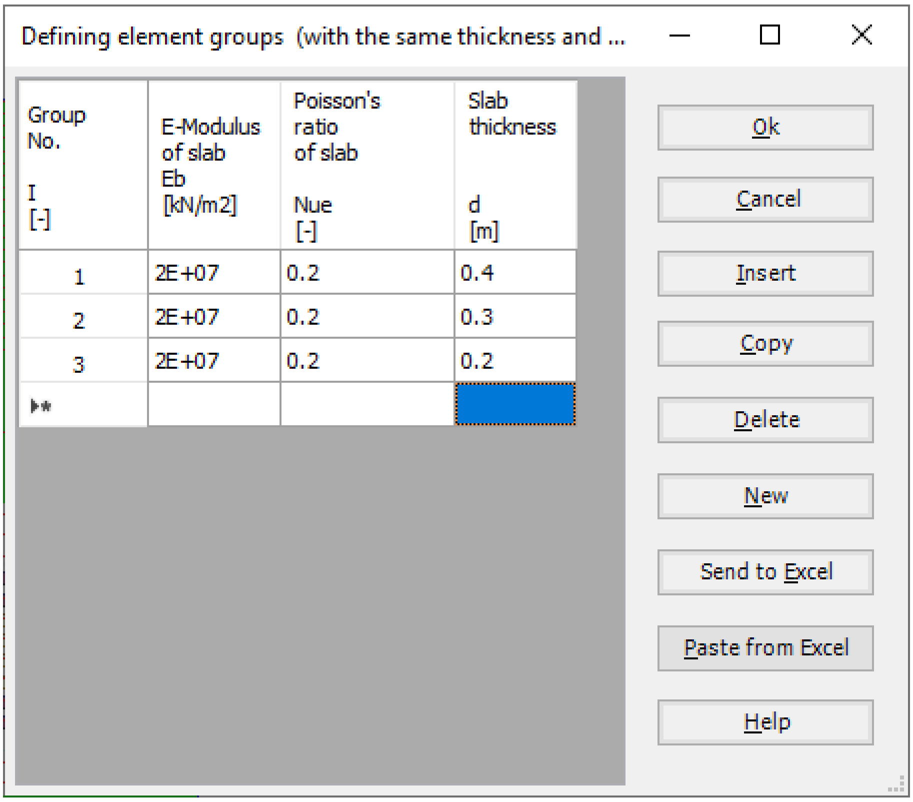


Figure 12.18 "Defining element groups" list box

Defining the slab thickness for materials on the net may be carried out either graphically or numerically (in a table). In the current example, the user will define the slab thickness on the net graphically.

To define the slab thickness for the container roof

- Choose "Select Elements" command from "Graphically" menu in the window of Figure 12.17.
- When "Select Elements" command is chosen, the cursor will change from an arrow to a cross hair. A group of elements can be selected by holding the left mouse button down at the corner of the region. Then, drag the mouse until a rectangle encompasses the required group of elements. When the left mouse button is released, all elements in the rectangle are selected
- Select the elements that include the container roof as Figure 12.19
- Choose "Elements Groups" command from "Graphically" menu in the window of Figure 12.17, "Group Regions" dialog box Figure 12.20 appears
- Define the "Group No." of the roof elements as type "3"
- Repeat the previous steps to define the "Group No." of the container wall as type "2", while "Group No." of the wall elements will be as type "1", where type "1" is the default "Group No.", then click "OK" button

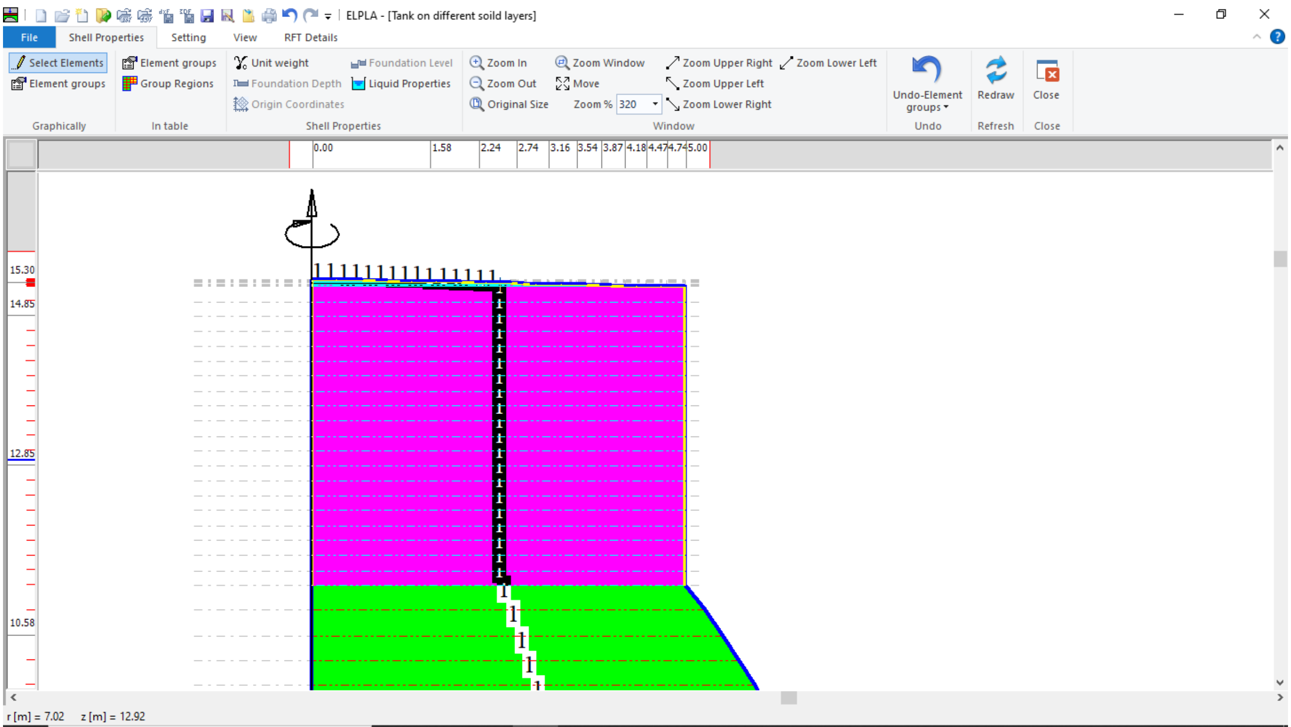


Figure 12.19 Selecting the elements that include the container roof

Figure 12.20 "Group Regions" dialog box

To enter the unit weight of the container, choose "Unit weight" command from "Shell Properties" menu in the window of Figure 12.17. The following dialog box in Figure 12.21 with a default unit weight of 25 [kN/m^3] appears. Click "OK" button.

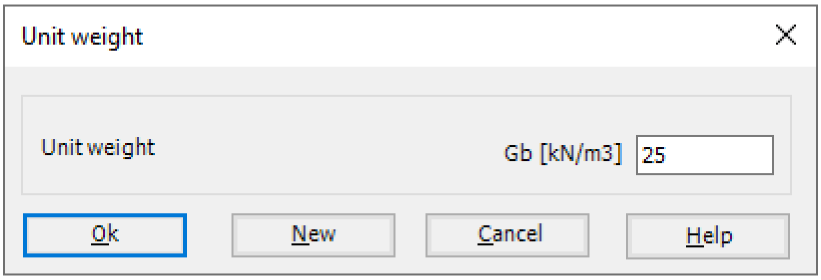


Figure 12.21 "Unit weight" dialog box

To define the filled material properties of the container, choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 12.17. The following form in Figure 12.22 appears.

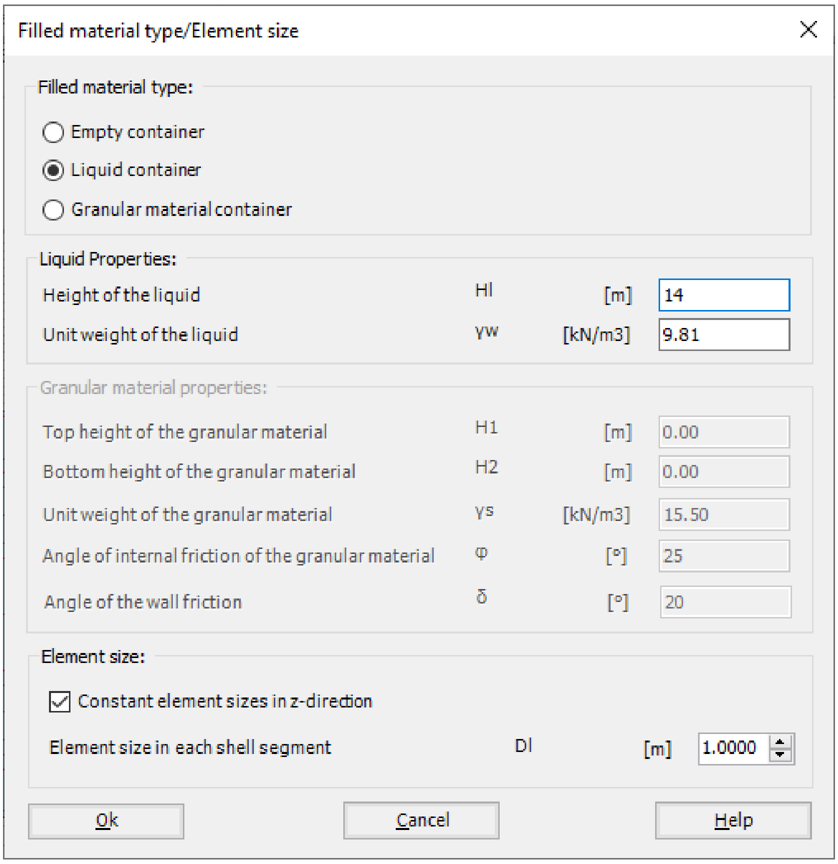


Figure 12.22 "Filled material type/Element size" Form

To define the filled material properties of the container:

- Select "Liquid container" option
- Type 14 in the "Height of the liquid" edit box
- Type 9.81 in the "Unit weight of the liquid" edit box

To define the element size of the container:

- Check the "Constant element sizes in z-direction" check box
- Type 1 in the "Element size in each shell segment" edit box. The element size is chosen to be 1 [m] larger than the segment size in order to ignore further subdivision of the segments into smaller elements. In some cases, it is necessary to divide the segment into smaller elements in order to make the analysis more precise. Nevertheless, the final results of the internal forces appear only at nodes of segments
- Click "OK" button

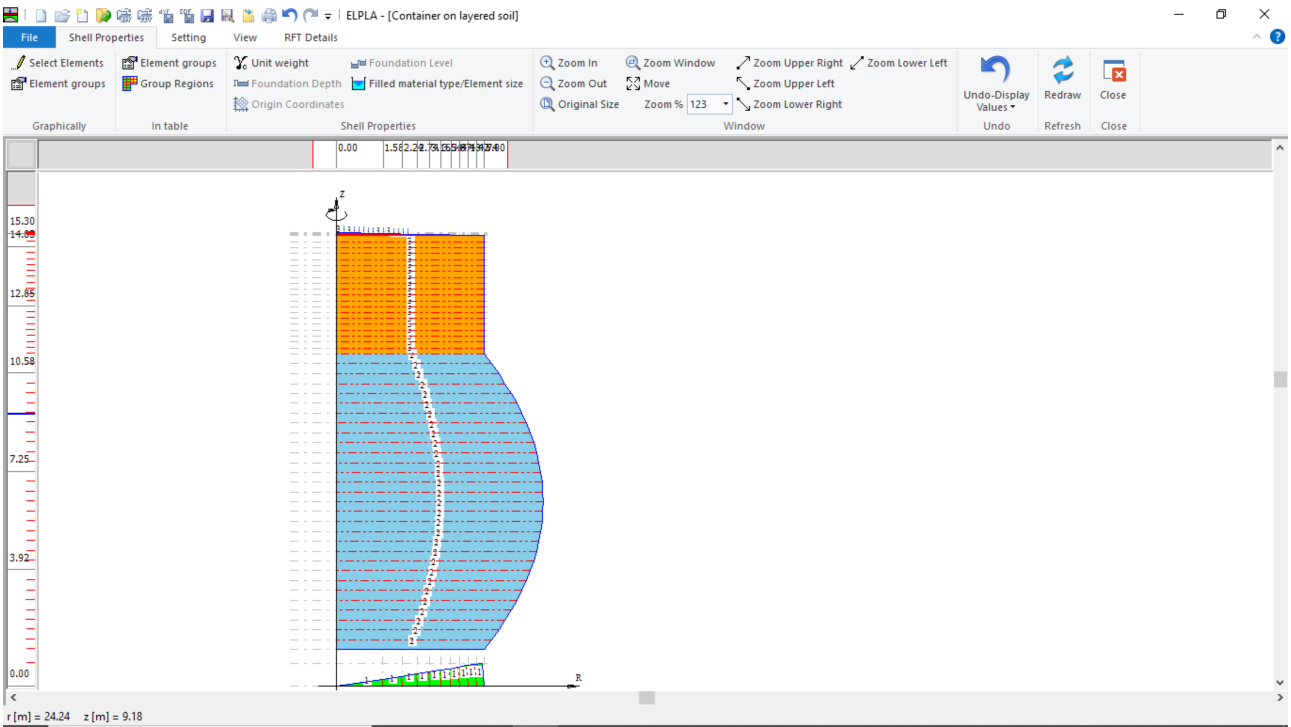


Figure 12.23 "Shell Properties" window after defining the container properties

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

- Choose "Save" command from "File" menu in Figure 12.23 to save the shell properties
- Choose "Close" command from "File" menu in Figure 12.23 to close the "Shell properties" window and return to *ELPLA* main window

3.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 12.24 appears.

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 12.24. In this example, there is not applied load, as the load has been already defined by the unit weight of the container material, while the hydrostatic pressure on the container is defined by the unit weight of water.

