

## **Example 9**

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

<b>Contents</b>	<b>Page</b>
1 Description of the problem .....	3
2 Tank geometry and properties .....	3
3 Numerical Analysis .....	4
4 Creating the project .....	5
4.1 Calculation method .....	5
4.2 Project identification .....	9
4.3 FE-Net data .....	9
4.4 Shell properties .....	14
4.5 Soil properties .....	19
4.6 Loads .....	20
5 Carrying out the calculations .....	21
6 Viewing data and results .....	24

## Example 9

### 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.

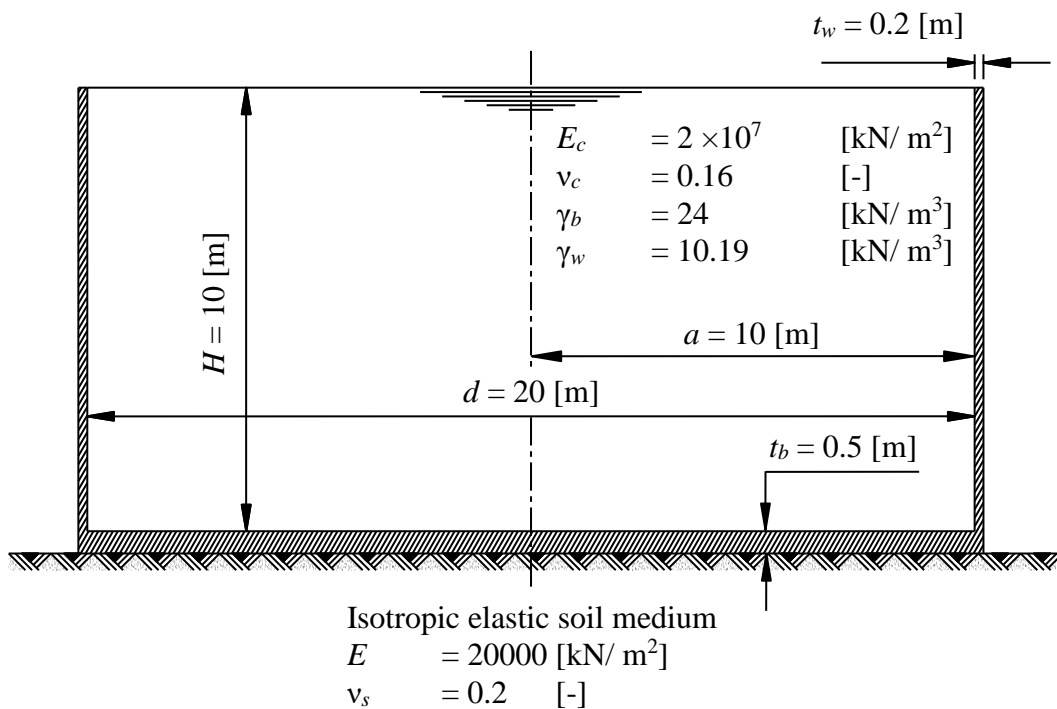


Figure 9.1 Circular cylindrical tank resting on an isotropic elastic soil medium

Table 9.1 Tank material and water unit weight

Modulus of Elasticity of the tank material	$E_c$	$= 2 \times 10^7$	[kN/m <sup>2</sup> ]
Poisson's ratio of the tank material	$\nu_b$	$= 0.16$	[-]
Unit weight of the tank material	$\gamma_b$	$= 24$	[kN/m <sup>3</sup> ]
Unit weight of the water	$\gamma_w$	$= 10.19$	[kN/m <sup>3</sup> ]

Table 9.2 Soil data

Modulus of Elasticity of the soil medium	$E$	$= 20000$	[kN/m <sup>2</sup> ]
Poisson's ratio of the soil medium	$\nu_s$	$= 0.2$	[-]

