

Example 10

**Analysis of a cylindrical
water container with a conical base**

Contents	Page
1 Description of the problem	3
2 Container geometry and properties	3
3 Numerical Analysis	4
4 Creating the project	5
4.1 Calculation method	5
4.2 Project identification	8
4.3 FE-Net data	8
4.4 Shell properties	12
4.5 Supports/ boundary conditions	18
4.6 Loads	22
5 Carrying out the calculations	23
6 Viewing data and results	25

Example 10

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]

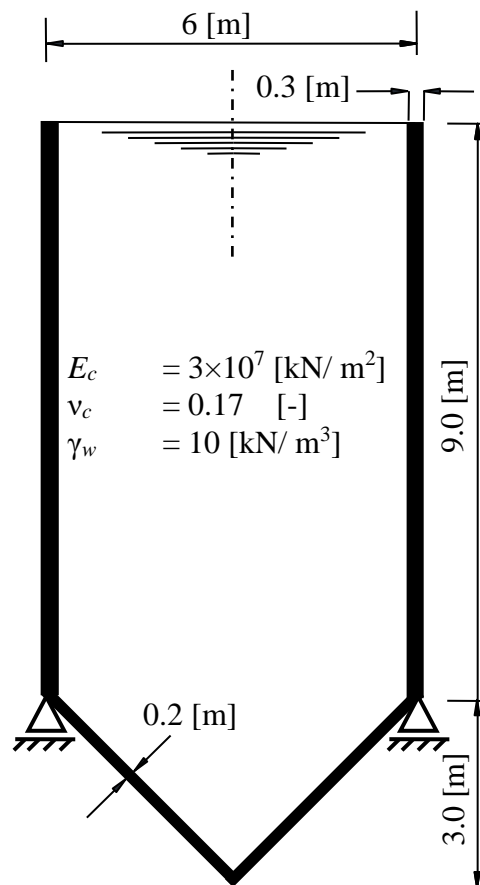


Figure 10.1 Cylindrical water container with dimensions and supports

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.

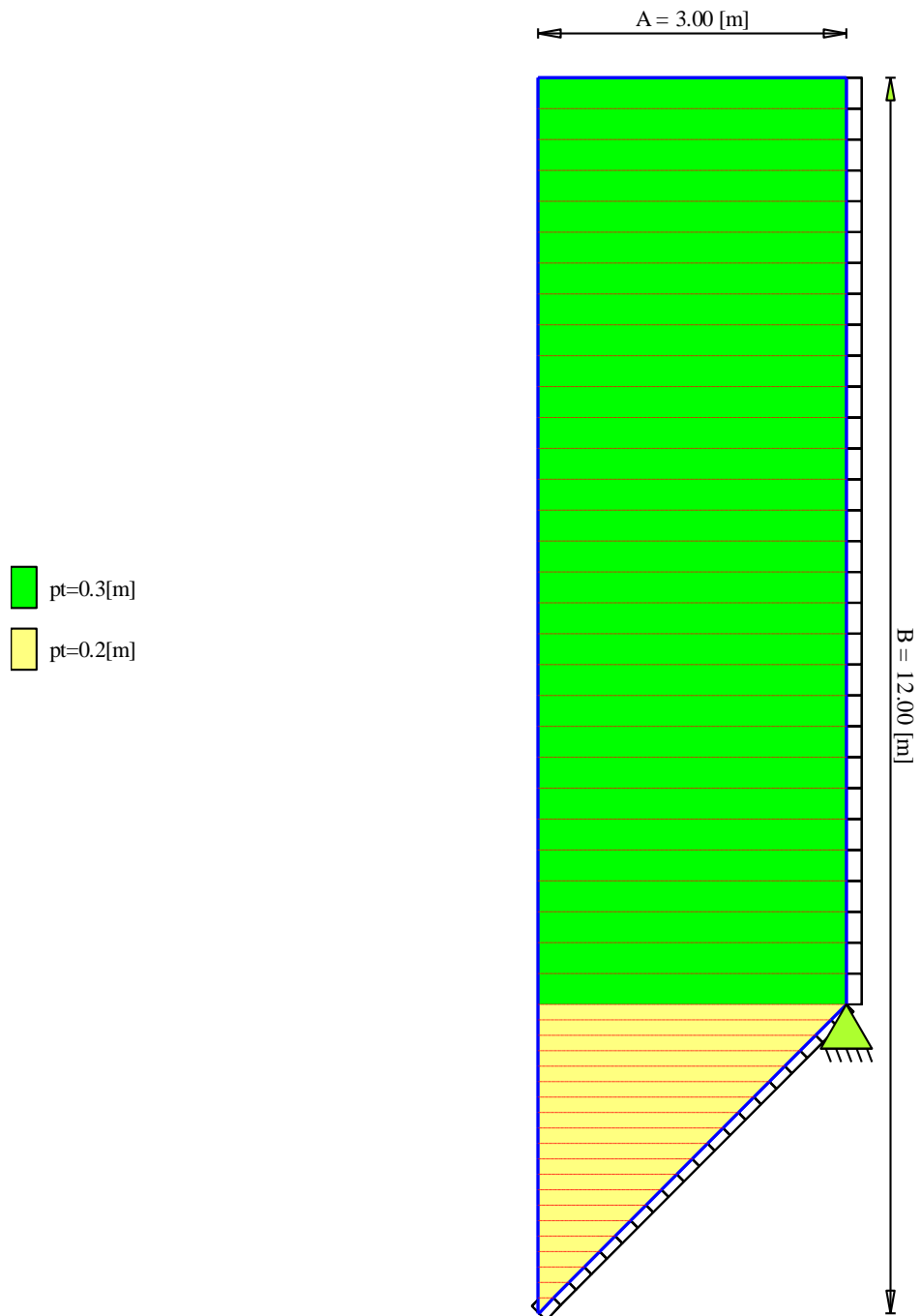


Figure 10.2 Finite element mesh of the container with boundary condition

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).