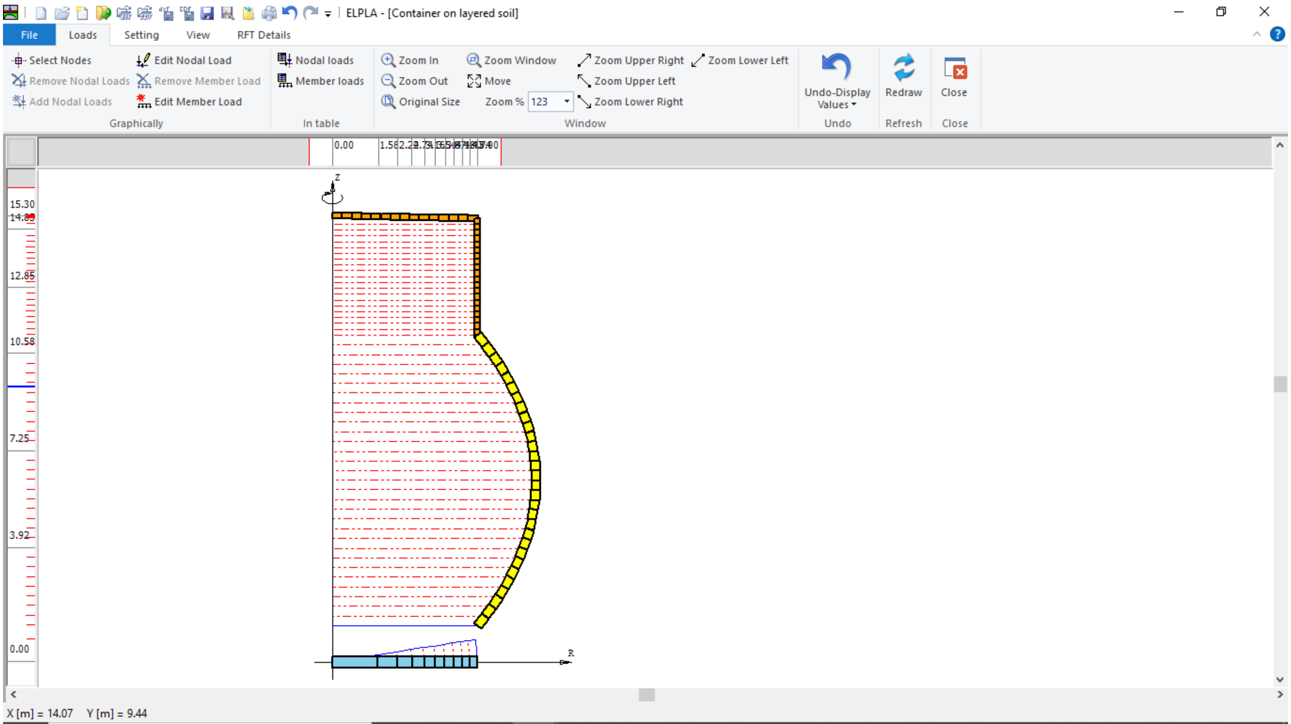


Figure 12.24 "Loads" Window

Do the following two steps:

- Choose "Save" command from "File" menu in Figure 12.24 to save the load data
- Choose "Close" command from "File" menu in Figure 12.24 to close the "Loads" Window and return to *ELPLA* main window

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

4 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 12.25.

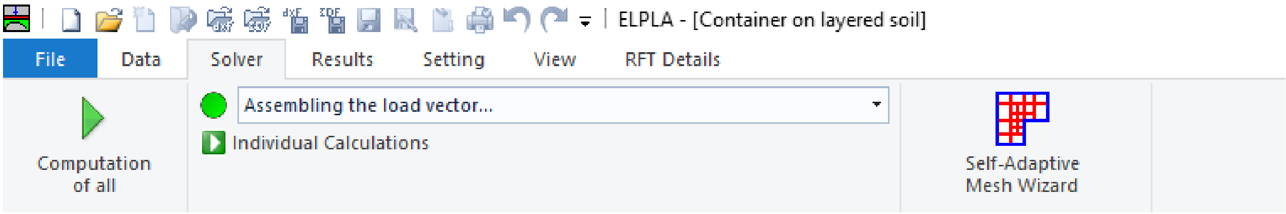


Figure 12.25 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

- Assembling the load vector
- Determining flexibility coefficients of the soil
- Assembling the soil stiffness matrix
- Assembling the slab stiffness matrix
- Solving the system of linear equations (unsymmetrical matrix)
- Determining deformation, internal forces, contact pressures

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

Carrying out all computations

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab window.
- The progress of all computations according to the defined method will be carried out automatically with displaying information through menus and messages.

Analysis progress

Analysis progress menu in Figure 12.26 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

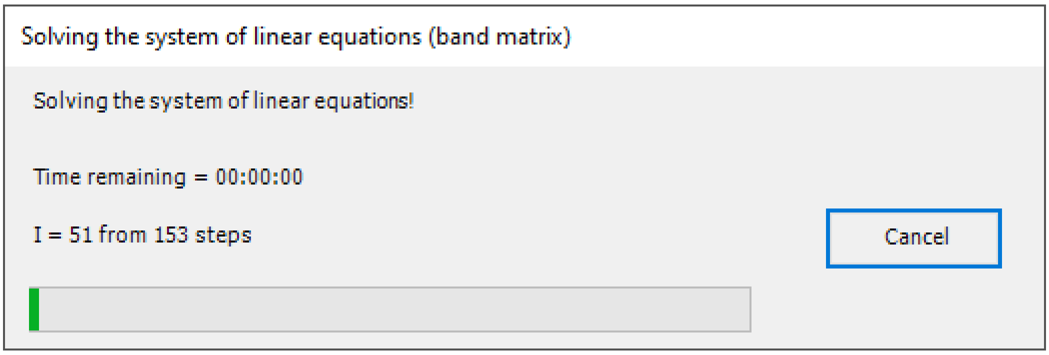


Figure 12.26 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 12.27. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

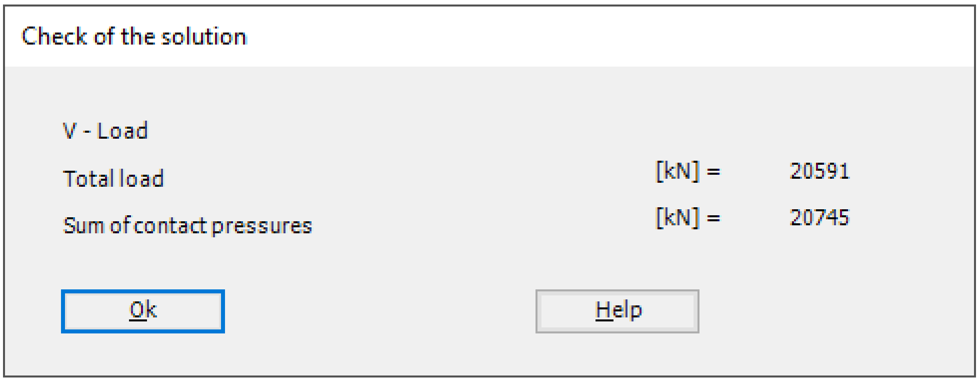


Figure 12.27 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

5 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab. To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 12.28).

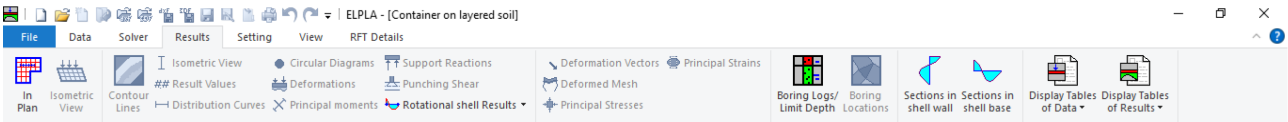


Figure 12.28 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Boring logs and limit depth
- Sections in the shell wall
- Sections in the shell base
- Display tables of data
- Display tables of results

To view sections in shell base

- Choose "Sections in shell base" command from "Section" menu. The following option box in Figure 12.29 appears
- In the "Sections in shell base" option box, select "Base contact pressures q" as an example for the results to be displayed
- Click "OK" button

The base contact pressures are now displayed as shown in Figure 12.30.

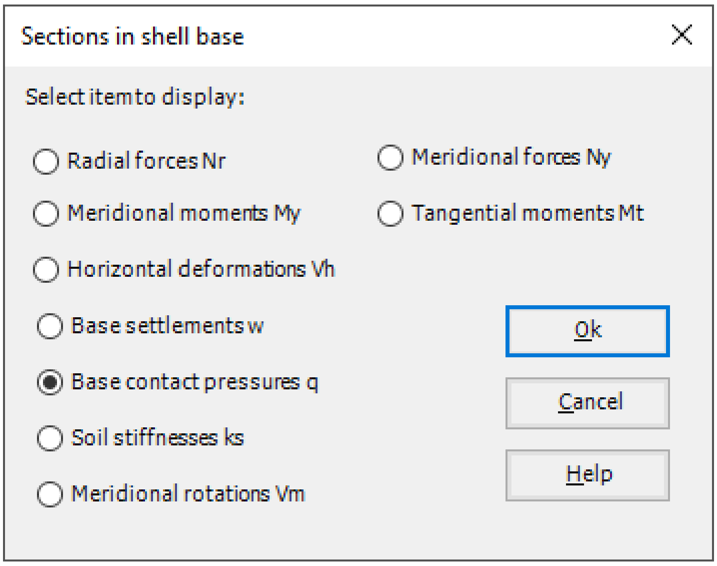


Figure 12.29 "Sections in shell base" option box

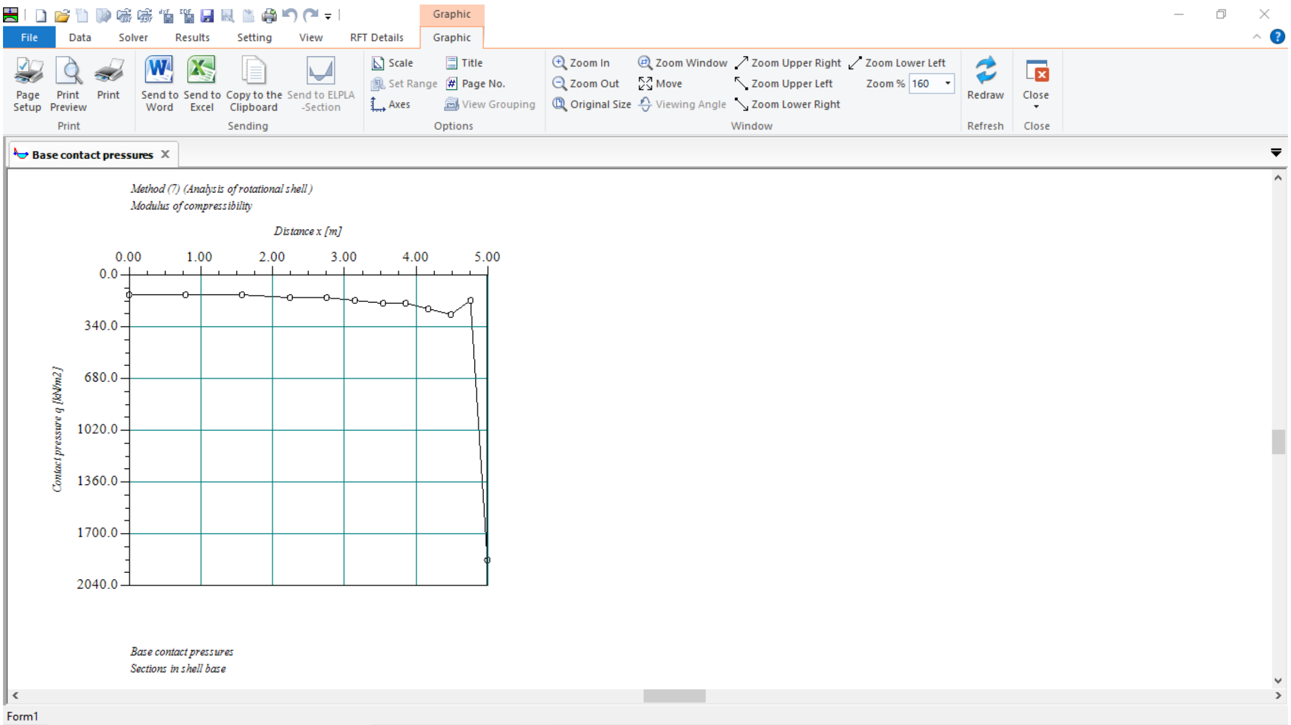


Figure 12.30 Base contact pressures in shell base

To view the base settlements

- Choose "Sections in shell base" command from "Section" menu. The following option box in Figure 12.31 appears
- In the "Sections in shell base" option box, select "Base settlements *w*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 12.32.

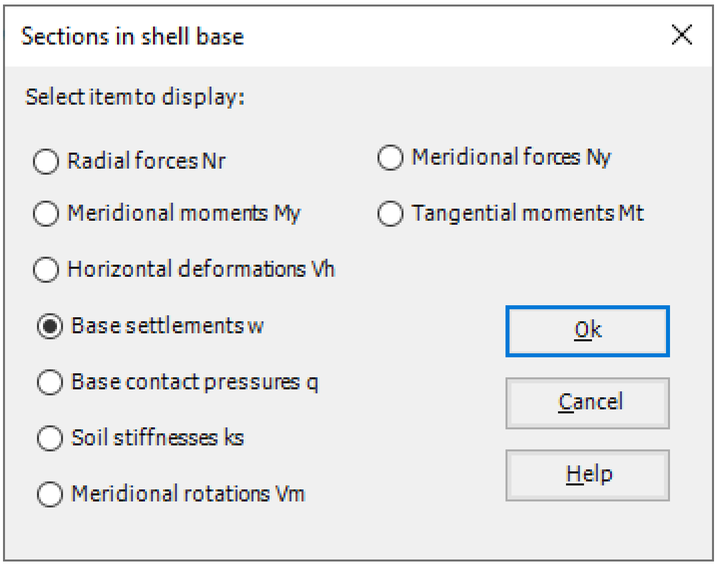


Figure 12.31 "Sections in shell base" option box

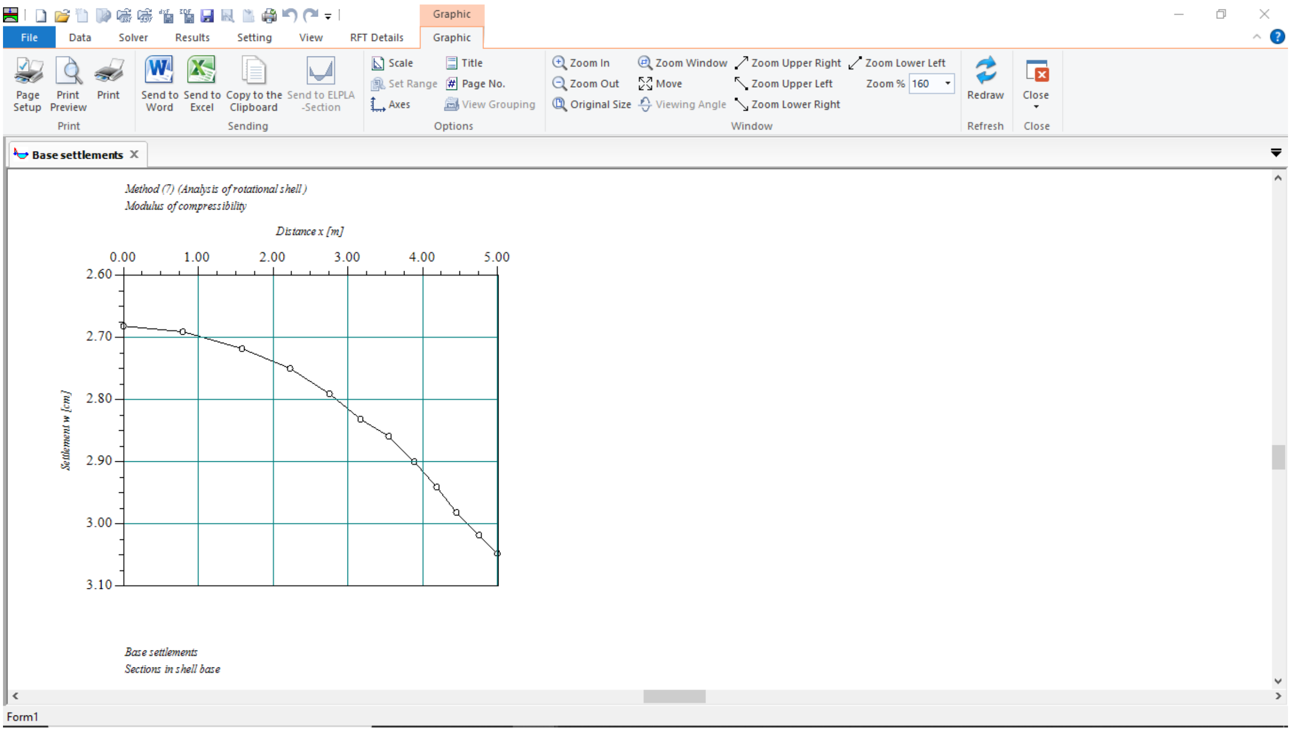


Figure 12.32 Base settlements

To view element groups of the container

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 12.33 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