### 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].

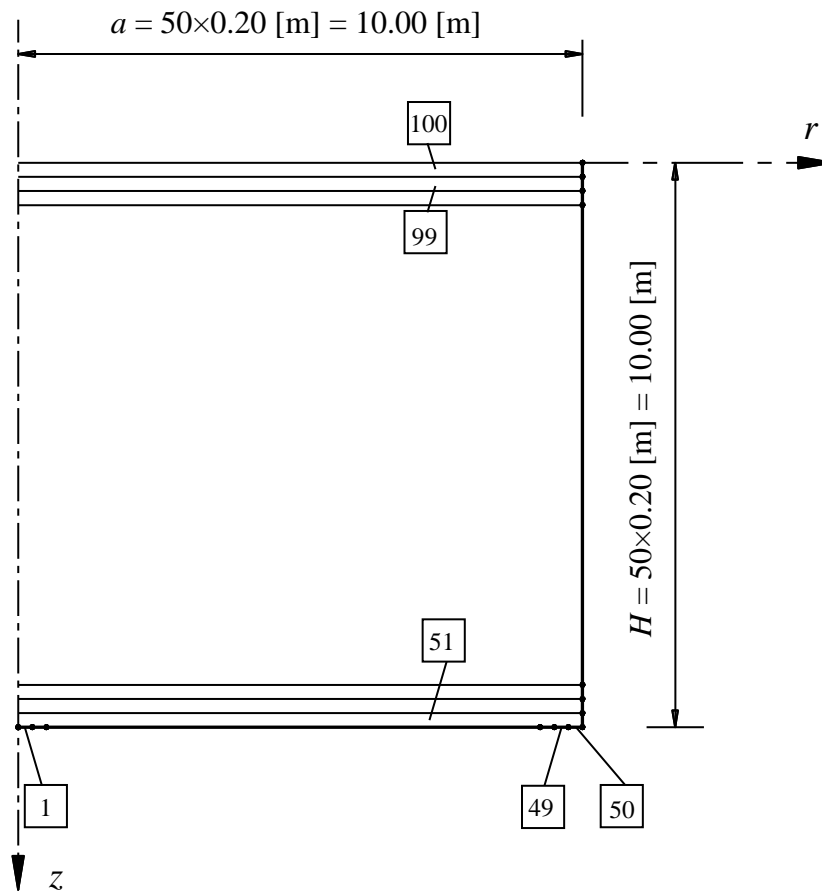


Figure 9.2 Finite element mesh of the tank

## 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).

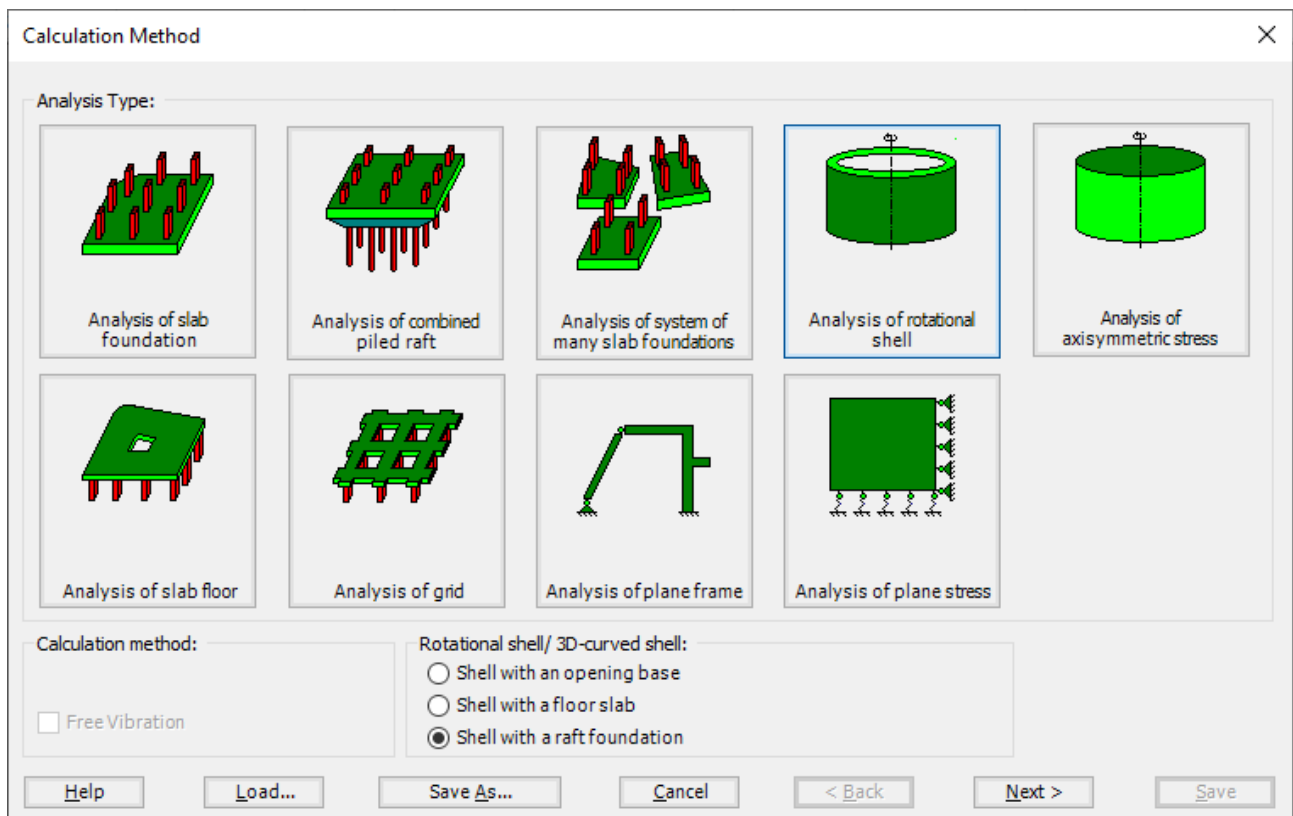


Figure 9.3 "Analysis Type" Form

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.

## Example 9

Calculation Method

Options:

- Slab With Girders
- Additional Springs
- Supports/ Boundary Conditions
- Determining Limit Depth
- Concrete Design
- Nonlinear Subsoil Model
- Determining Displacements in Soil
- Determining Stresses in Soil
- Determining Strains in Soil
- Influence of Neighboring Foundations on Raft
- Influence of Temperature Change on the Raft
- Influence of Additional Settlements on the Raft

Select All

Nonlinear analysis of piled raft:

- Nonlinear analysis using a hyperbolic function for load-settlement
- Nonlinear analysis using German standard DIN 4014 for load-settlement
- Nonlinear analysis using German recommendations EA-Piles for load-settlement
- Nonlinear analysis using a given load-settlement curve

Help Load... Save As... Cancel < Back Next > Save

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.