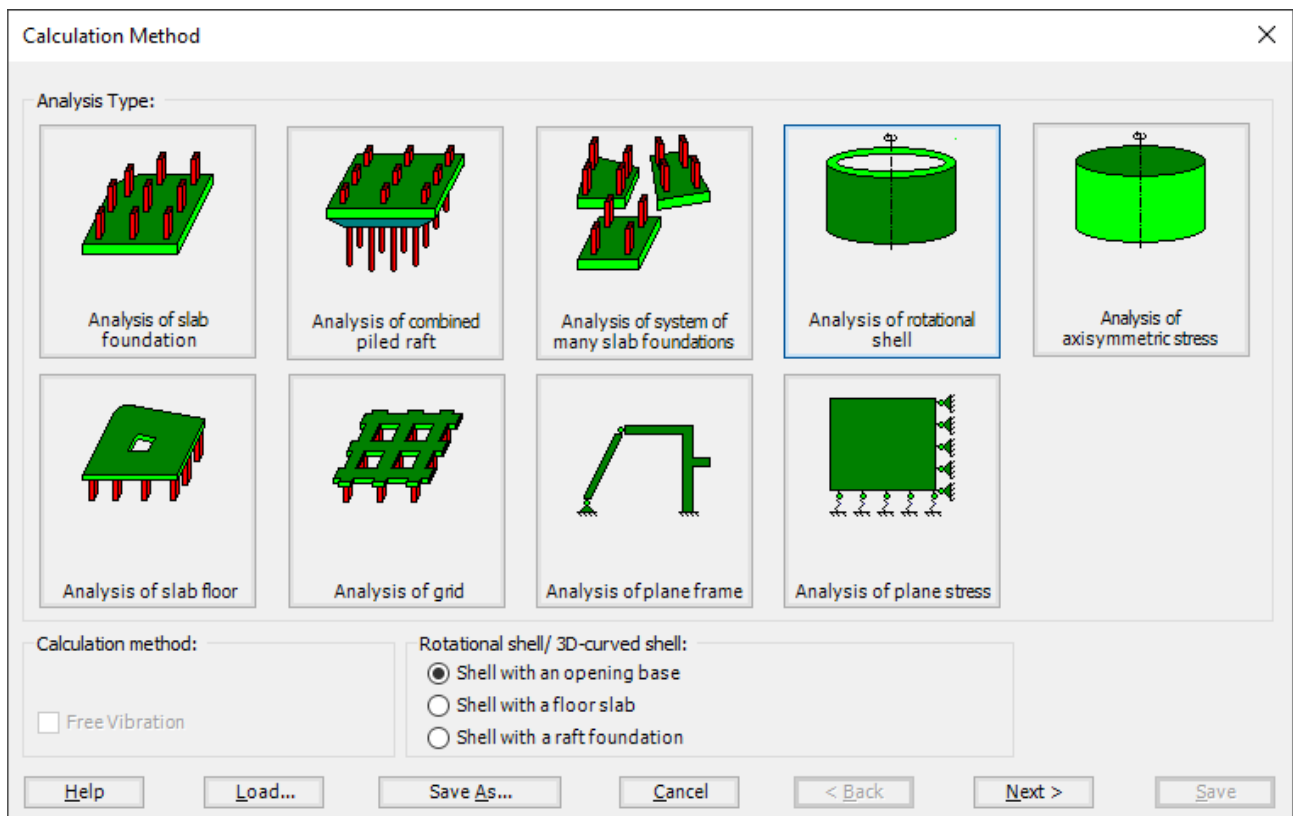


Figure 10.3 "Analysis Type" Form

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.

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 the 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 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.