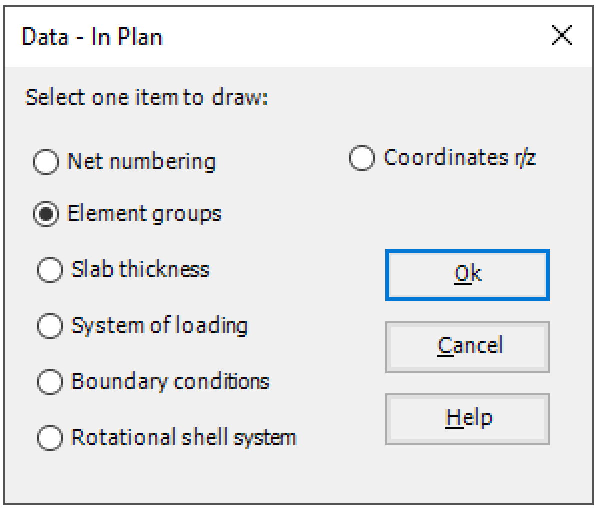


Figure 12.33 "Data – In Plan" option box

To view the internal forces on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 12.34 appears
- In this check group box, the user can choose any data to be viewed
- Click "OK" button

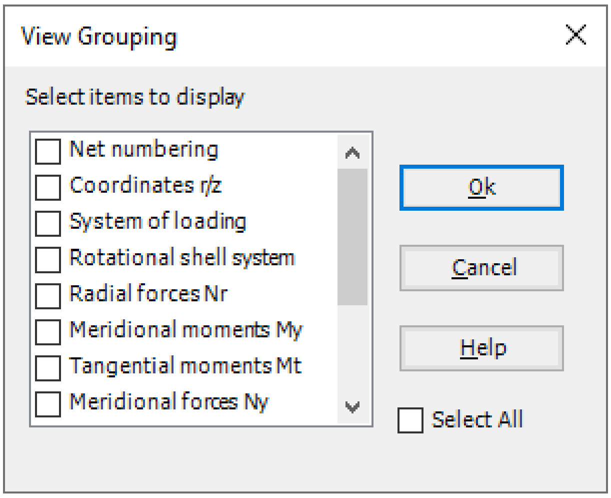


Figure 12.34 "View Grouping" check group box

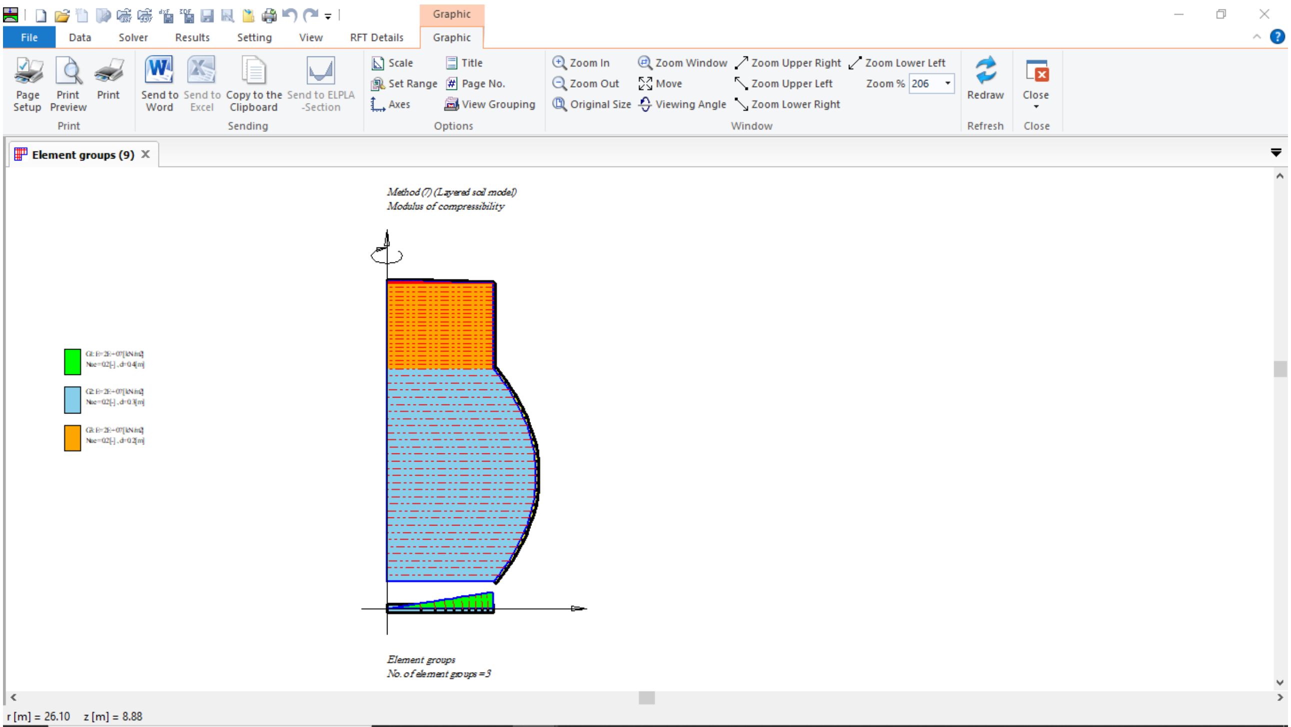


Figure 12.35 Element groups of the container

Example 13

Analysis of a circular silo for storing bulk materials

Contents Page

1 Description of the problem

An example of a circular concrete silo is selected to illustrate some features of *ELPLA* for analyzing circular silos for storing bulk materials.

2 Silo geometry and properties

A circular concrete silo having a conical hopper at the bottom part and a conical roof at the upper part is considered. The main height of the silo is 8 [m] and its diameter is 4 [m]. The stored material is cement of a unit weight of 15.5 [kN/m^3]. The angle of internal friction of cement is 25 [°] and the angle of wall friction is 25 [°]. The thickness of the roof and the wall is 0.28 [m], while the thickness of the hopper is 0.25 [m]. The conical hopper bottom slope is 45 [°], an opening at the bottom with diameter 0.5 [m] and hopper bottom height is 3 [m]. Table 13.1 shows the geometry of the silo with dimensions and support, while the silo shell material is listed in Table 13.1.

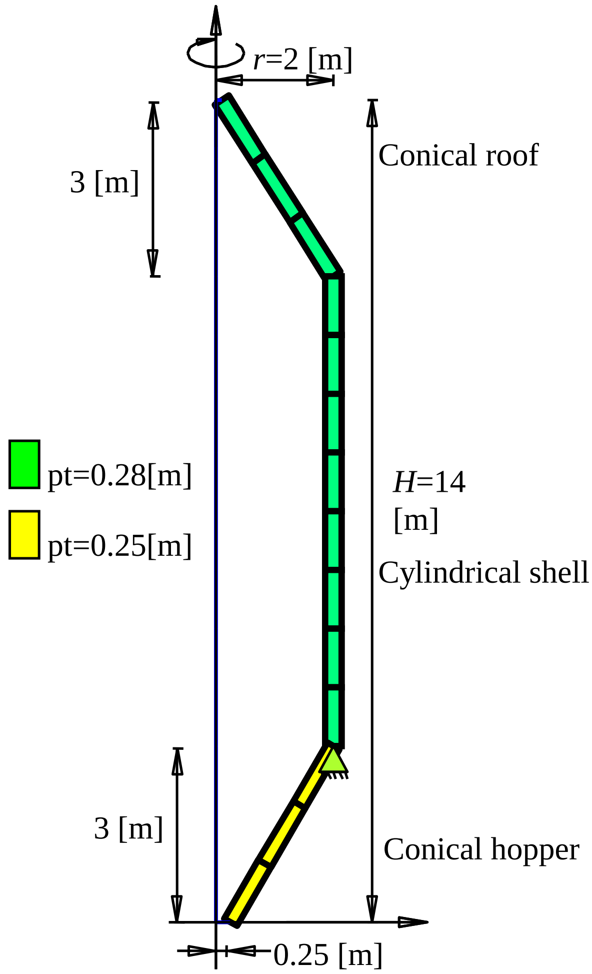


Figure 13.1 Geometry of the silo with dimensions and support

Table 13.1 Silo shell material

Modulus of Elasticity of the shell material	E_c	= 2.486×10^7	[kN/m^2]
Poisson's ratio of the shell material	ν_c	= 0.2	[-]
Unit weight of the shell material	γ_c	= 23.563	[kN/m^3]

3 Pressure on the silo wall

According to *Janssen's* silo theory, the horizontal pressure P_h [kN/m^2] on the silo wall at a depth h [m] below the free surface of the stored material is given by:

$$P_h = \frac{\gamma_s\ R}{\mu}\left[1 - \mathrm{Exp}\left(\frac{-\mu\ k\ h}{R}\right)\right]$$

in which k is the ratio of horizontal to vertical pressures, usually assumed equal to *Rankine's* coefficient of active earth pressure

$$k = \frac{1 - \sin\varphi}{1 + \sin\varphi}$$

h	Depth from the material top to the calculation section, [m]
k	Wall pressure coefficient, [-]
φ	Angel of internal friction of the stored material, [°]
γ_s	Unit weight of the stored material, [kN/m^3]
$R=A/U$	Hydraulic radius of the net horizontal cross section, [m]
$\mu=\tan\delta$	Friction coefficient between the silo wall and the stored material
δ	Angle of the wall friction, [°]
$A=\pi D/4$	Cross-sectional area of the silo, [m^2]
$U=\pi D$	Parameter of the silo, [m]
D	Diameter of the silo, [m]

Using the above relations and equations, the lateral pressure P_h on the main silo wall various depth is determined and presented in Table 13.2.

Table 13.2 Lateral pressure P_h on the main silo wall

Height from the top h [m]	Lateral pressure on the silo wall P_h [kN/m2]
0.5	3.00
1.5	8.22
2.5	12.53
3.5	16.10
4.5	19.06
5.5	21.50
6.5	23.53
7.5	25.20

4 Numerical Analysis

The wall of the silo is divided into three parts:
1. The roof part where no lateral pressure is applied on it
2. The main silo part where the lateral pressure p_h is applied.
3. The hopper part where no lateral pressure is applied on it

In the analysis, these three parts are divided into 14 segments; each segment is 1.0 [m]. Then these segments are divided into a number of elements, each element is 0.2 [m]. Segment dimensions and number of segments are shown in Figure 13.2.

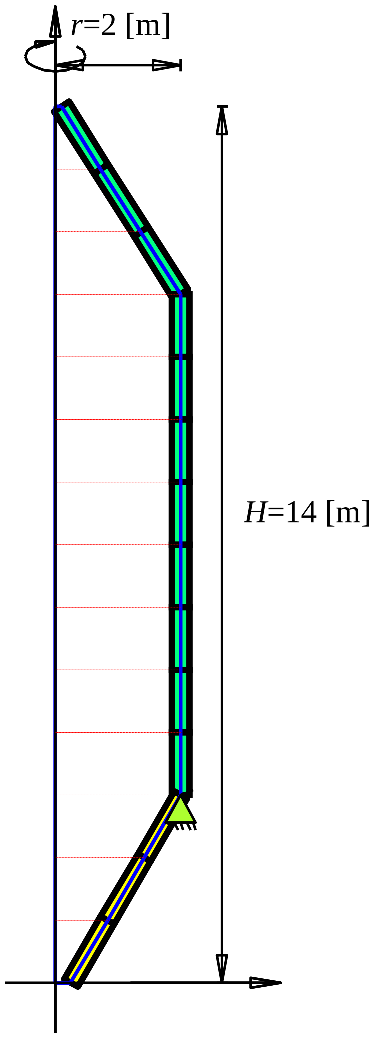


Figure 13.2 Segment dimensions

5 Creating the project

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

5.1 Calculation method

Choose "New Project" command from the "File" menu. The following "Calculation Methods" wizard appears, Figure 13.3. This wizard will help the user to define the analysis type and the calculation method of the problem through a series of forms. The first form of "Calculation Methods" wizard is the "Analysis Type" Form (Figure 13.3).