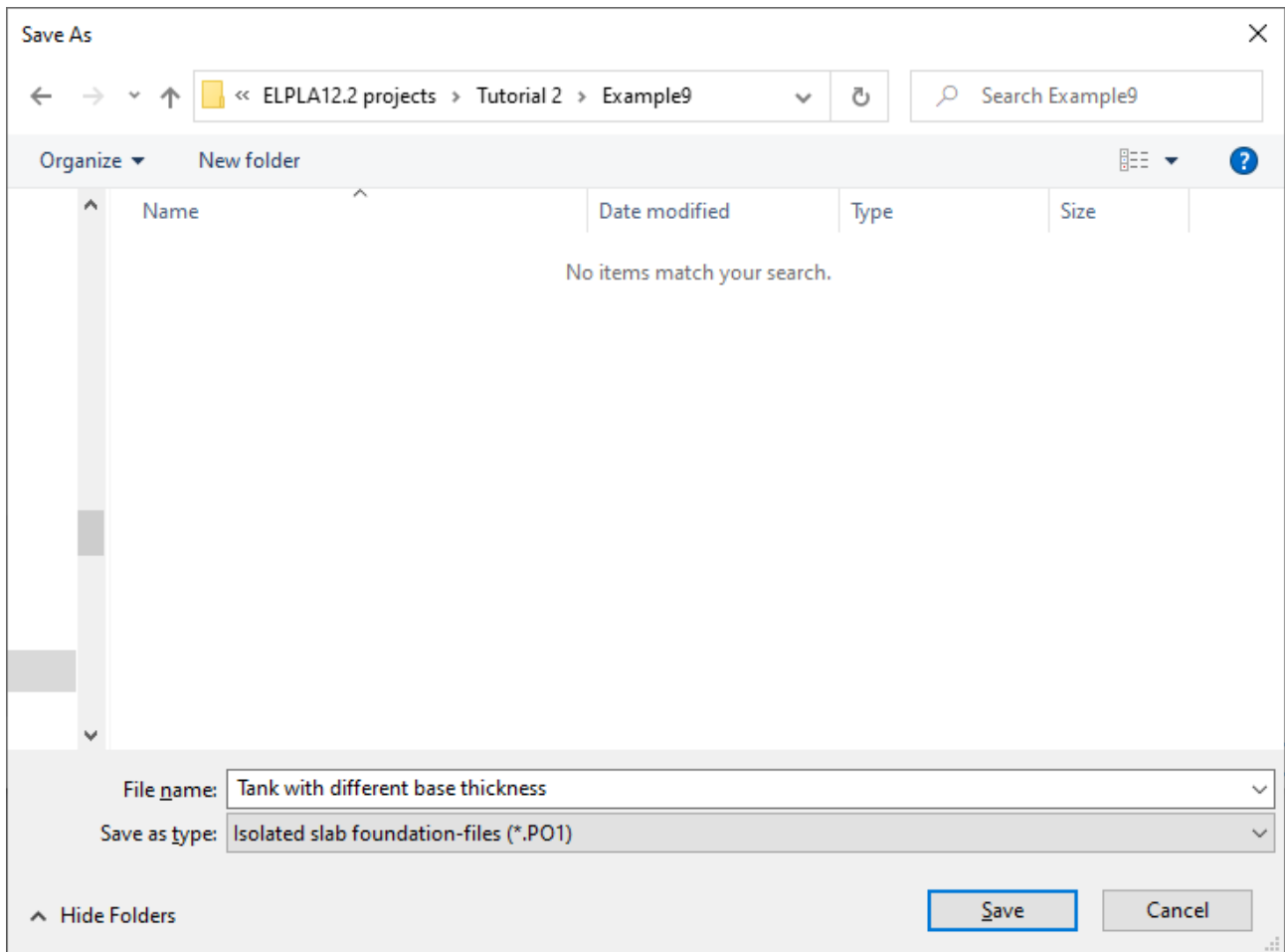


Figure 9.6 "Save as" dialog box

*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

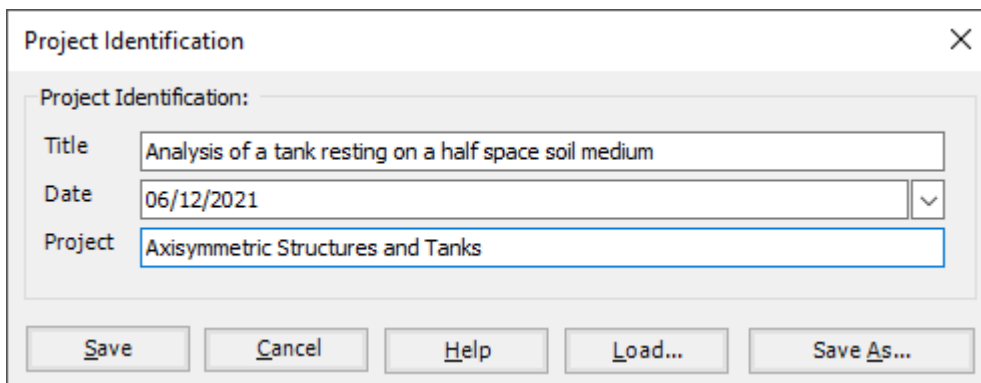


Figure 9.7 "Project Identification" dialog box

## 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.

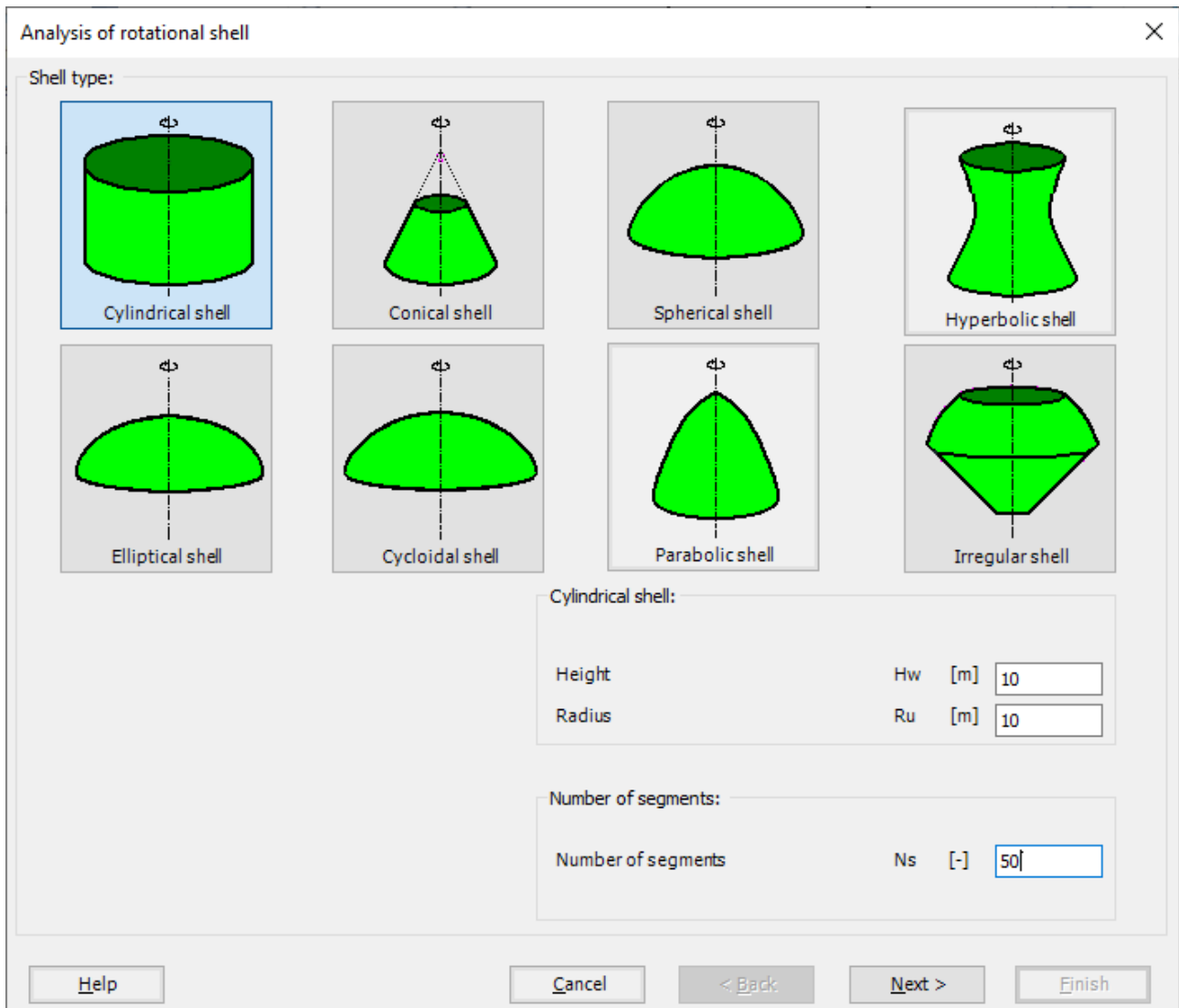


Figure 9.8 "Analysis of rotational shell" wizard with "Shell type" Form

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.

## Example 9

Analysis of rotational shell

Cylindrical shell:

Segment No. 1 from 53 segments:

Segment data:			
Start position	r1	[m]	10.0
	z1	[m]	0.0
End position	r2	[m]	10.0
	z2	[m]	0.2

In Table

Modify  
Refresh  
New  
Insert Segment  
Delete Segment  
Copy Segment

Help Cancel < Back **Next >** Finish

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]