Example 10

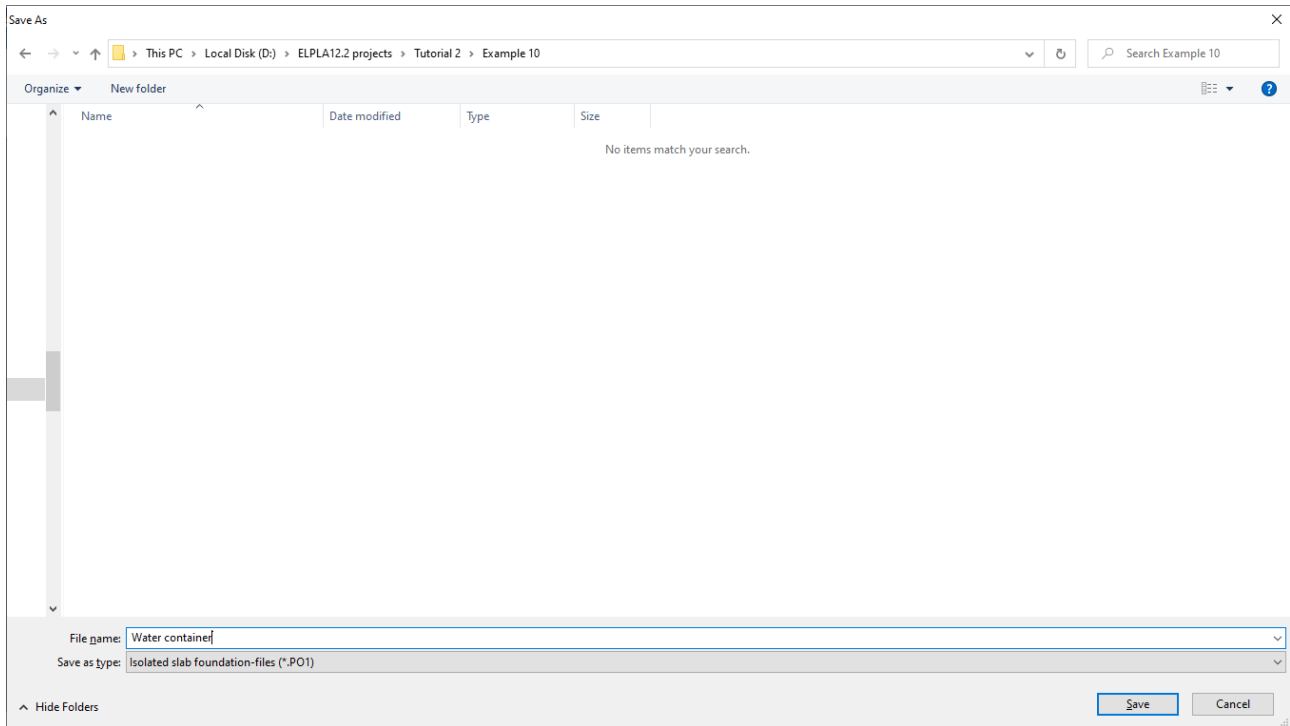


Figure 10.5 "Save as" dialog box

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

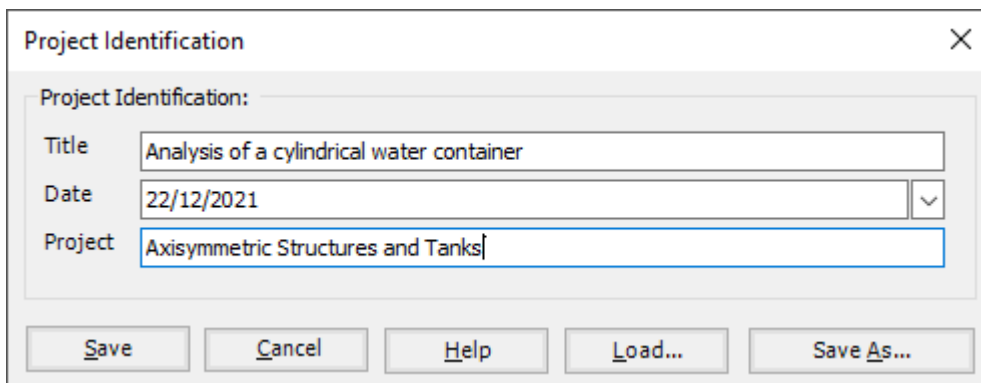


Figure 10.6 "Project Identification" dialog box

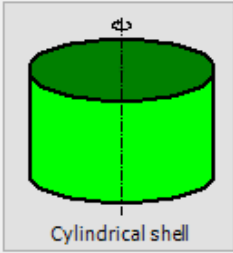
4.3 FE-Net data

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.

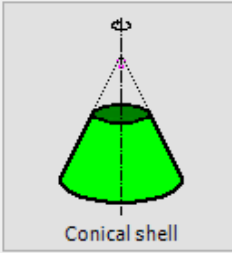
Example 10

Analysis of rotational shell

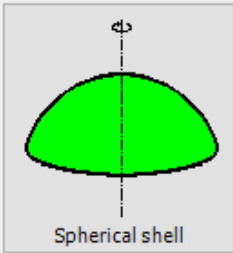
Shell type:



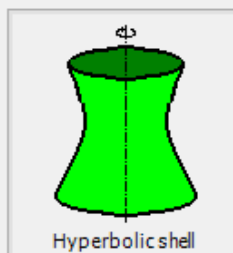
Cylindrical shell



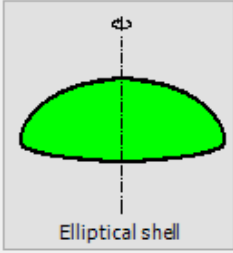
Conical shell



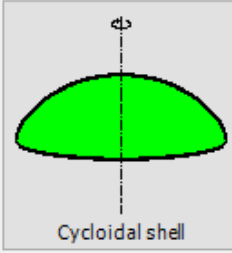
Spherical shell



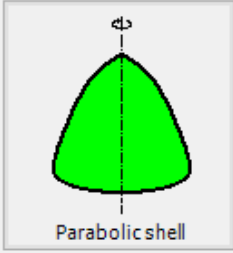
Hyperbolic shell



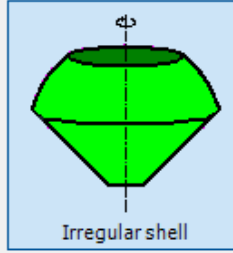
Elliptical shell



Cycloidal shell



Parabolic shell



Irregular shell

Irregular shell

No. I [-]	Height Hw [m]	Upper radius Ro [m]	Number of segments Ns I [-]
▶ 1	3.00	3.00	20
2	9.00	3.00	30
*			