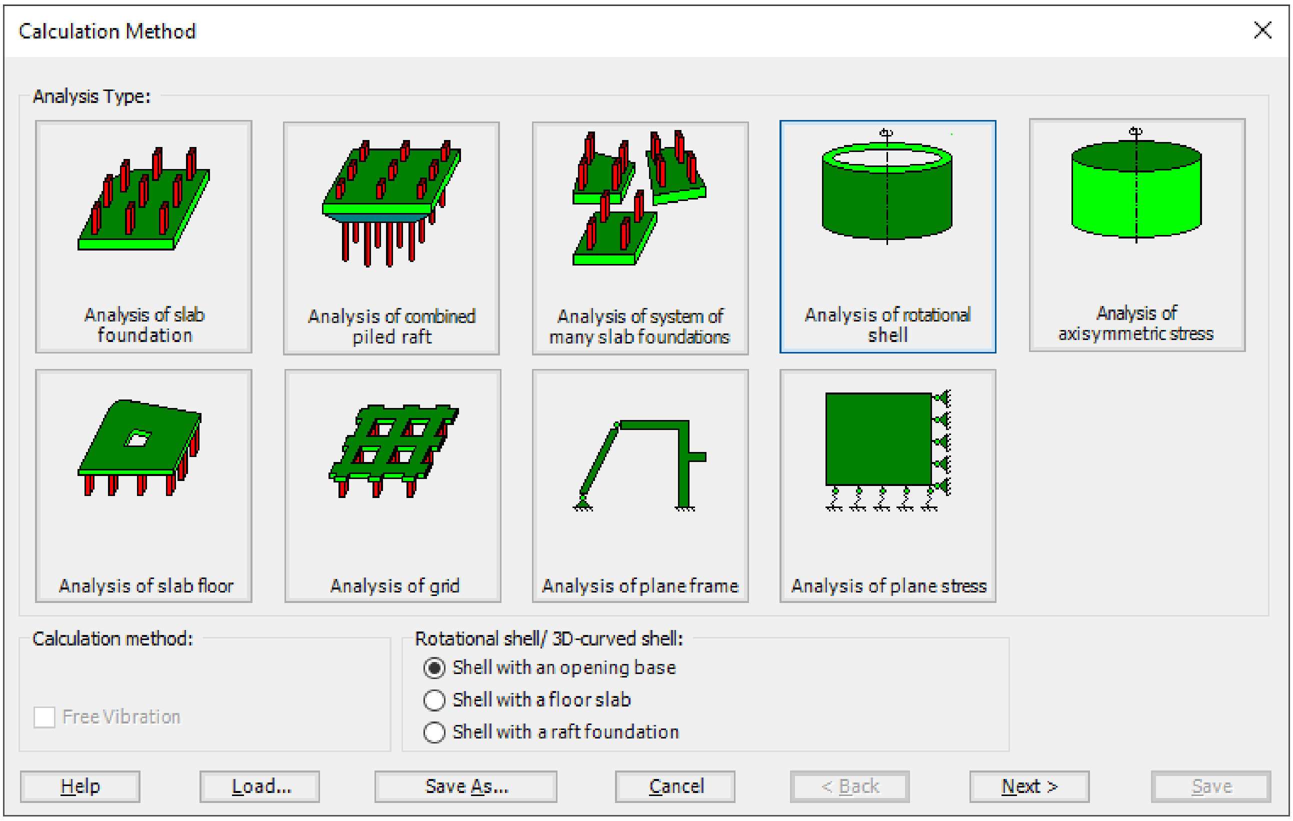


Figure 13.3 "Analysis Type" Form

In the "Analysis Type" Form in Figure 13.3, define the analysis type of the problem. As the analysis type is a circular concrete silo problem, select "Analysis of rotational Shell" button, and check "Shell with an opening base" option, then click "Next" button to go to the next Form. After clicking "Next" button, the "Options" Form appears, Figure 13.4.

The last Form in the wizard is the "Options" Form, Figure 13.4. In this Form, *ELPLA* displays some available options corresponding to the chosen numerical model, which differ from model to other. Select "Supports/ Boundary Conditions", then click the "Save" button.

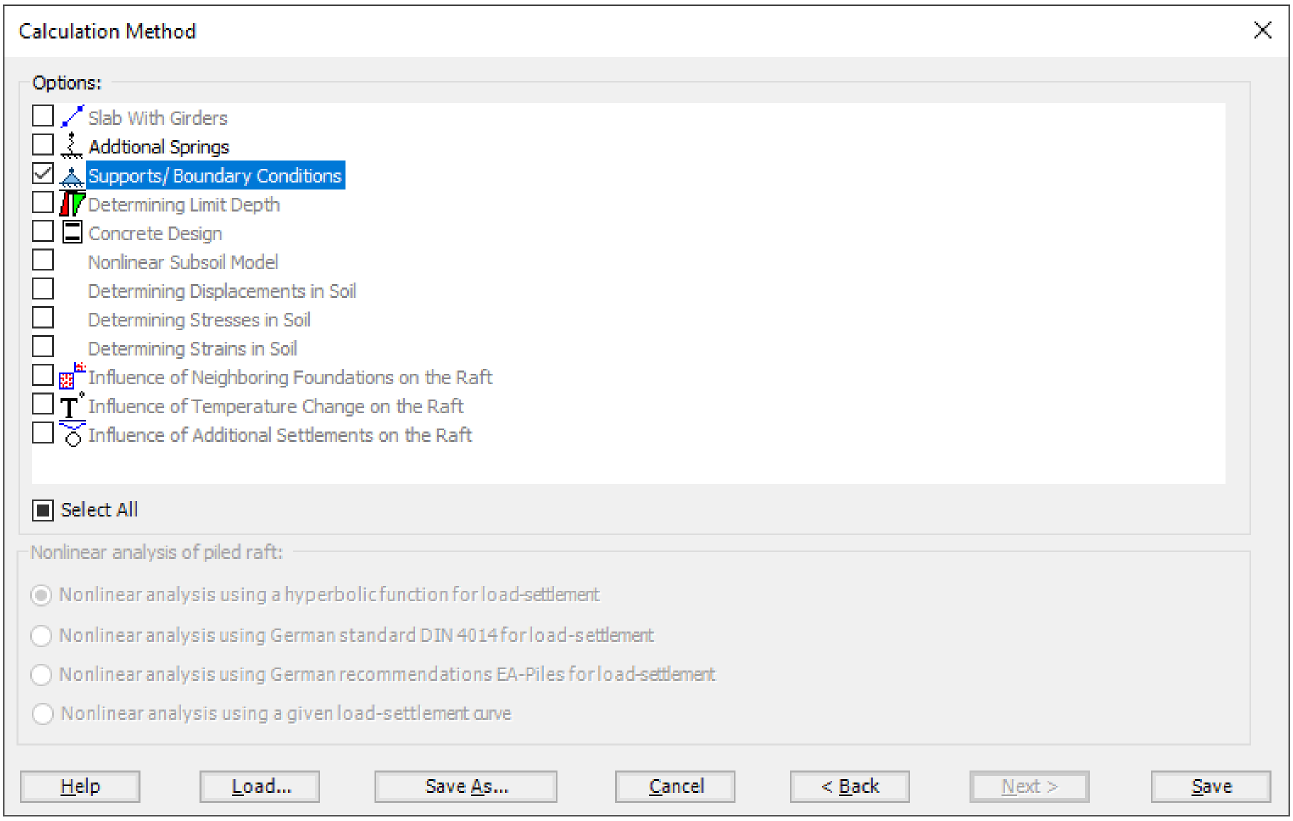


Figure 13.4 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 13.5. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Silo". *ELPLA* will use automatically this file name in all reading and writing processes.

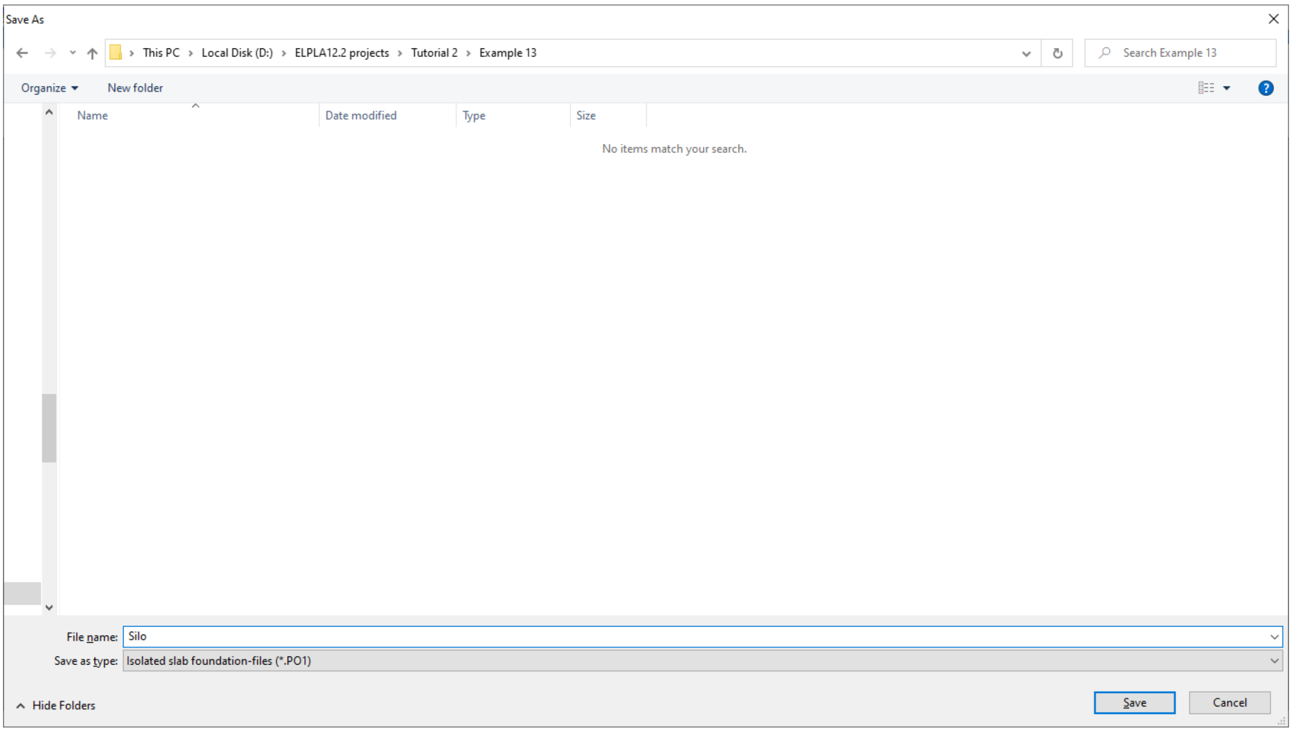


Figure 13.5 "Save as" dialog box

ELPLA will activate the "Data" Tab. In addition, the file name of the current project [Silo] will be displayed instead of the word [Untitled] in the *ELPLA* title bar.

5.2 Project identification

The user can enter three lines of texts to describe the problem and the basic information about the task. These texts are required only for printing and plotting the data and results. Project identification does not play any role in the analysis. The three lines are optionally and maybe not completely entered. To identify the project, choose "Project Identification" command from the “Data” Tab. The dialog box in Figure 13.6 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box: "Analysis of a circular silo for storing bulk materials"
- Type the date of the project in the "Date" edit box
- Type "Axisymmetric Structures and Tanks" in the "Project" edit box
- Click "Save" button

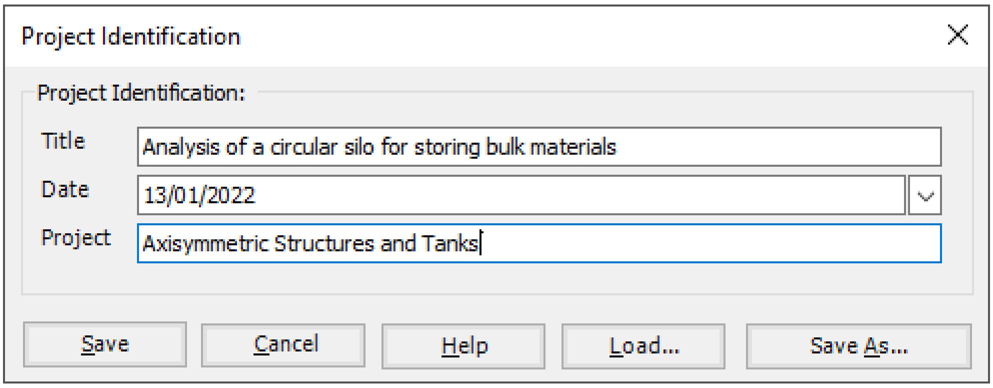


Figure 13.6 "Project Identification" dialog box

5.3 FE-Net data

For the given problem, a circular concrete silo having a conical hopper at the bottom part and a conical roof at the upper part is considered. The main height of the silo is 8 [m] and its diameter is 4 [m], The wall of the silo is divided into three parts, these three parts are divided into 14 segments; each segment is 1.0 [m]. Then these segments are divided into a number of elements, each element is 0.1 [m]. To define the FE-Net for this silo, choose "FE-Net Data" command from the “Data” Tab. "Analysis of rotational shell" wizard appears as shown in Figure 13.7. This wizard will guide you through the steps required to generate a FE-Net, the first Form of the wizard is the "Shell type" Form, which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets.

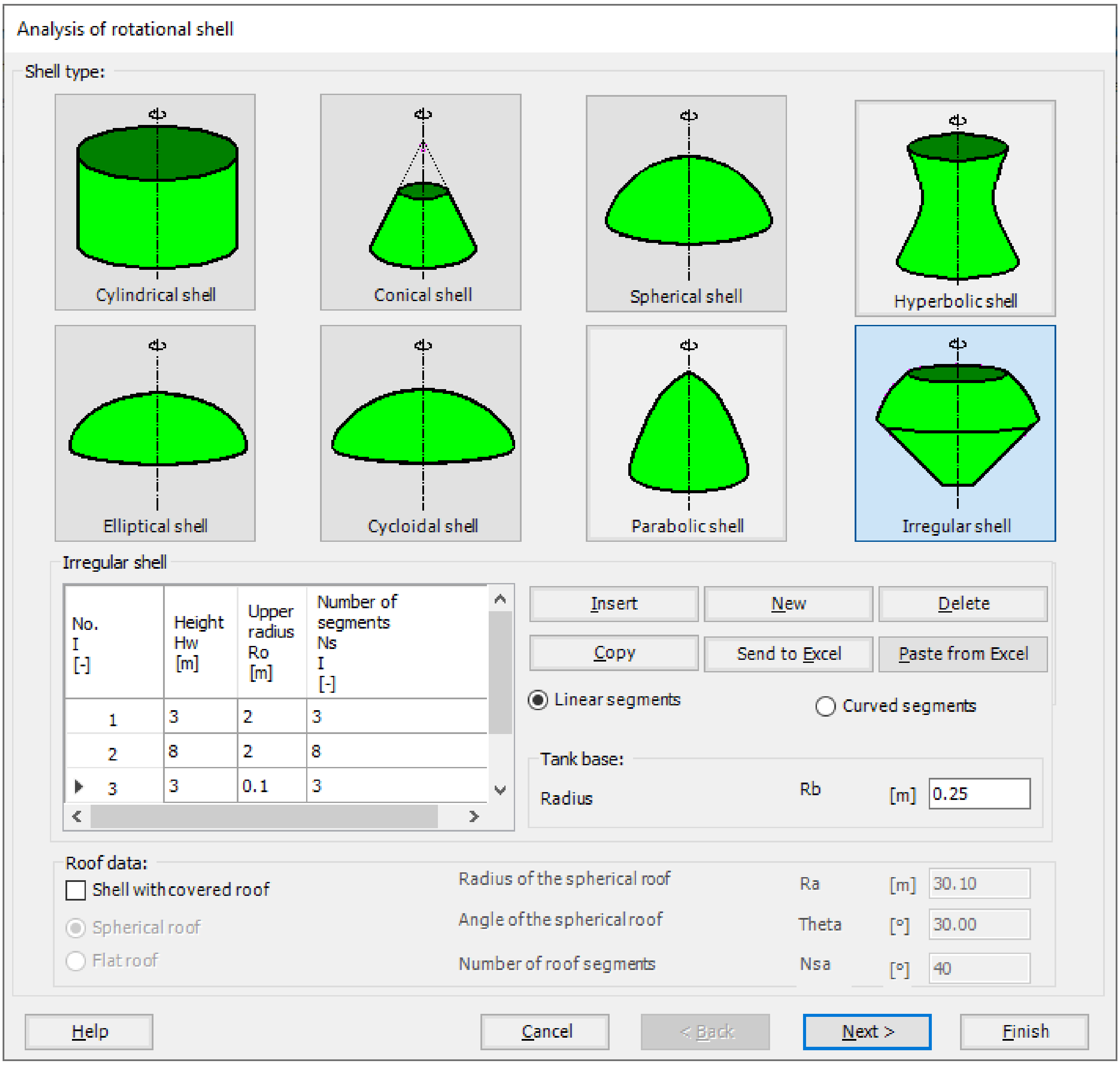


Figure 13.7 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Irregular shell" button
- Type 0.25 in the "Tank base radius *Rb*" edit box, as the opening diameter at the bottom is 0.5 [m]

To identify the hopper part

- Type 3 in the "Height *Hw*" edit box
- Type 2 in the "Upper radius *Ro*" edit box
- Type 3 in the "Number of segments *Ns*" edit box

To identify the main wall

- Type 8 in the "Height *Hw*" edit box
- Type 2 in the "Upper radius *Ro*" edit box
- Type 8 in the "Number of segments *Ns*" edit box

To identify the roof part

- Type 3 in the "Height *Hw*" edit box
- Type 0.1 in the "Upper radius *Ro*" edit box, which is approximately zero
- Type 3 in the "Number of segments *Ns*" edit box

- Click "Next" button to go to the next Form

After clicking "Next" in "Analysis of rotational shell" wizard, the following "Irregular shell" Form containing the data of the segments appears in Figure 13.8, the user can edit the data of each segment individually or all of them by using "In Table" button, if it is necessary.