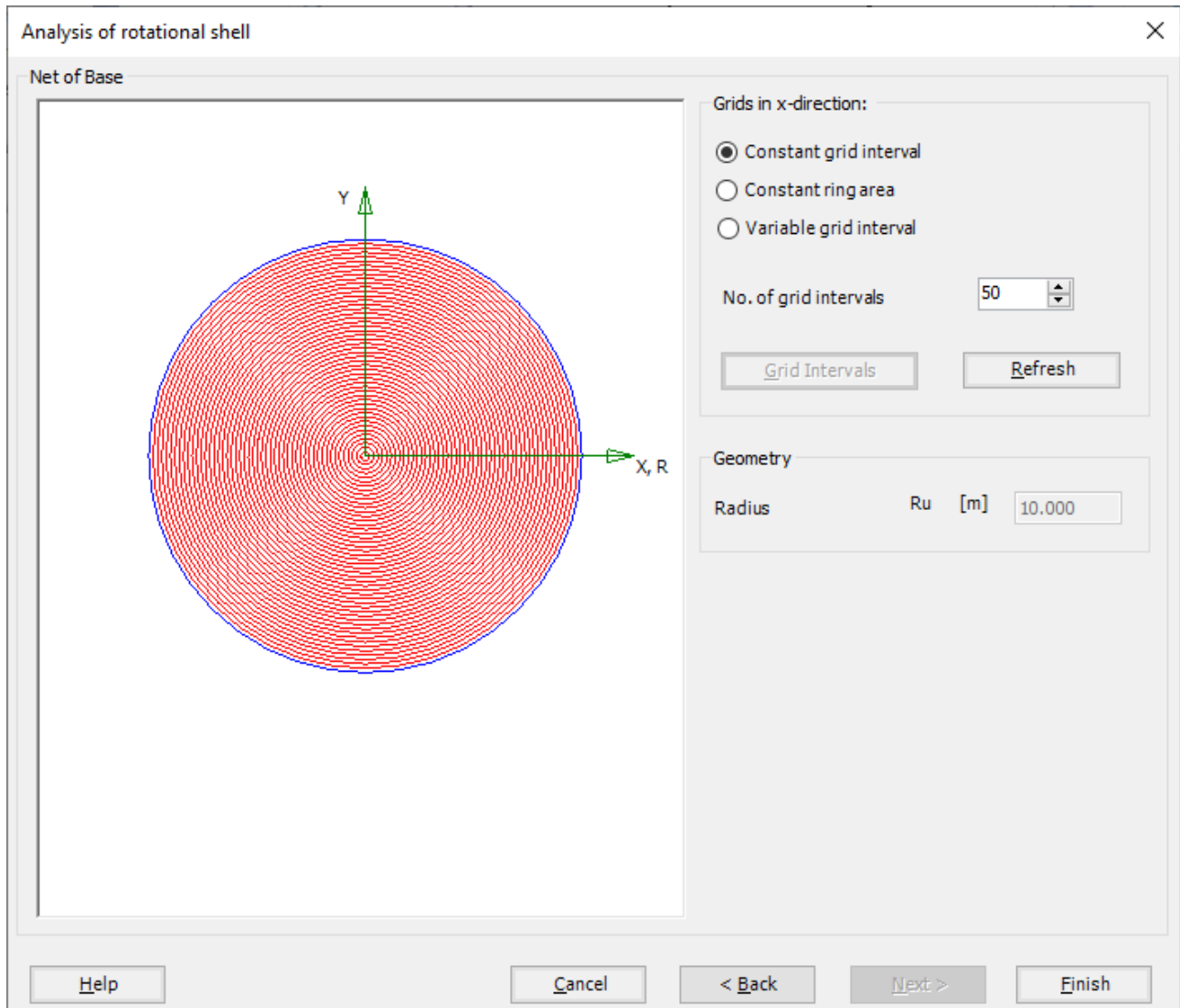


Figure 9.10 "Net of Base" Form

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

## Example 9

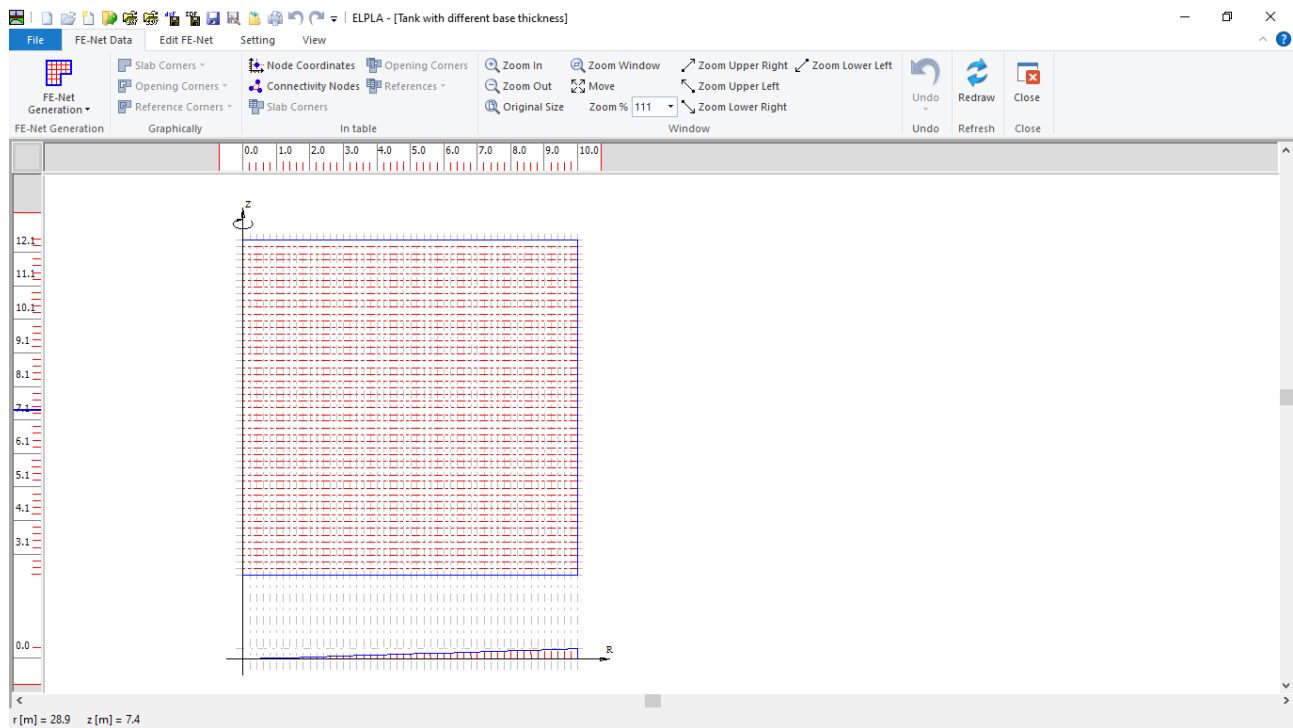


Figure 9.11 Generated FE-Net

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.

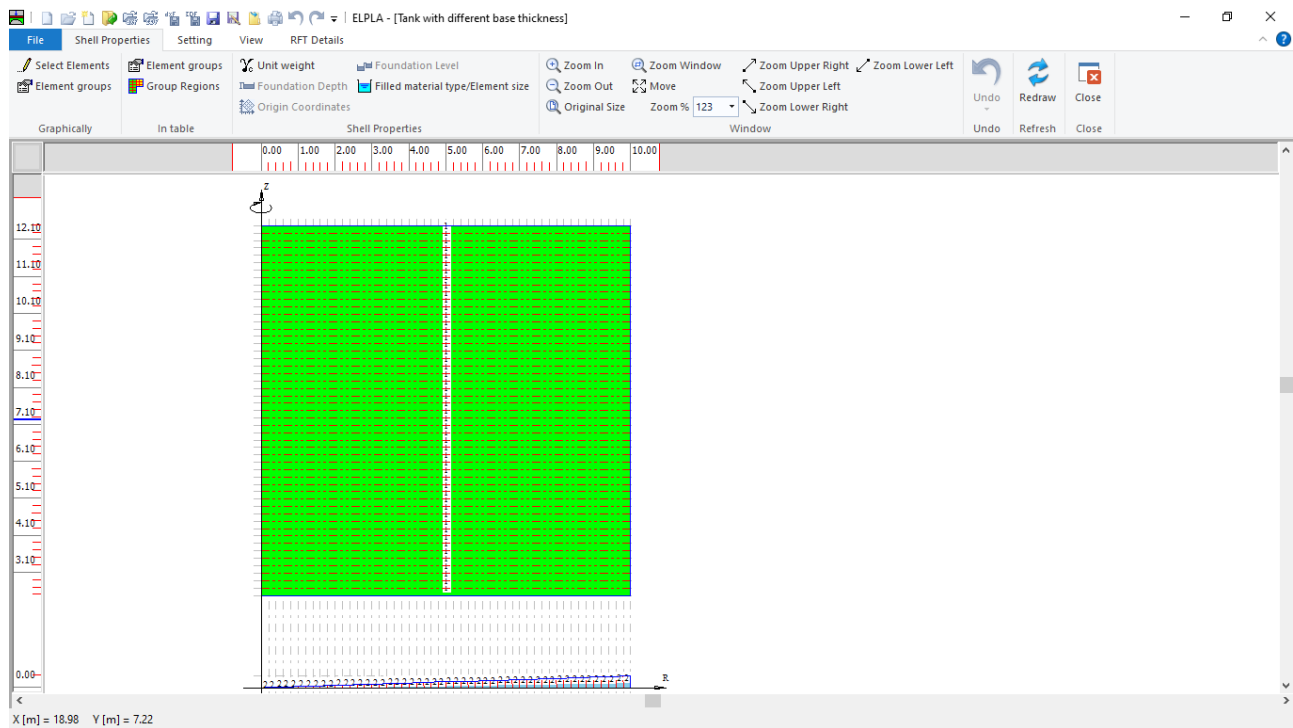


Figure 9.12 "Shell Properties" Window

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.