Linear segments Curved segments

Tank base:
 Radius [m]

Roof data:

Shell with covered roof

Spherical roof

Flat roof

Radius of the spherical roof Ra [m]

Angle of the spherical roof Theta [°]

Number of roof segments Nsa [°]

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 R_b " 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 H_w " edit box
- Type 3 in the "Upper radius R_o " edit box
- Type 20 in the "Number of segments N_s " edit box

To identify the second main segment

- Type 9 in the "Height H_w " edit box
 - Type 3 in the "Upper radius R_o " edit box
 - Type 30 in the "Number of segments N_s " 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.

Analysis of rotational shell

Irregular shell:

Segment No. 1 from 53 segments:

Segment data:

Start position	r1	[m]	0.01
	z1	[m]	0.00
End position	r2	[m]	0.16
	z2	[m]	0.15

In Table

Refresh

New

Insert Segment

Delete Segment

Copy Segment

Help Cancel < Back Next > Finish

Figure 10.8 "Irregular shell" Form

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

Example 10

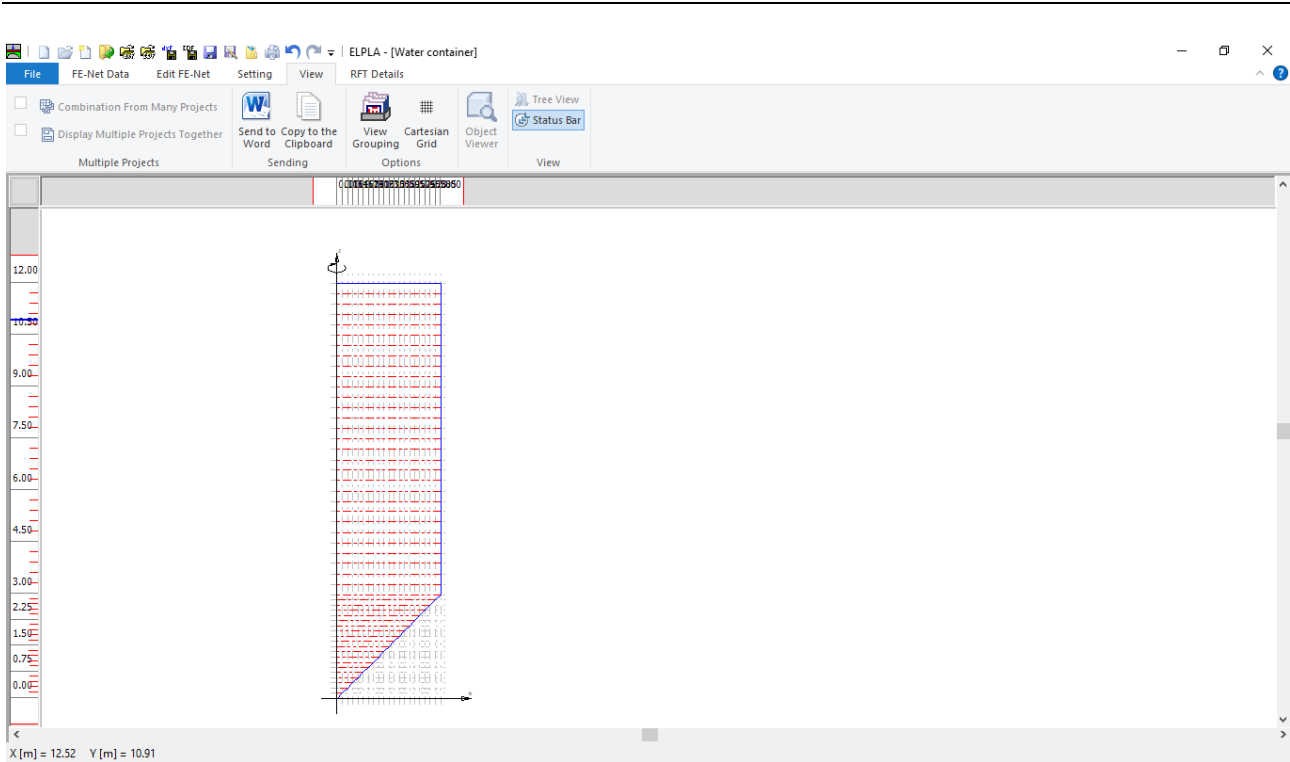


Figure 10.9 Generated FE-Net

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.