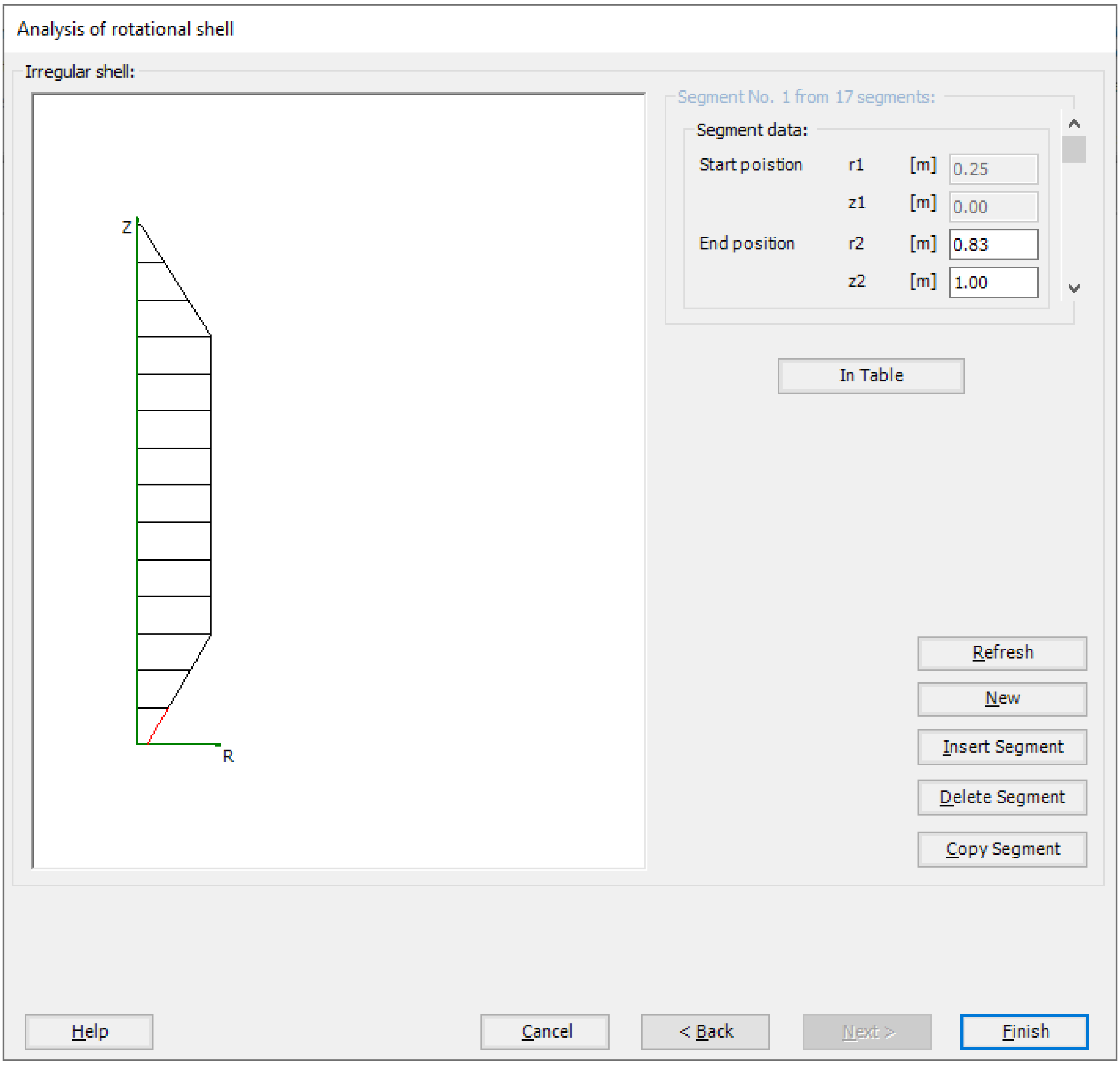


Figure 13.8 "Irregular shell" Form

Click "Finish" in "Analysis of rotational shell" wizard, the generated FE-Net appears Figure 13.9.

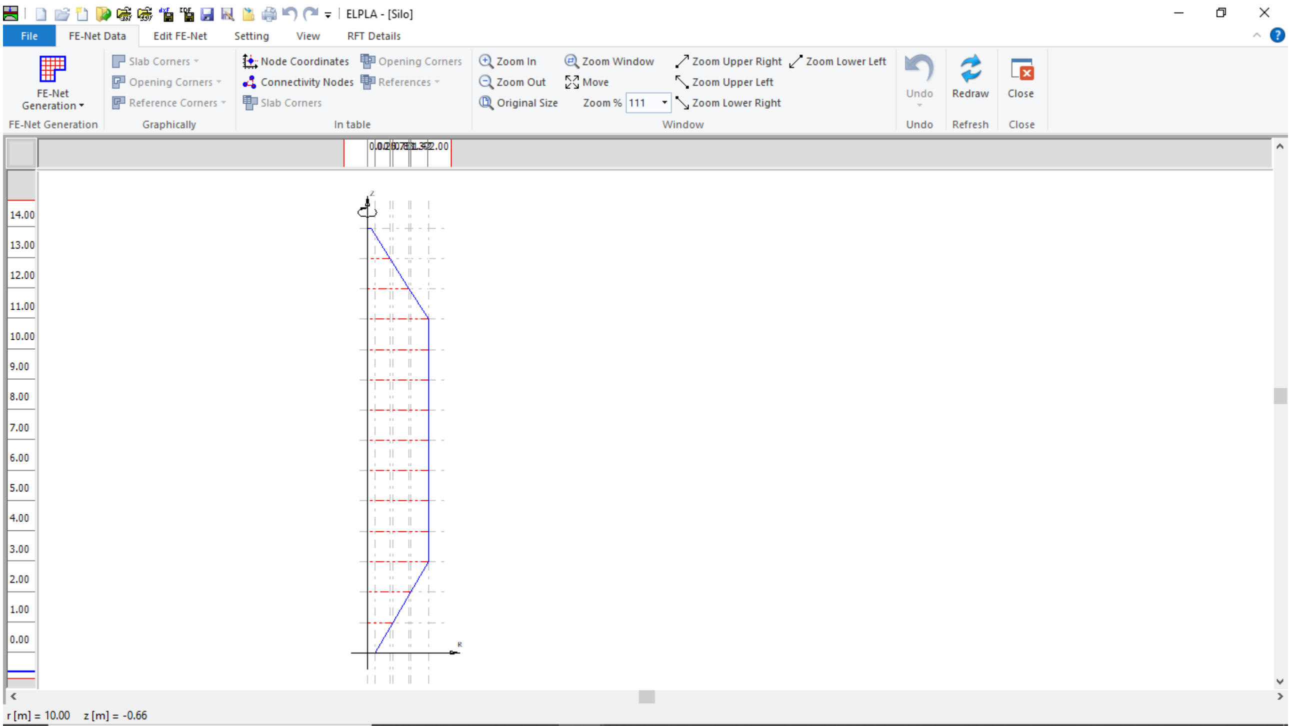


Figure 13.9 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 13.9 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 13.9 to close the "FE-Net" window and return to *ELPLA* main window

5.4 Shell properties

To define the silo properties, choose "Shell Properties" command from “Data” Tab. The following window in Figure 13.10 appears with default shell properties. The data of shell properties for the current example, which are required to be defined, are element groups, group regions, unit weight of the silo material, filled material properties and element size.

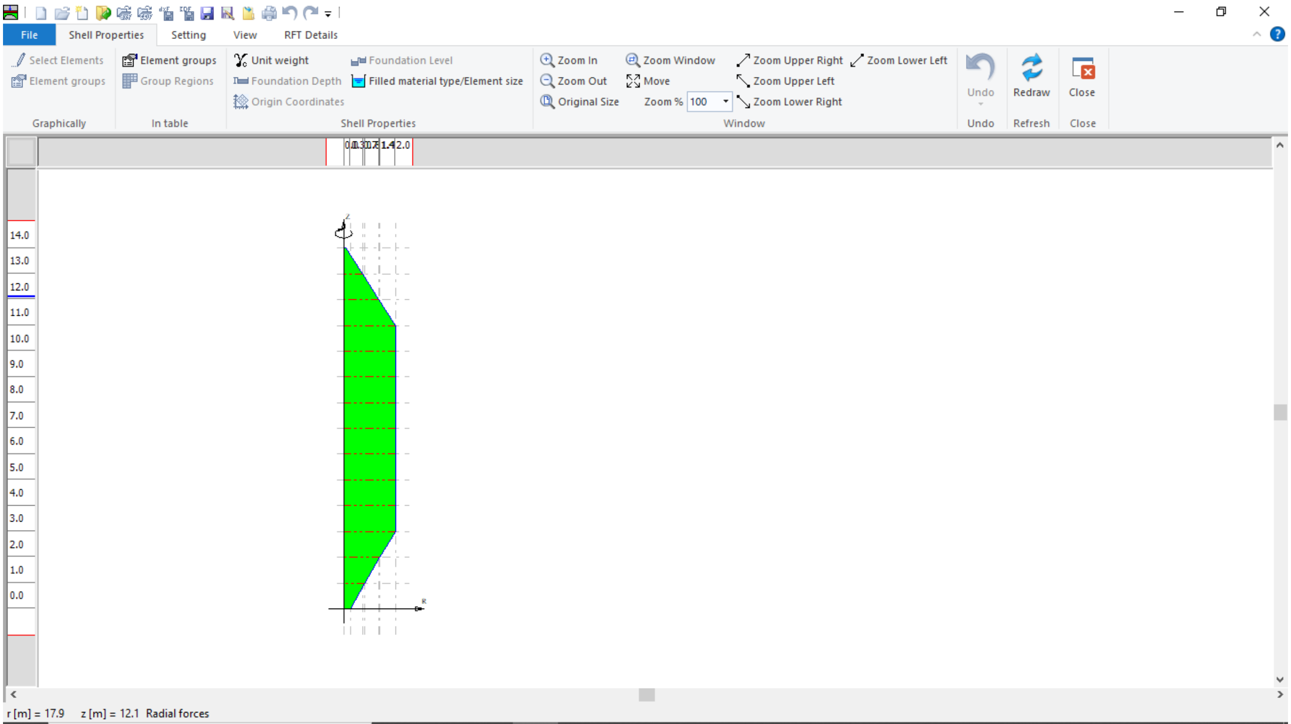


Figure 13.10 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 13.11 appears. In this list box, define E-Modulus, *Poisson’s* ratio and slab thickness for both the wall and the hopper of the silo as they differ in thickness. Then click "OK" button.

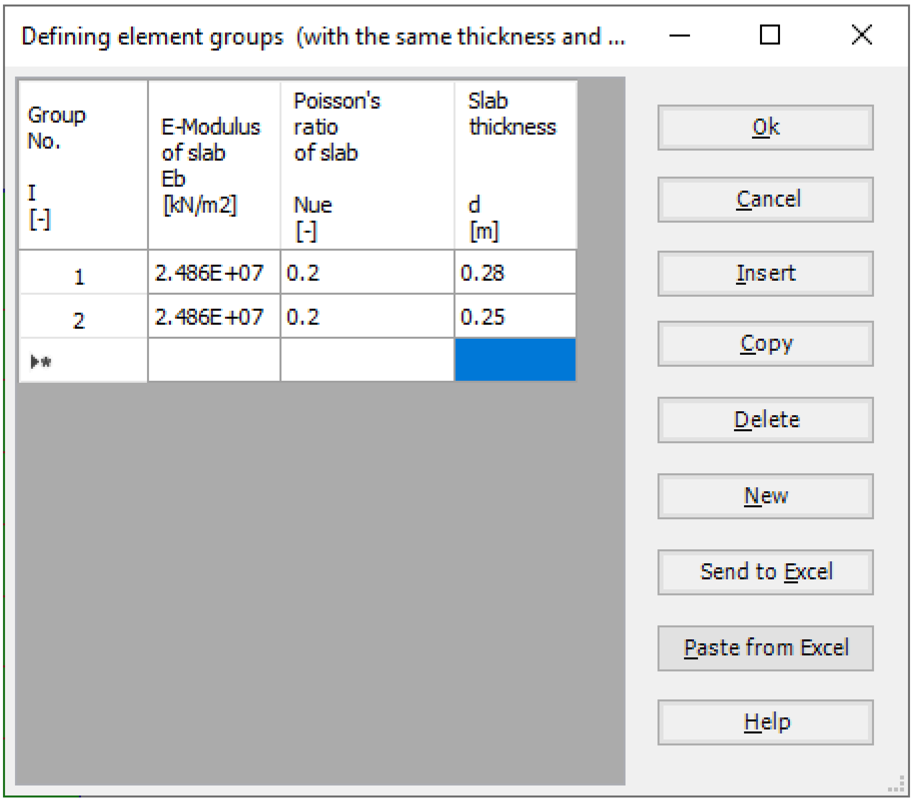


Figure 13.11 "Defining element groups" list box

Defining the slab thickness for materials on the net may be carried out either graphically or numerically (in a table). In the current example, the user will define the slab thickness on the net graphically.