## Example 9

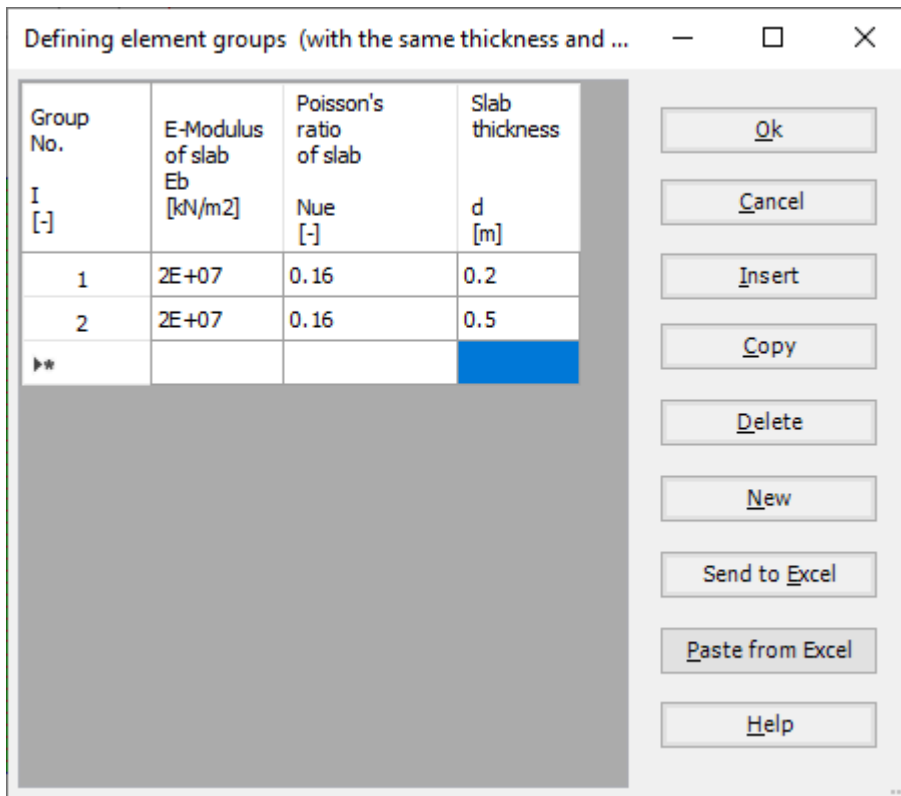


Figure 9.13 "Defining element groups" list box

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

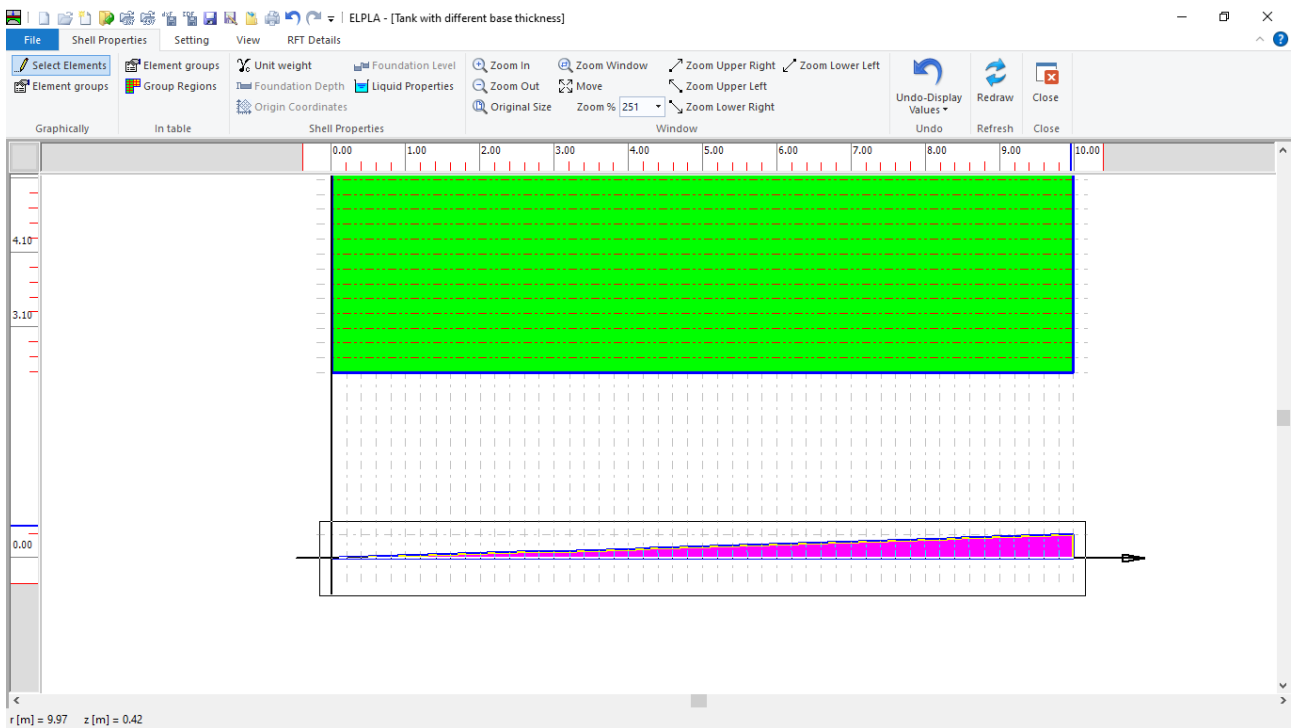


Figure 9.14 Selecting the nodes that include the tank base

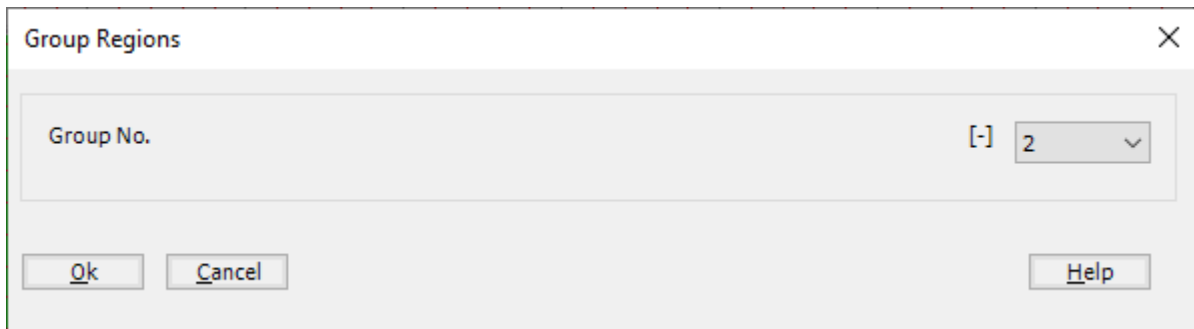


Figure 9.15 "Group Regions" dialog box

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<sup>3</sup>] appears, type 24 in the "Unit Weight" edit box, then click "OK" button.

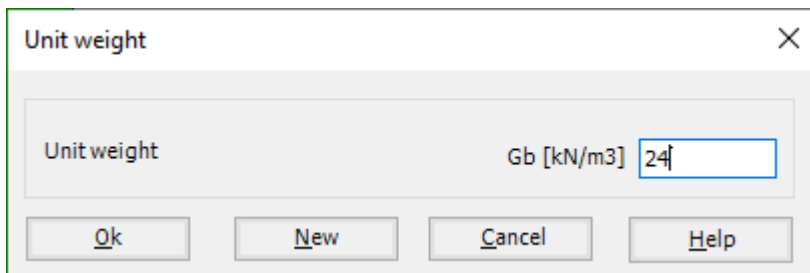


Figure 9.16 "Unit weight" dialog box



## Example 9

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.

Filled material type/Element size			
Filled material type:			
<input type="radio"/>	Empty container		
<input checked="" type="radio"/>	Liquid container		
<input type="radio"/>	Granular material container		
Liquid Properties:			
Height of the liquid	Hl	[m]	10
Unit weight of the liquid	Yw	[kN/m3]	10.19
Granular material properties:			
Top height of the granular material	H1	[m]	0.00
Bottom height of the granular material	H2	[m]	0.00
Unit weight of the granular material	Ys	[kN/m3]	15.50
Angle of internal friction of the granular material	φ	[°]	25
Angle of the wall friction	δ	[°]	20
Element size:			
<input checked="" type="checkbox"/>	Constant element sizes in z-direction		
Element size in each shell segment	Dl	[m]	1

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

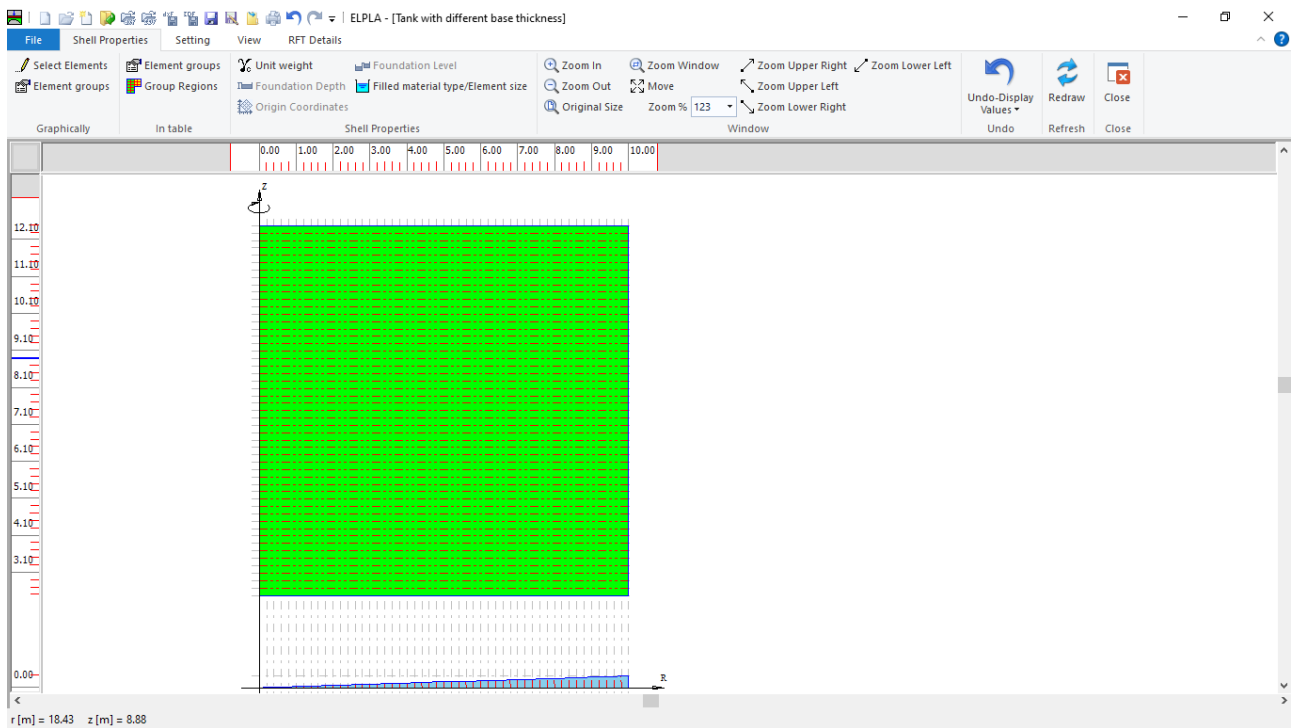


Figure 9.18 "Shell Properties" window after defining the shell data

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

## Example 9

### 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 <sup>2</sup> ]
Unit weight of the soil	$GAM$	= 18	[kN/m <sup>3</sup> ]
Angle of internal friction	$FHI$	= 30	[°]
Cohesion of the soil	$c$	= 0	[kN/m <sup>2</sup> ]
Poisson's ratio of the soil medium	$\nu_s$	= 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.