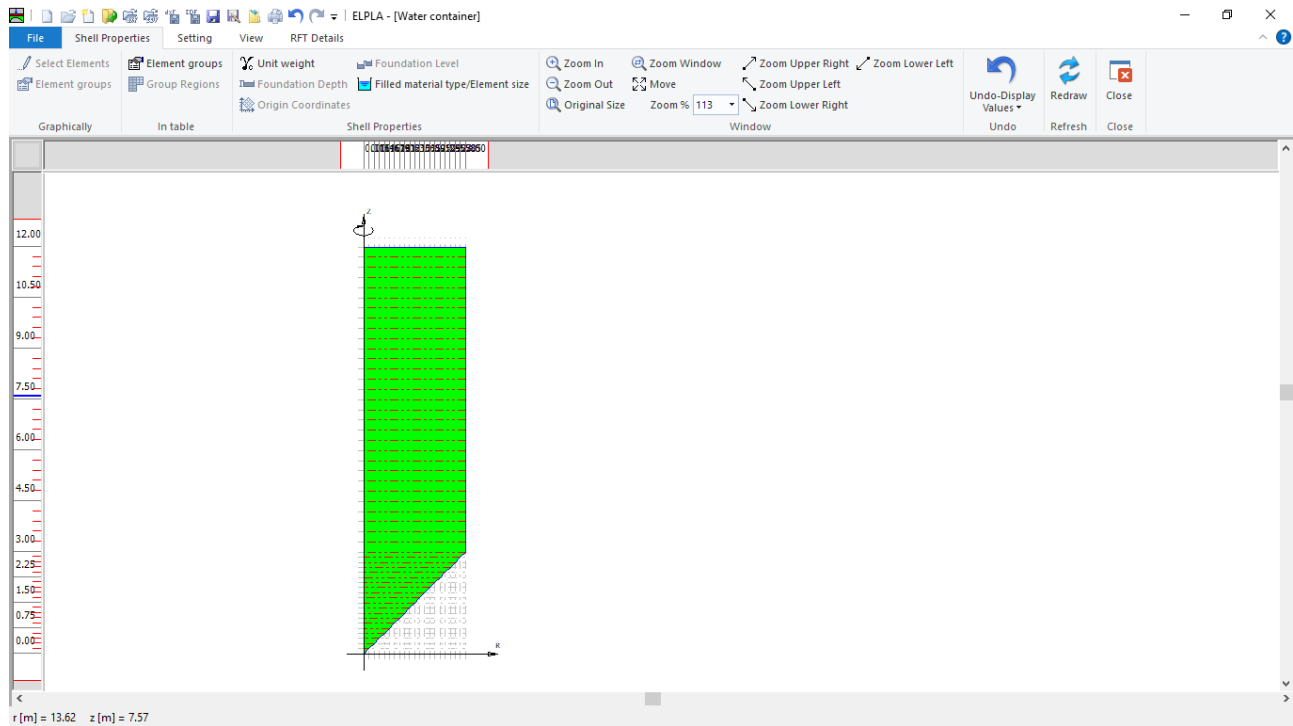


Figure 10.10 "Shell Properties" Window

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.