To define the slab thickness for the silo hopper

- Choose "Select Elements" command from "Graphically" menu in the window of Figure 13.10
- When "Select Elements" command is chosen, the cursor will change from an arrow to a cross hair. A group of elements can be selected by holding the left mouse button down at the corner of the region. Then, drag the mouse until a rectangle encompasses the required group of elements. When the left mouse button is released, all elements in the rectangle are selected
- Select the elements that include the silo hopper as Figure 13.12
- Choose "Elements Groups" command from "Graphically" menu in the window of Figure 13.10, "Group Regions" dialog box Figure 13.13 appears
- Define the "Group No." of the hopper elements as type "2", while "Group No." of the wall and the roof elements will be as type "1", where type "1" is the default "Group No.", then click "OK" button

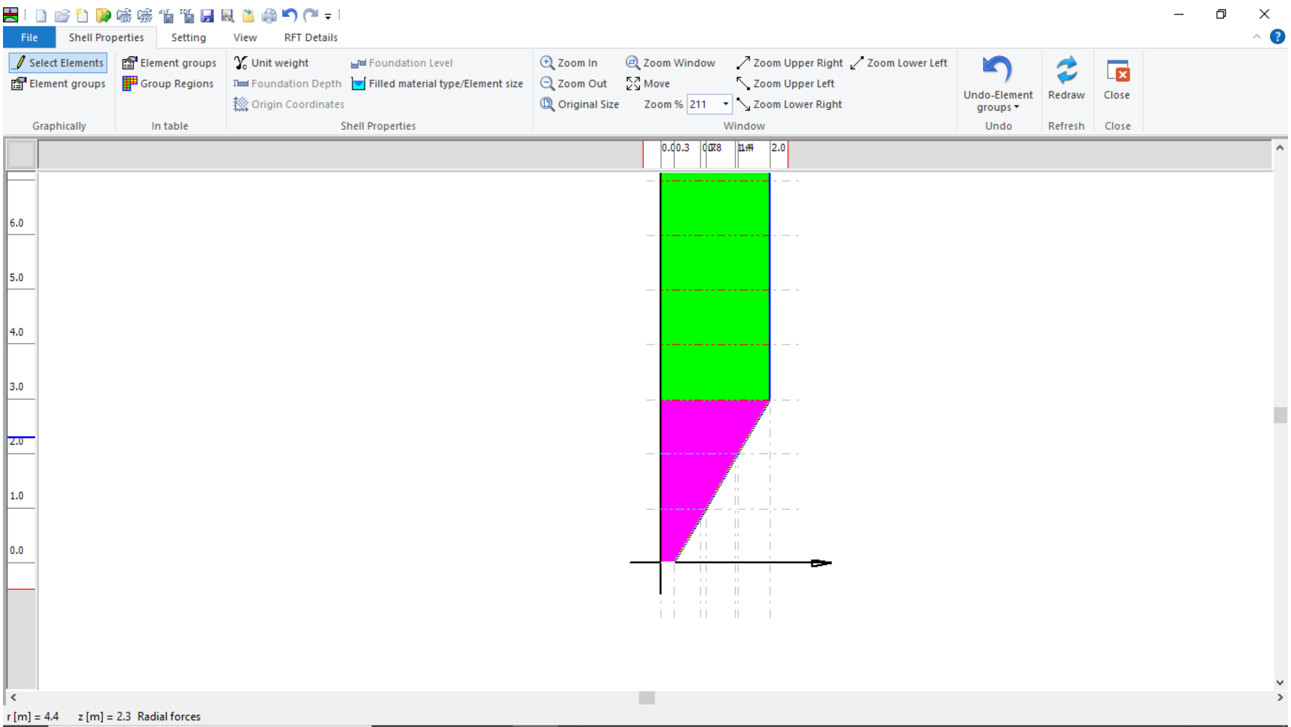


Figure 13.12 Selecting the elements that include the silo hopper

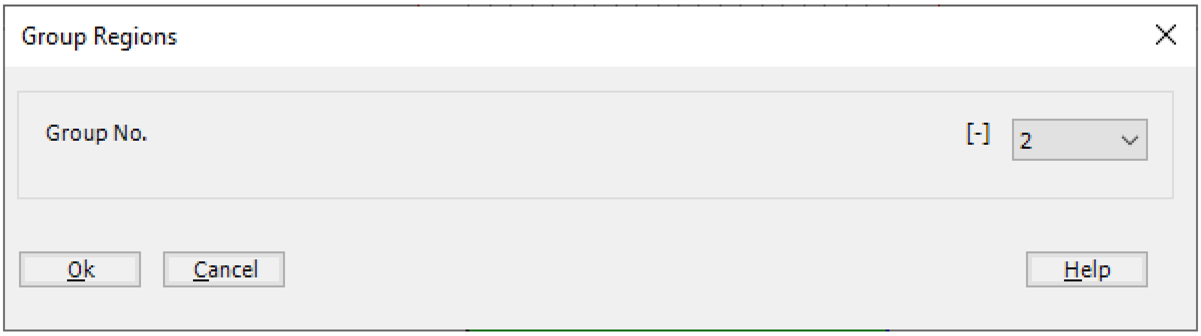


Figure 13.13 "Group Regions" dialog box

To enter the unit weight of the silo material, choose "Unit weight" command from "Shell Properties" menu in Figure 13.10. The following dialog box in Figure 13.14 with a default unit weight of 25 [kN/m^3] appears, type 23.5 in the "Unit weight" edit box, then click "OK" button.

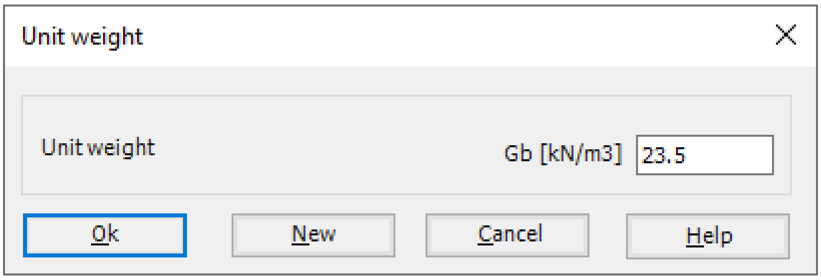


Figure 13.14 "Unit weight" dialog box

To define the filled material properties:

- Choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 13.10. The following form in Figure 13.15 appears
- Select "Granular material container" option
- Define the granular material properties as follows in Figure 13.15

To define the element size of the container:

- Check the "Constant element sizes in z-direction" check box
- Type 0.2 in the "Element size in each shell segment" edit box
- Click "OK" button

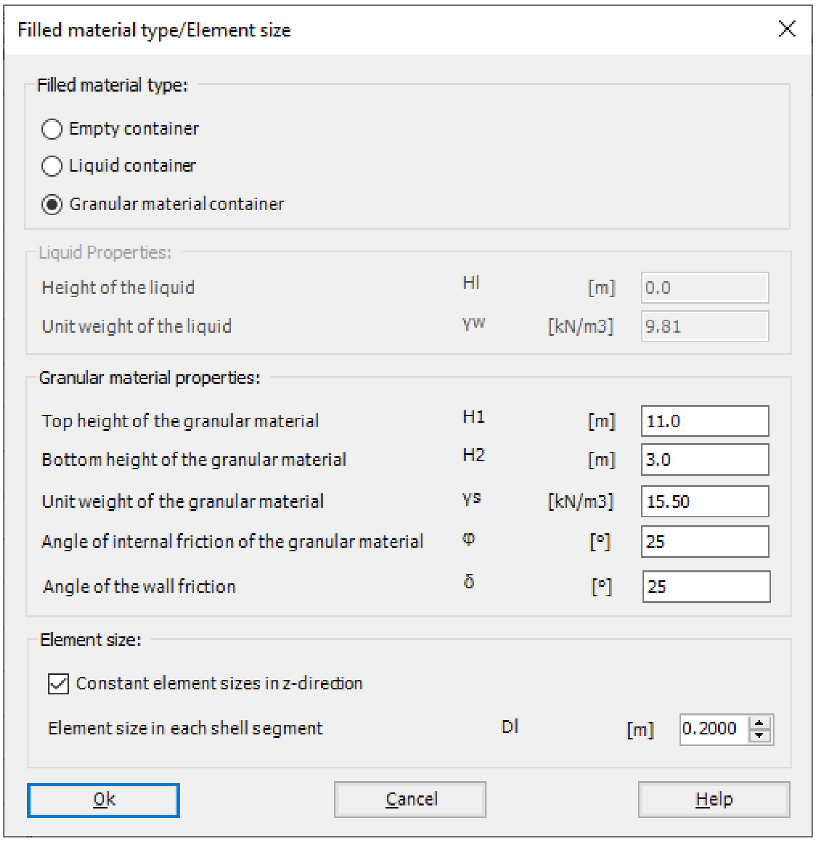


Figure 13.15 "Filled material type/Element size" Form

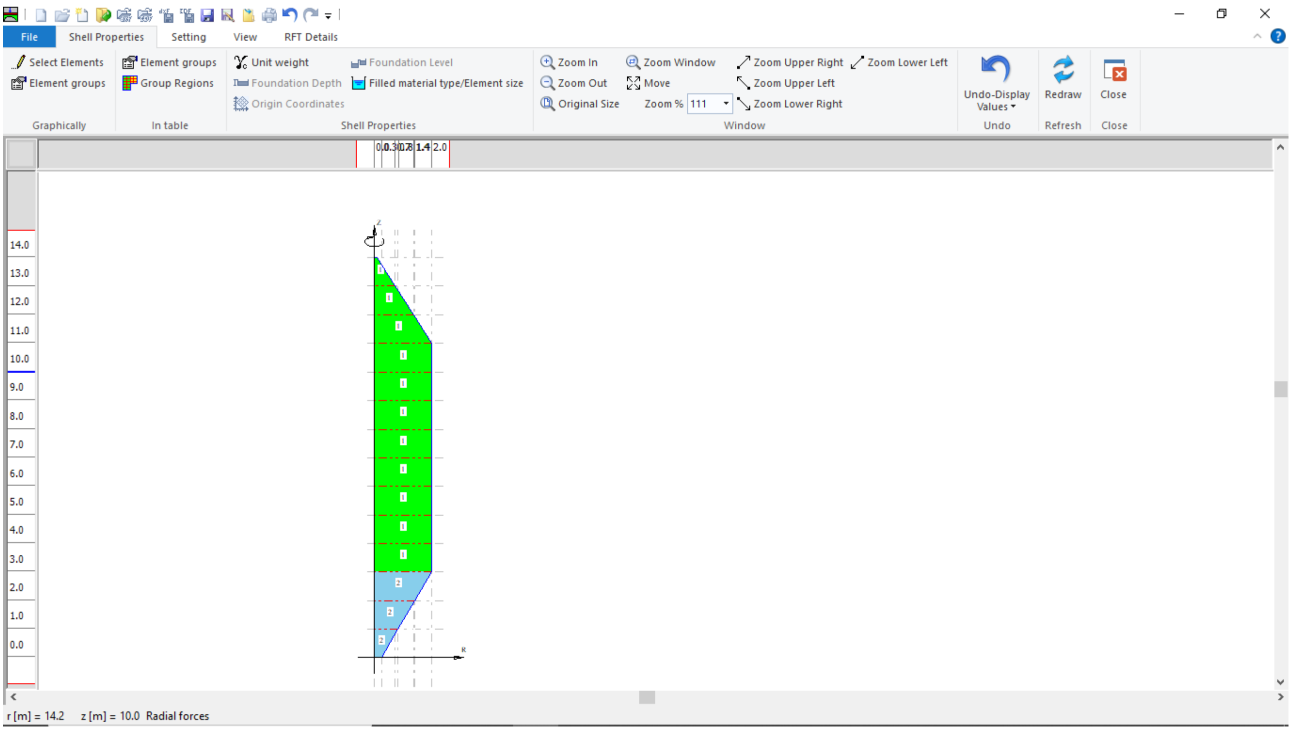


Figure 13.16 "Shell Properties" window after defining the silo properties

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

- Choose "Save" command from "File" menu in Figure 13.16 to save the shell properties
- Choose "Close" command from "File" menu in Figure 13.16 to close the "Shell properties" window and return to *ELPLA* main window

5.5 Supports/ boundary conditions

To define the support, choose "Supports/ Boundary Conditions" command from “Data” Tab. The following window in Figure 13.17 appears.

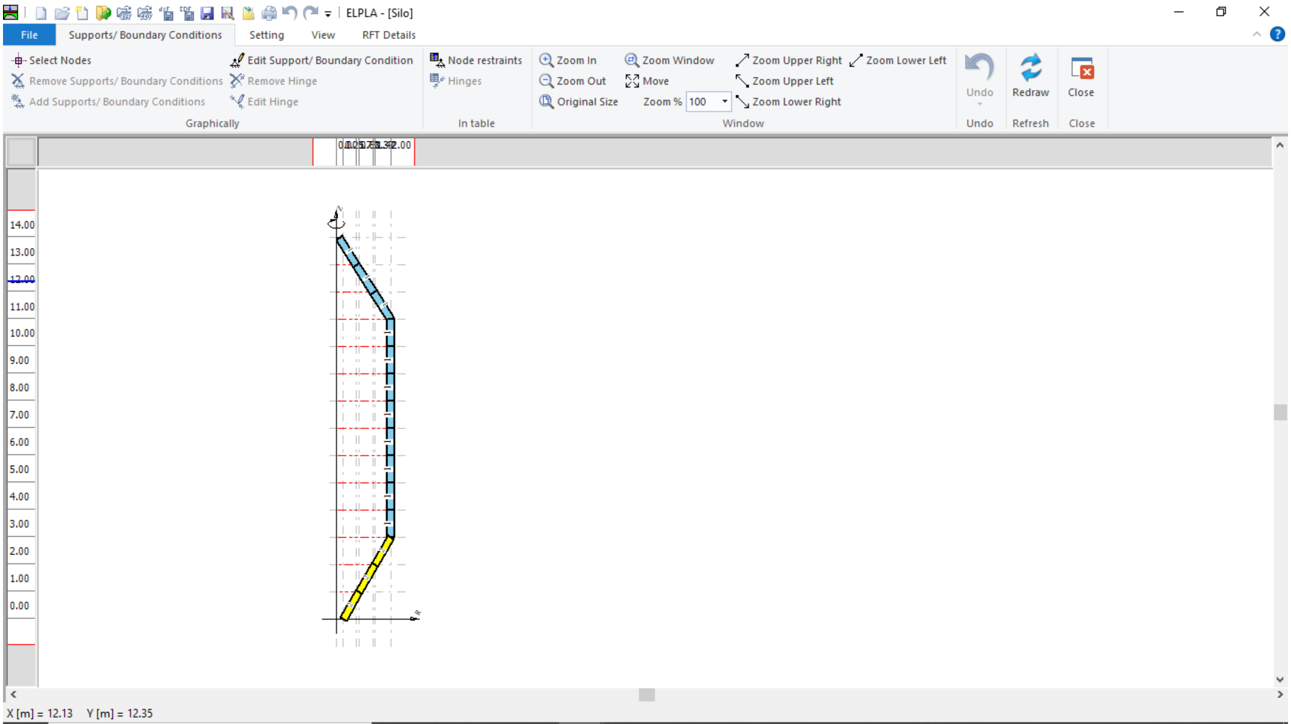


Figure 13.17 "Supports/ Boundary Conditions" Window

To define the support on the net:

- Choose "Select Nodes" command from "Graphically" menu in Figure 13.17. When "Select Nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on the node that has the support as shown in Figure 13.18
- After selecting the node, choose "Add Supports/ Boundary Conditions" command from "Graphically" menu (Figure 13.17). The "Supports/ Boundary Conditions" dialog box in Figure 13.19 appears

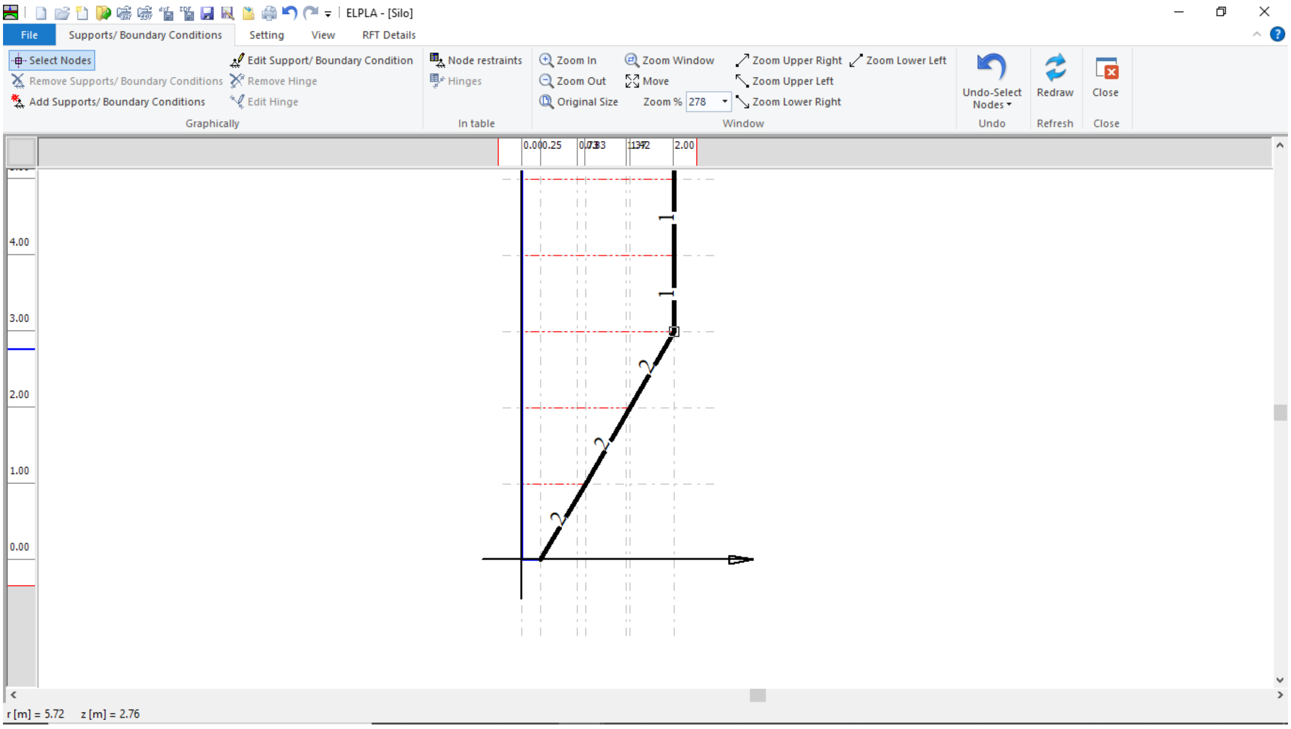


Figure 13.18 Selection of the node that has the support

In this dialog box

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

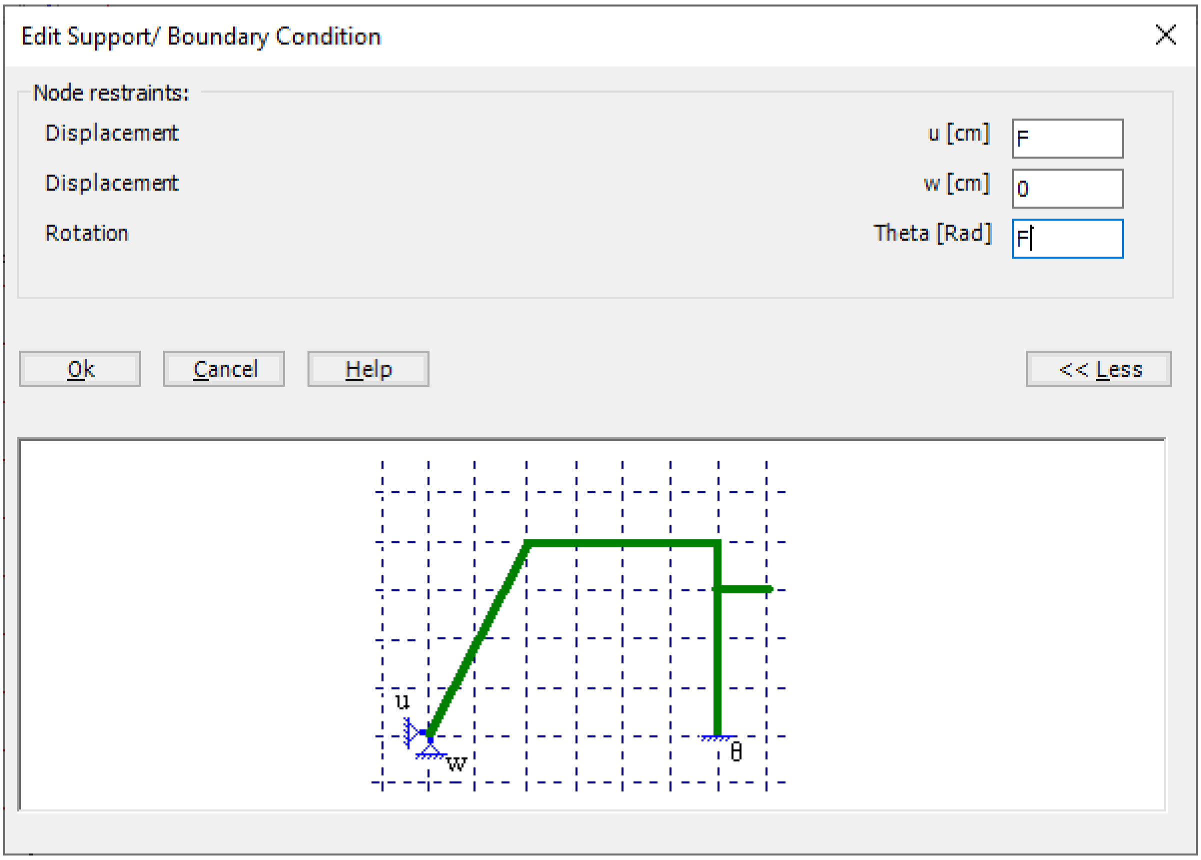


Figure 13.19 "Supports/ Boundary Conditions" dialog box

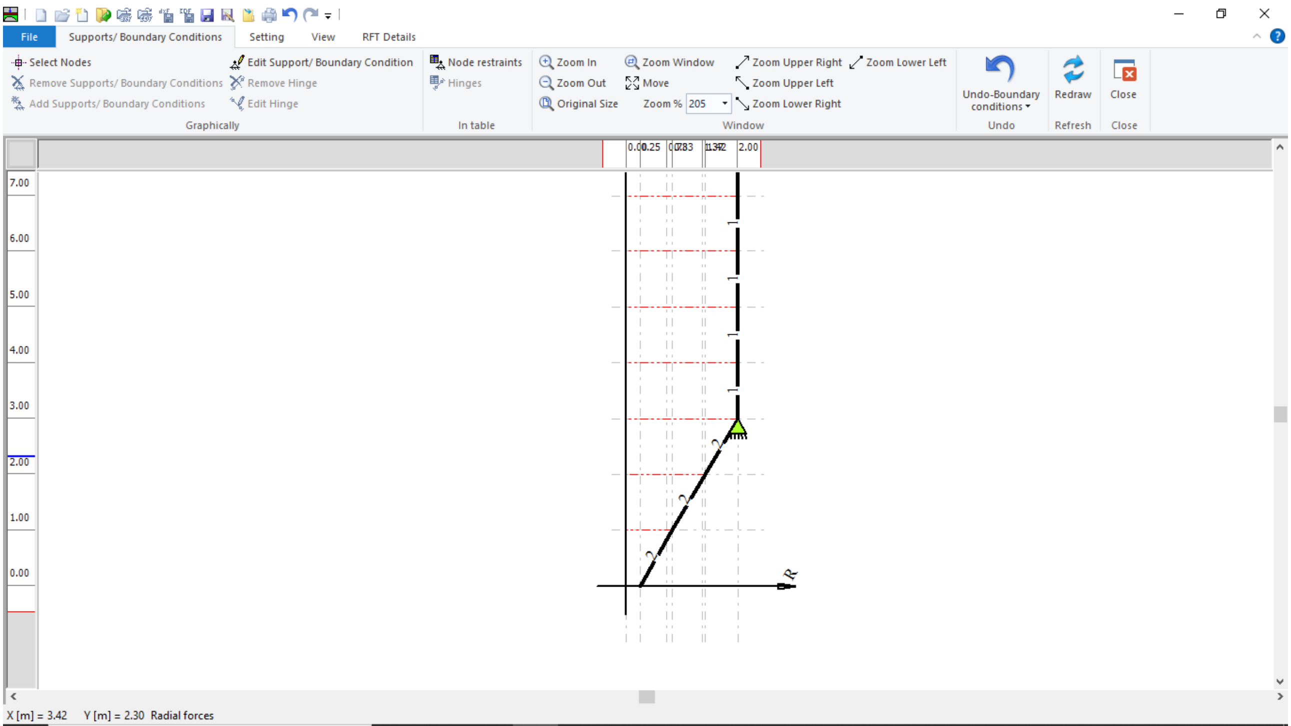


Figure 13.20 "Supports/ Boundary Conditions" window after defining the supports

After defining the supports, do the following two steps

- Choose "Save " command from "File" menu in Figure 13.20 to save the data of supports
- Choose "Close" command from "File" menu in Figure 13.20 to close the "Supports/ Boundary conditions" window and return to the main window

5.6 Loads

To define the loads, choose "Loads" command from “Data” Tab. The following window in Figure 13.21 appears.

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 13.21. In this example, there is not applied load, as the lateral load on the main wall has been already defined by P_h.

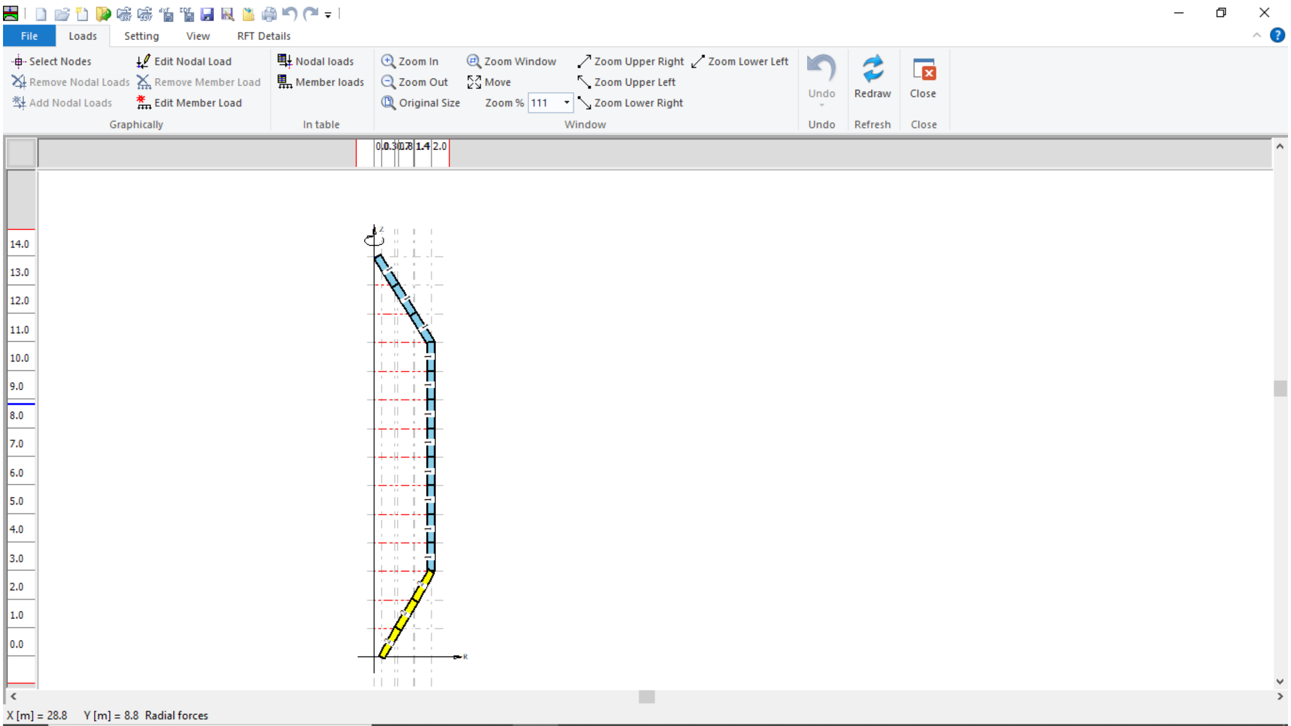


Figure 13.21 "Loads" Window

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

- Choose "Save" command from "File" menu in Figure 13.21 to save the load data
- Choose "Close" command from "File" menu in Figure 13.21 to close the "Loads" window and return to *ELPLA* main window

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

6 Carrying out the calculations

To analyze the problem, switch to "Solver" Tab, Figure 13.22.