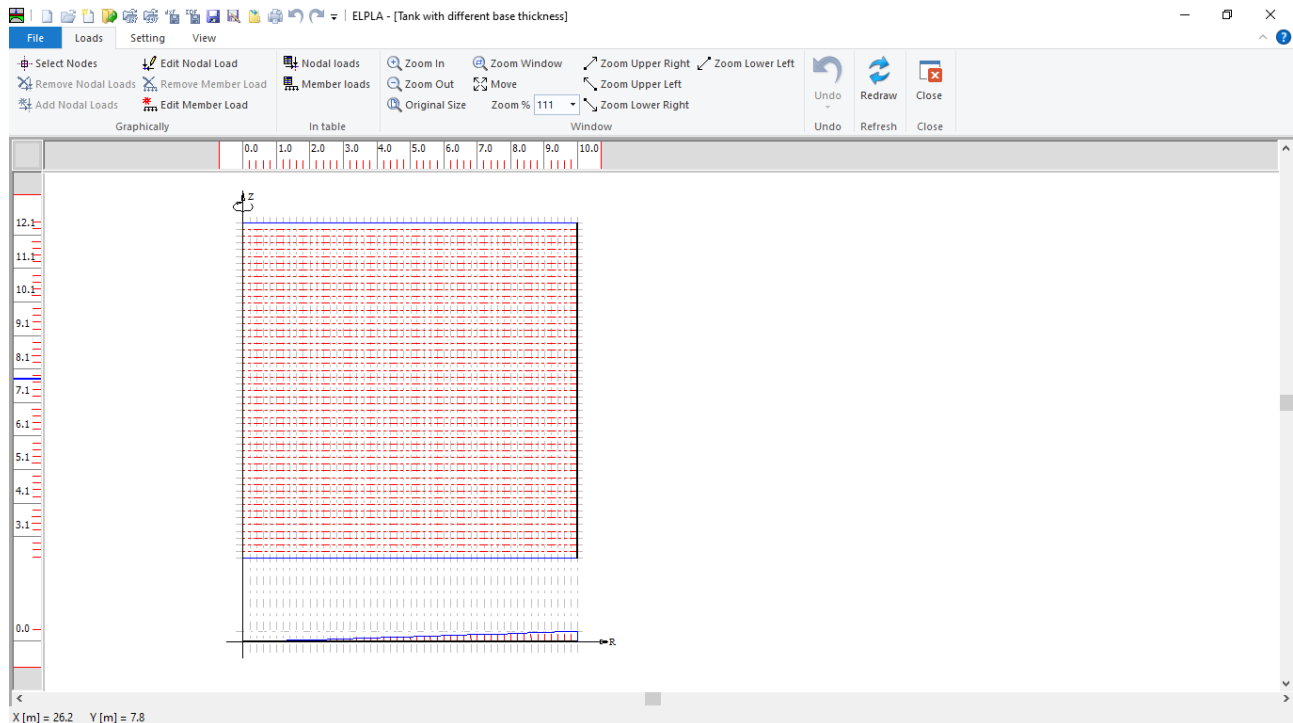


Figure 9.20 "Loads" Window

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.

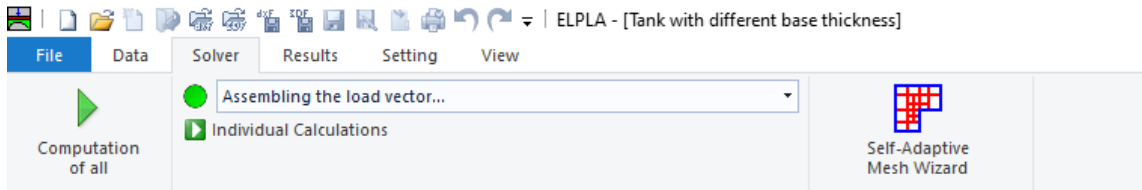


Figure 9.21 "Solver" Tab

*ELPLA* will activate 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

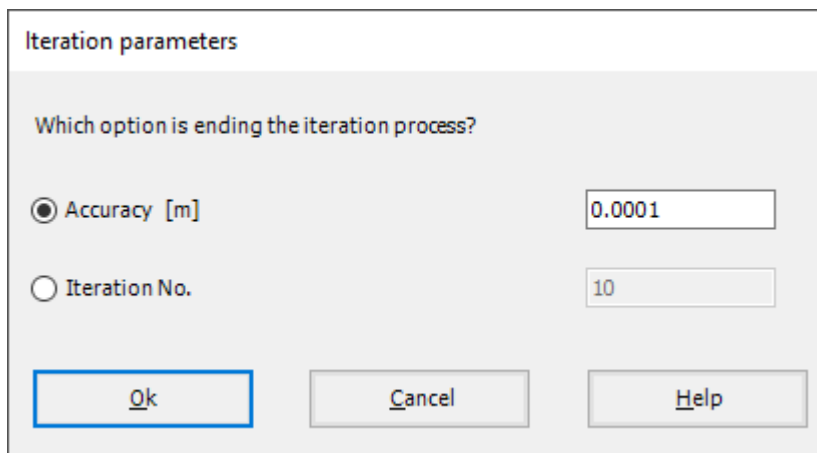


Figure 9.22 "Iteration parameters" option box

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.