Example 10

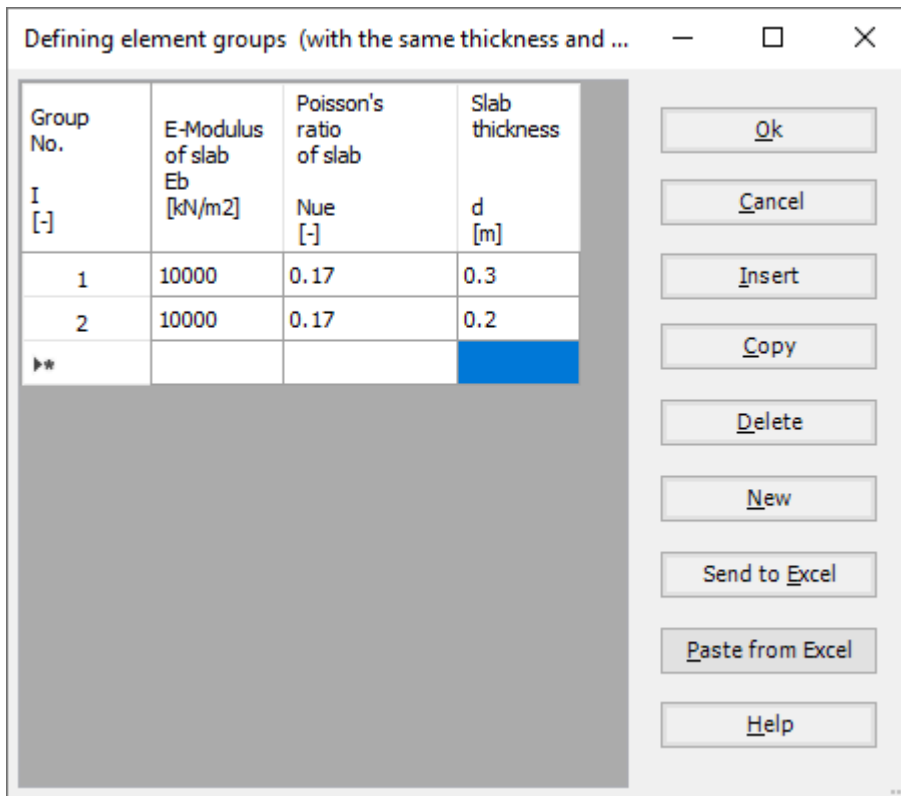


Figure 10.11 "Defining element groups" list box

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

Analyzing Axisymmetric Structures and Tanks by *ELPLA*

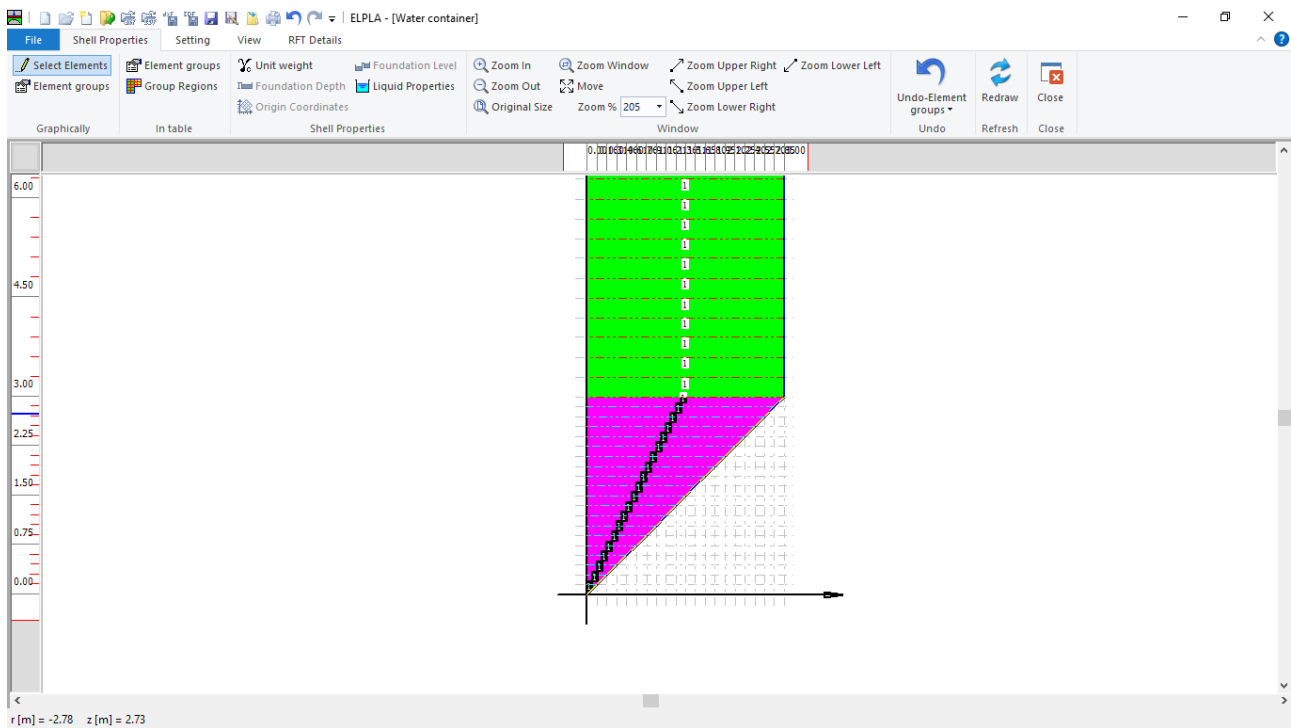


Figure 10.12 Selecting the elements that include the tank base

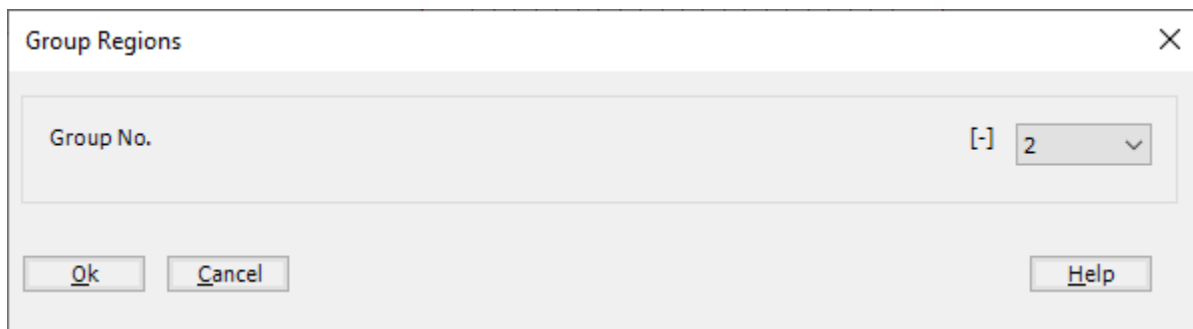


Figure 10.13 "Group Regions" dialog box

Example 10

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/m³] appears, type 0 in the "Unit weight" edit box to neglect the self-weight of the tank, then click "OK" button.

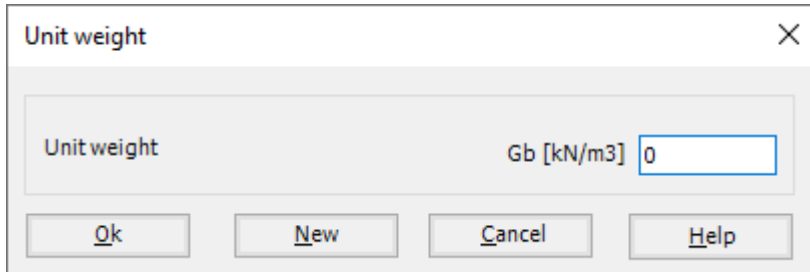


Figure 10.14 "Unit weight" dialog box

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.