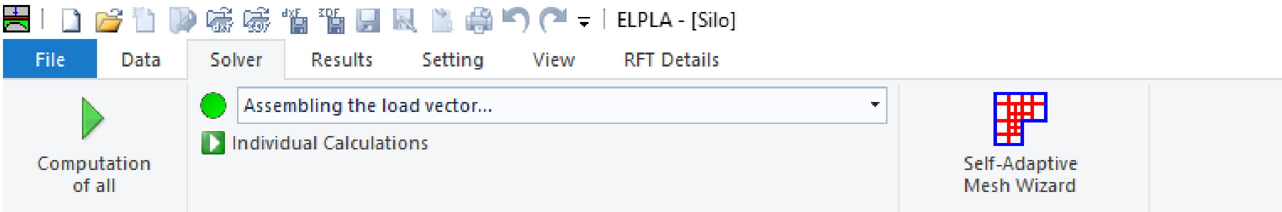


Figure 13.22 "Solver" Tab

ELPLA will active the "Individual Calculations" list, which contains commands of all calculations. Commands of calculation depend on the used calculation method in the analysis. For this project, the items that are required to be calculated are:

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

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

To carry out all computations in one time

- Choose "Computation of all" command from "Solver" Tab window.

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

Analysis progress

Analysis progress menu in Figure 13.23 appears in which various phases of calculation are progressively reported as the program analyzes the problem. In addition, a status bar down of the "Solver" Tab window displays Information about the progress of calculation.

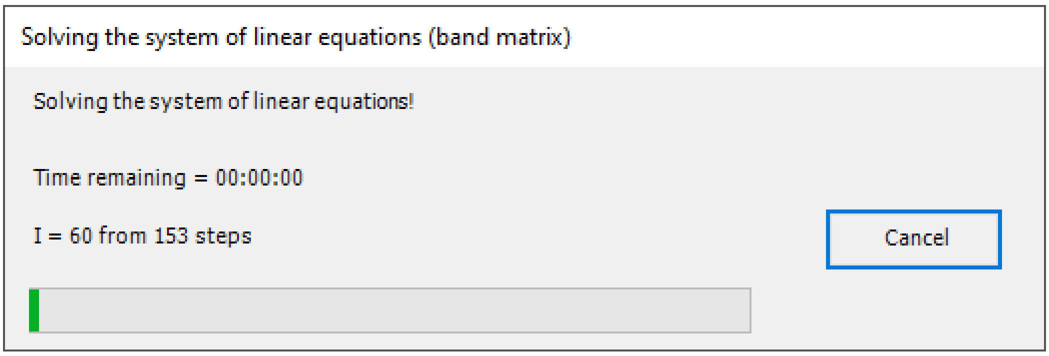


Figure 13.23 Analysis progress menu

Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 13.24. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

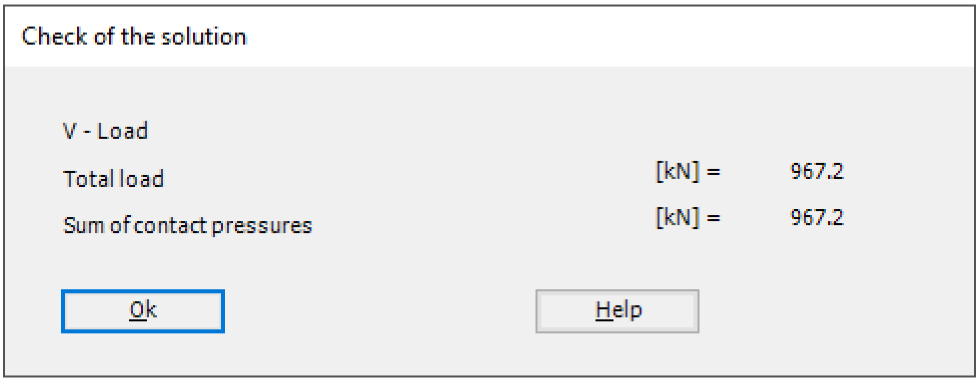


Figure 13.24 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

7 Viewing data and results

ELPLA can display and print a wide variety of results in graphics, diagrams or tables through the "Results" Tab. To view the data and results of a problem that has already been defined and analyzed graphically, switch to "Results" Tab (Figure 13.25).

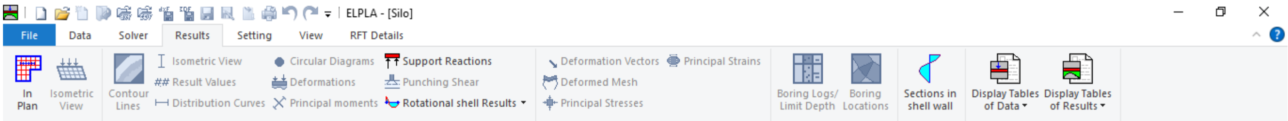


Figure 13.25 "Results" Tab

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

- Data in the plan
- Rotational shell results
- Support Reactions
- Sections in shell wall
- Display tables of data
- Display tables of results

To view the radial forces on the silo wall

- Choose "Sections in shell wall" command from "Section" menu. The following option box in Figure 13.26 appears
- In the "Sections in shell wall" option box, select "Radial forces *Nr*" as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 13.27.

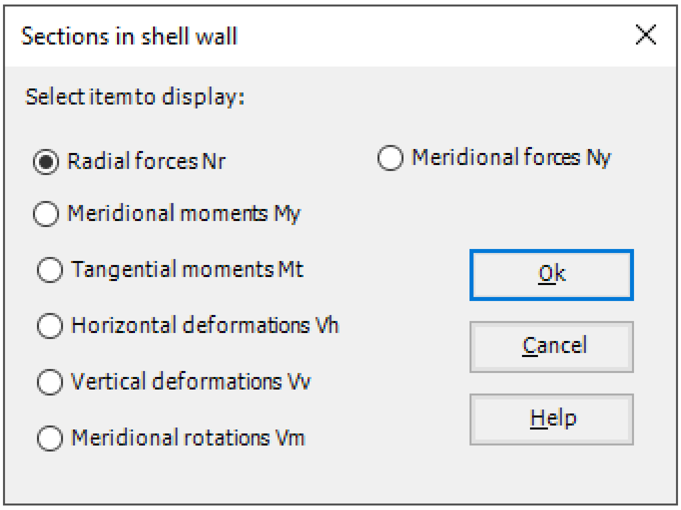


Figure 13.26 "Sections in shell wall" option box

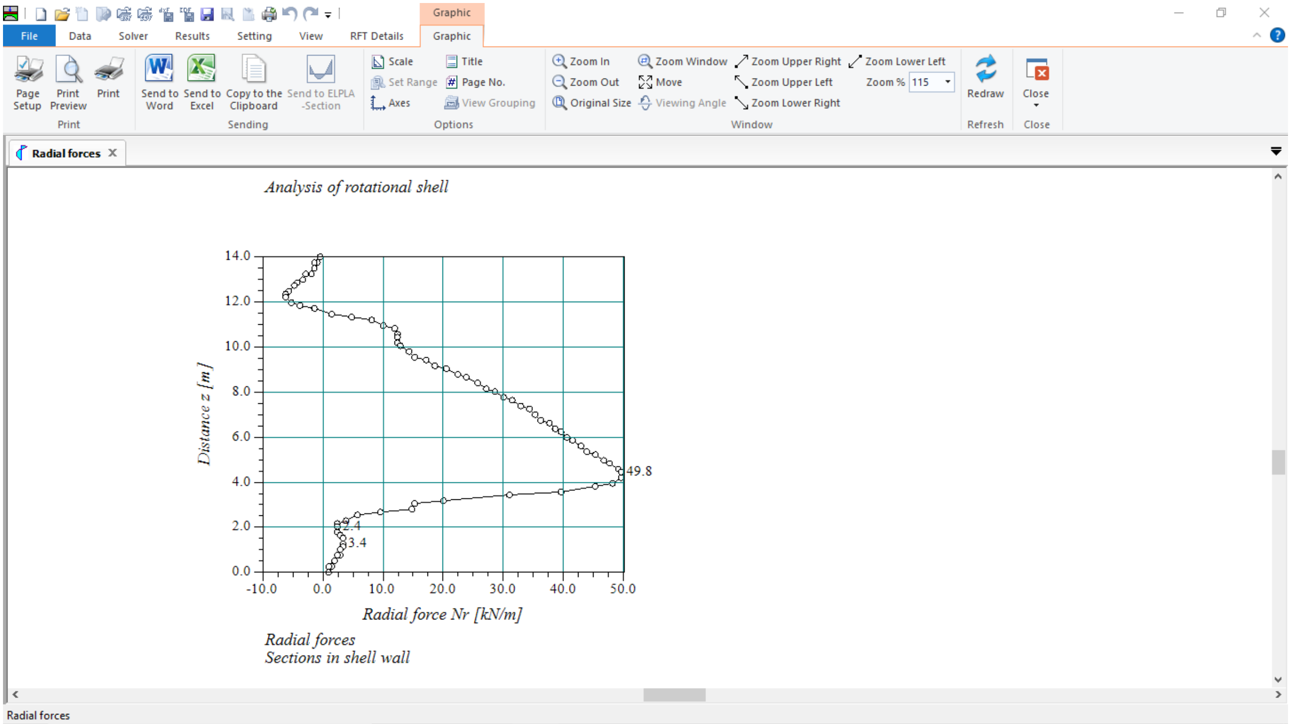


Figure 13.27 Radial forces on the silo wall

To view element groups of the silo

- Choose "Element groups" from "In Plan" command in "Data" menu. The following option box in Figure 13.28 appears
- In the "Data – In Plan" option box, select "Element groups" as an example for the results to be displayed
- Click "OK" button

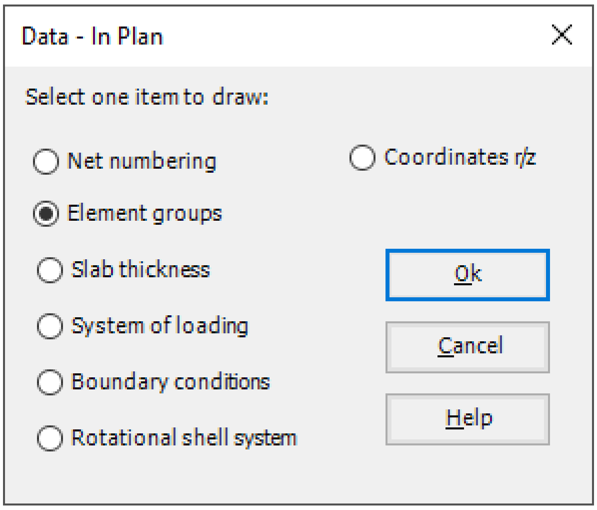


Figure 13.28 "Data – In Plan" option box

To view the supports / boundary conditions on the FE-Net and any other data

- From "Options" menu in the "Graphic" tab, choose "View Grouping" command. The "View Grouping" check group box in Figure 13.29 appears
- In this check group box, check "Radial forces *Nr*", "Supports Reactions *RV*" and "Supports /Boundary Conditions" check boxes
- The user can choose any other data to be viewed
- Click "OK" button

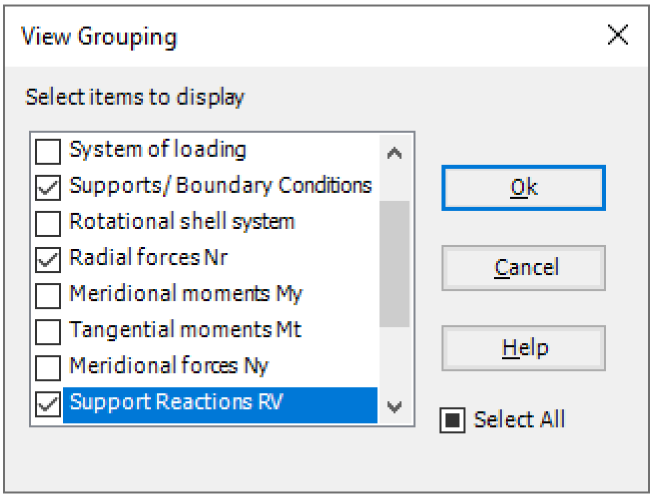


Figure 13.29 "View Grouping" check group box

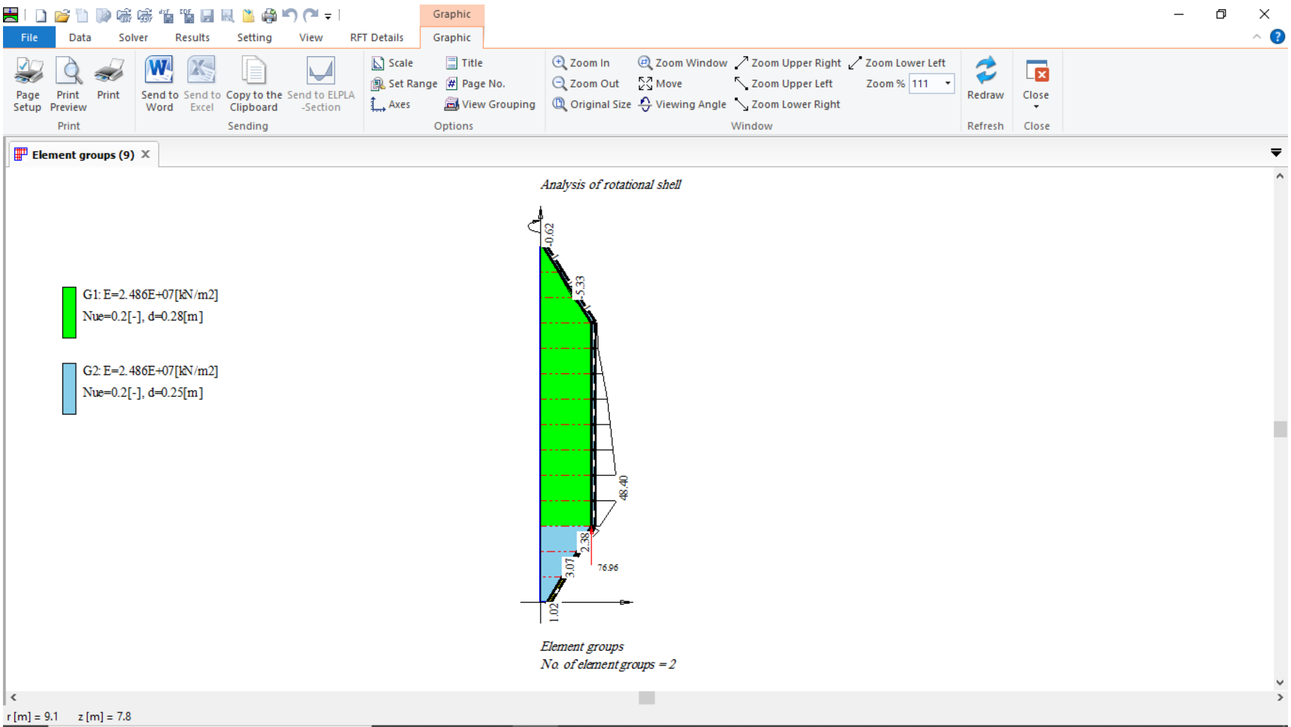


Figure 13.30 Element groups of the silo