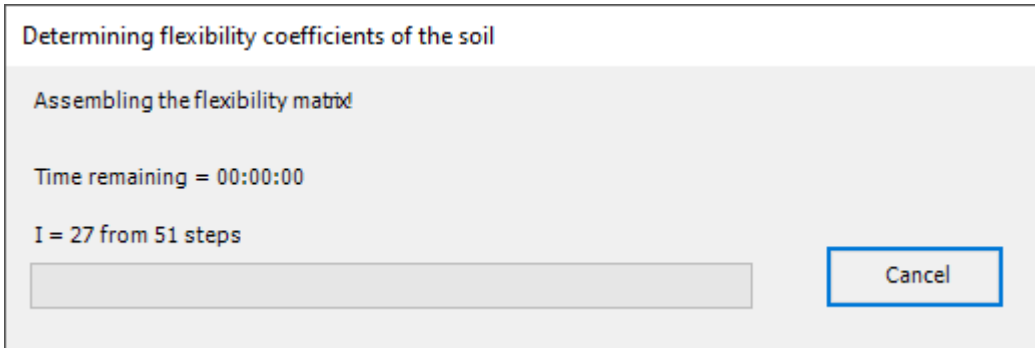


Figure 9.23 Analysis progress menu

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

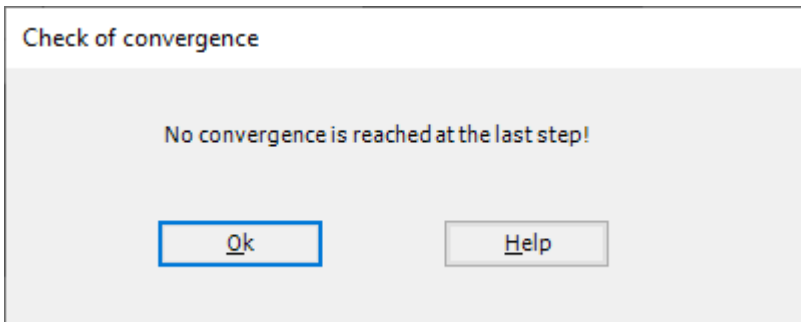


Figure 9.24 "Check of convergence" message

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

## Example 9

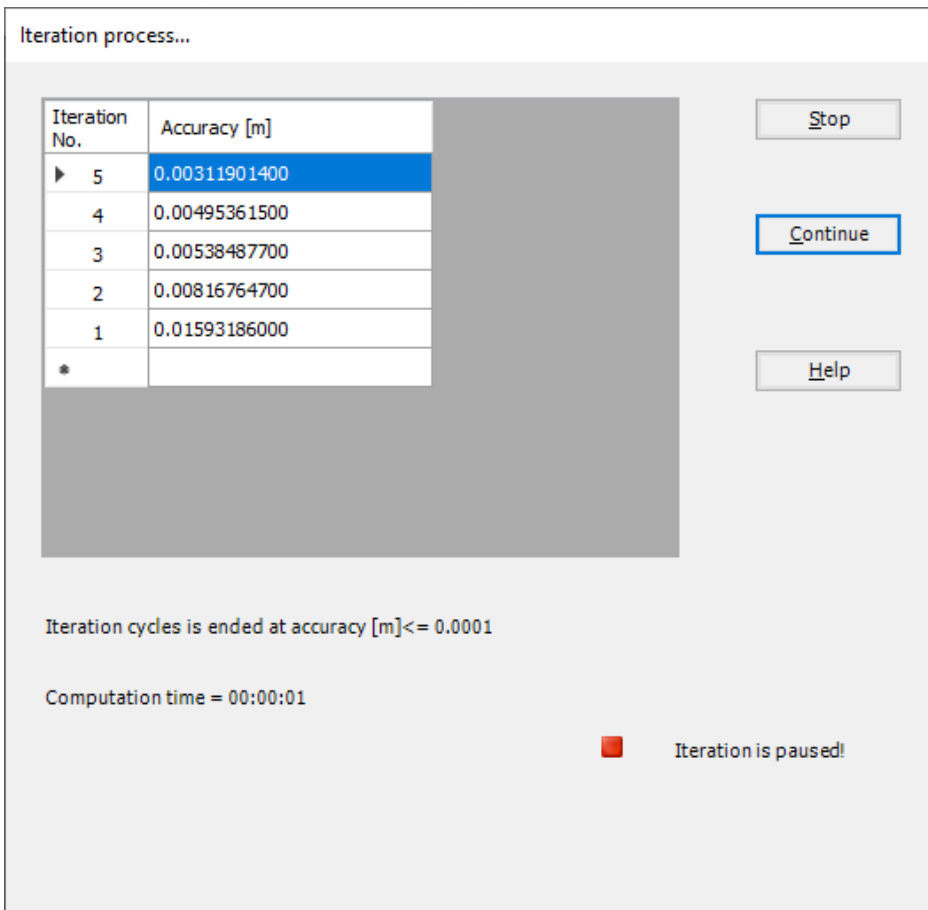


Figure 9.25 "Iteration process" list box

### 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.

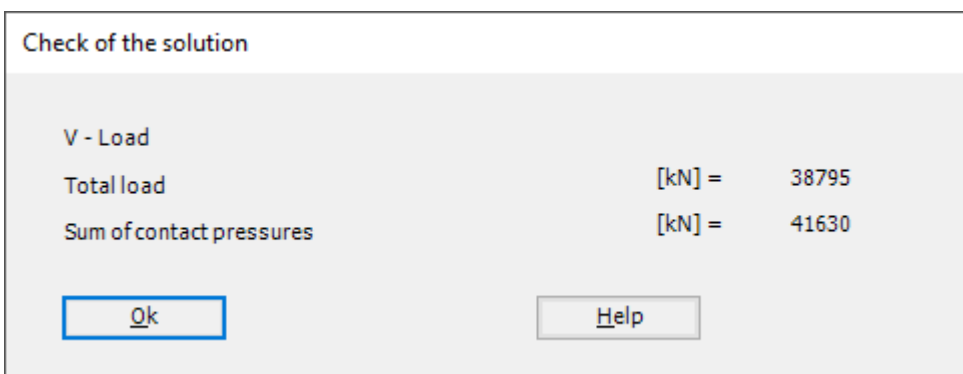


Figure 9.26 Menu "Check of the solution"

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.



## Example 9

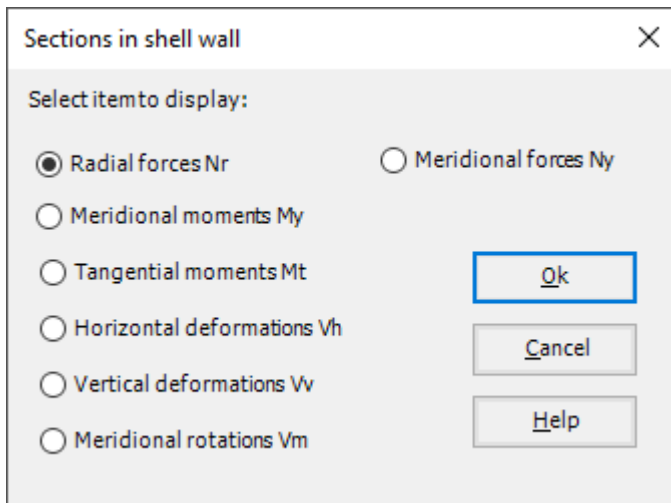


Figure 9.28 "Sections in shell wall" option box

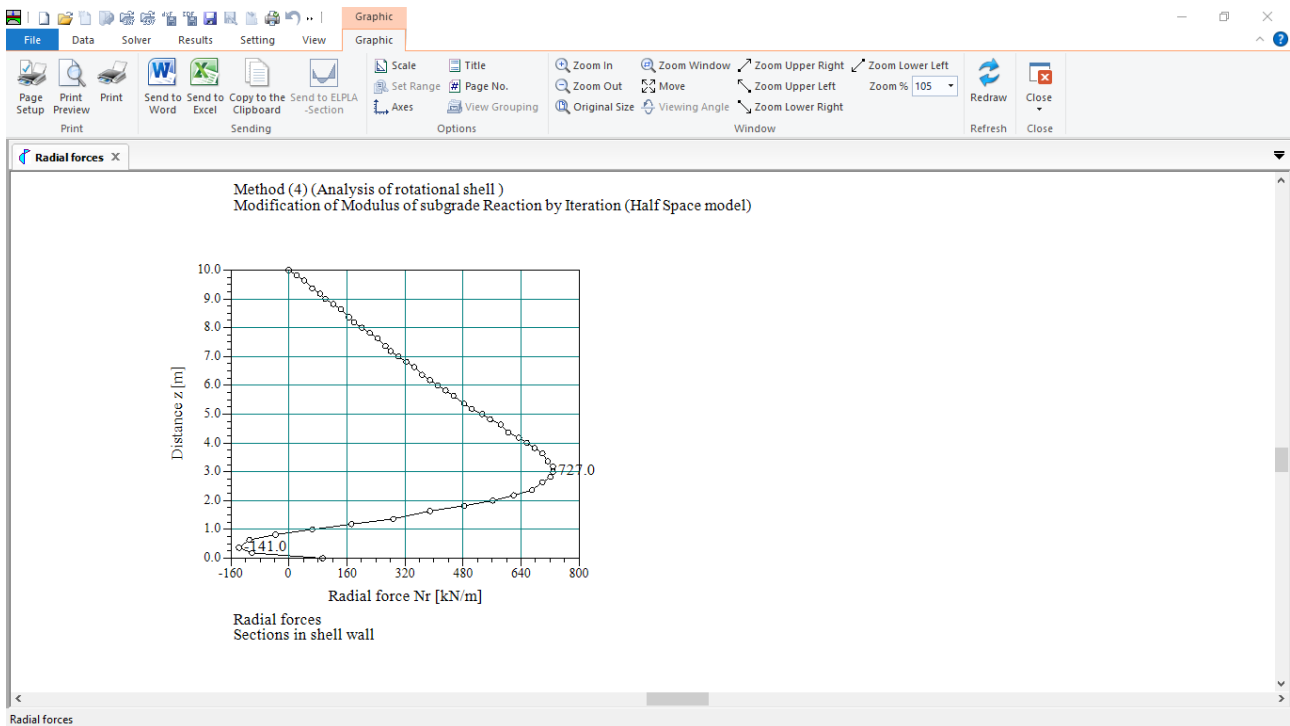


Figure 9.29 Radial forces in shell wall

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  $M_y$ " as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 9.31.