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]	12
Unit weight of the liquid	Yw	[kN/m3]	10
Granular material properties:			
Top height of the granular material	H1	[m]	1.00
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	[°]	22
Element size:			
<input checked="" type="checkbox"/>	Constant element sizes in z-direction		
Element size in each shell segment	Dl	[m]	1

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

Example 10

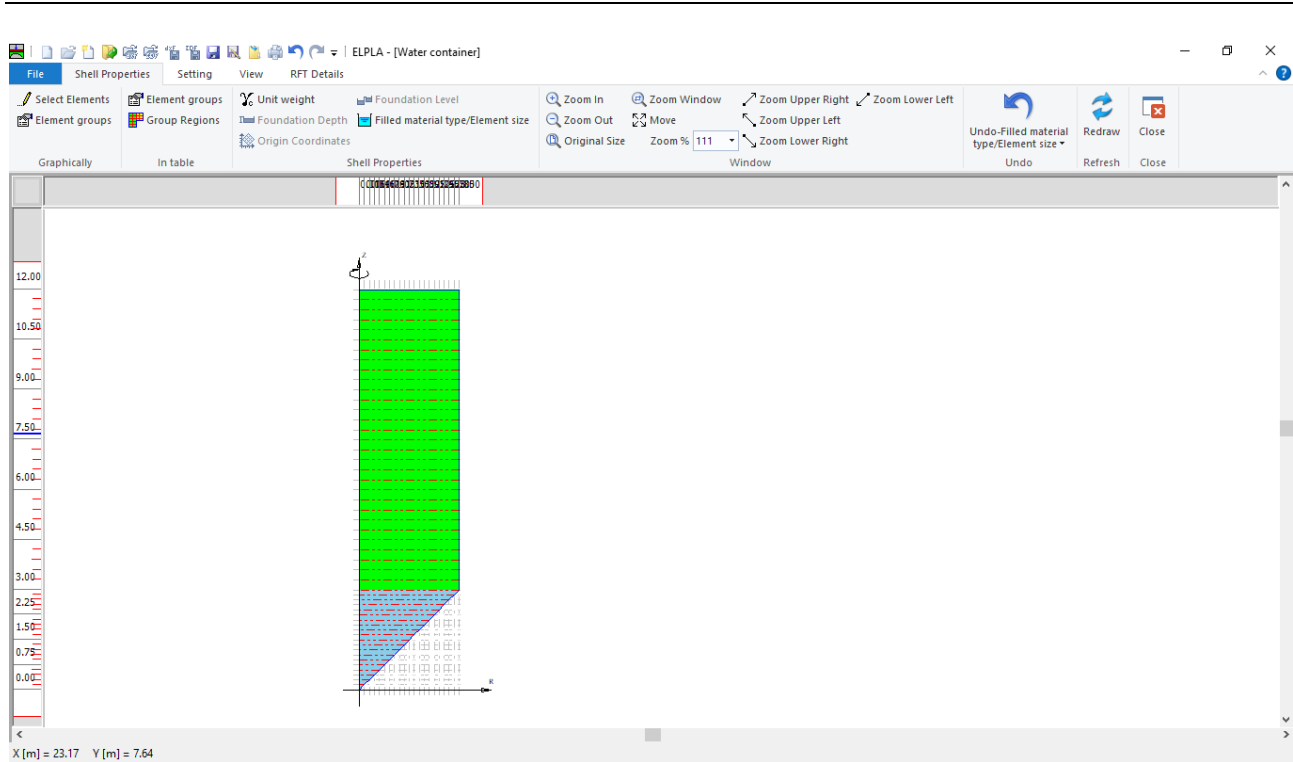


Figure 10.16 "Shell Properties" window after defining the container properties

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.