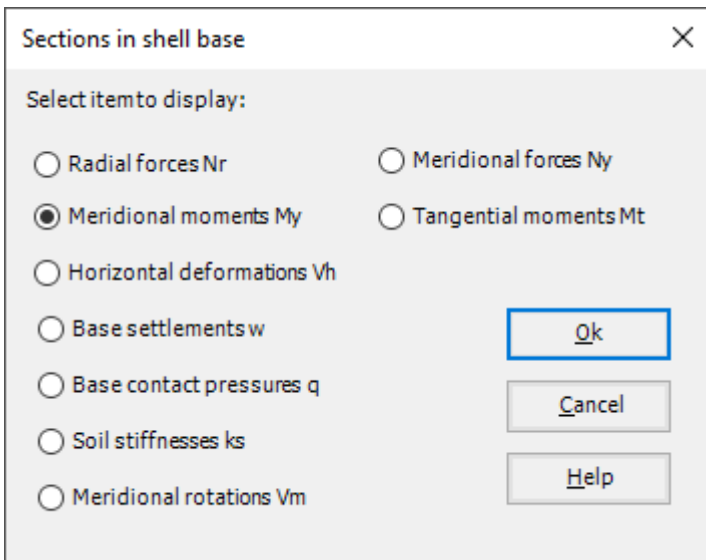


Figure 9.30 "Sections in shell base" option box

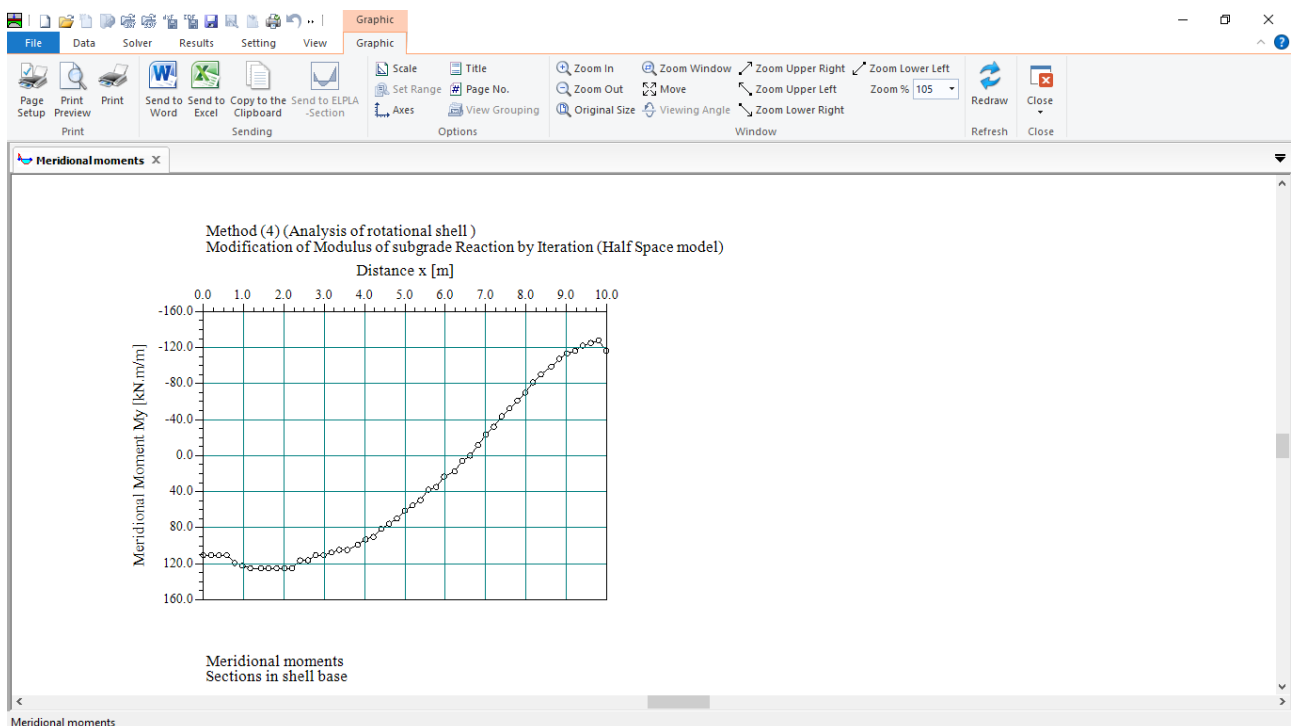


Figure 9.31 Meridional moments in shell base

## Example 9

---

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

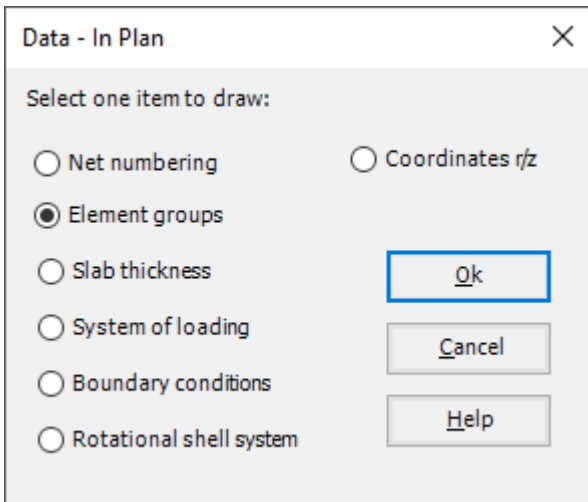


Figure 9.32 "Data – In Plan" option box

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  $M_y$ " check box
- The user can choose any other data to be displayed
- Click "OK" button

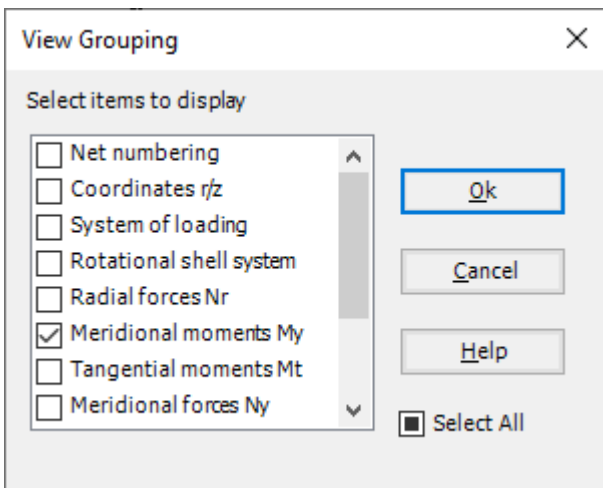


Figure 9.33 "View Grouping" check group box

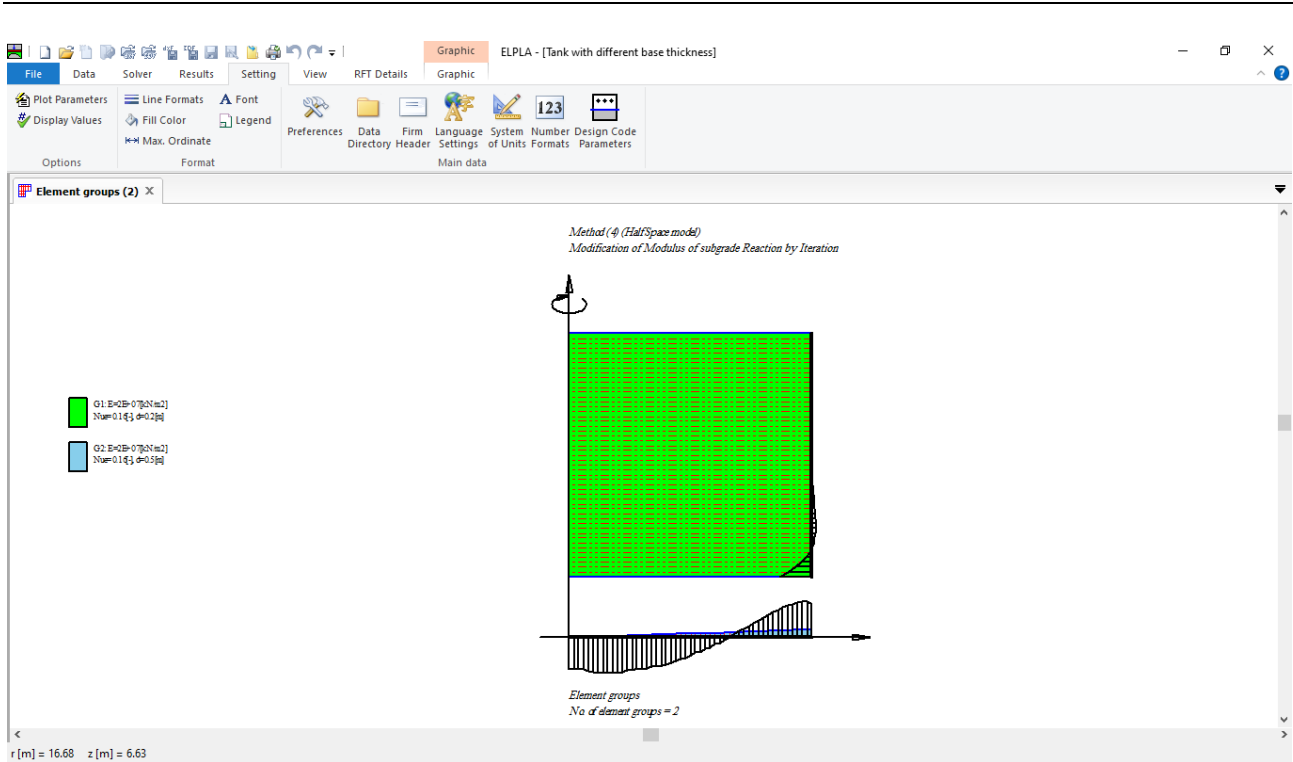


Figure 9.34 Element groups of the tank with the meridional moments