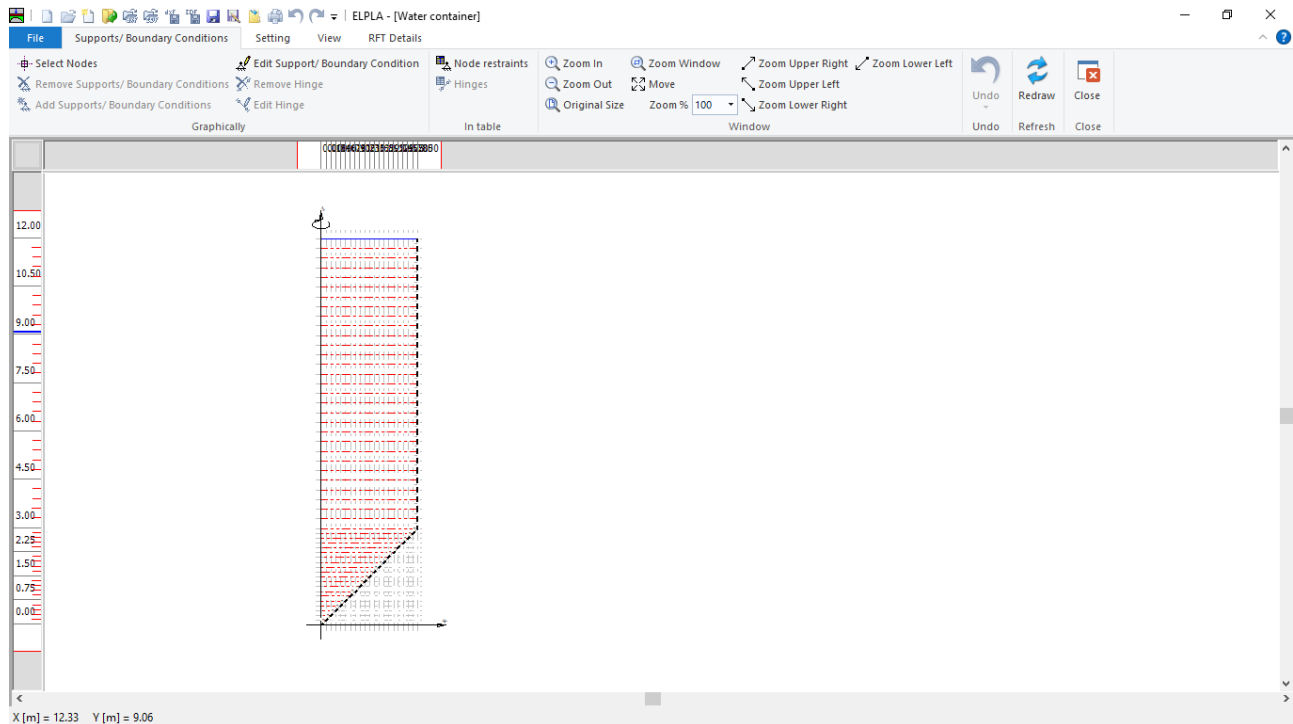


Figure 10.17 "Supports/ Boundary Conditions" Window

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.

Example 10

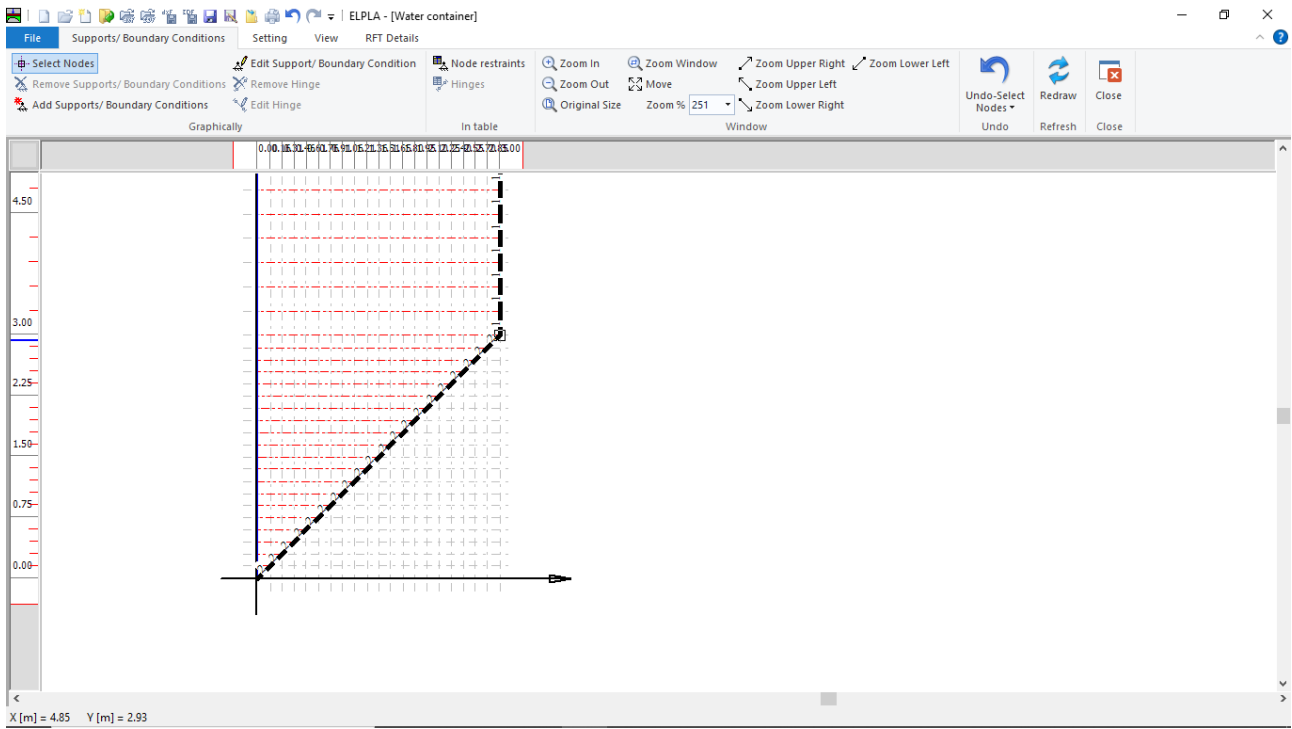


Figure 10.18 Selection of the node that has the support

In this dialog box

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

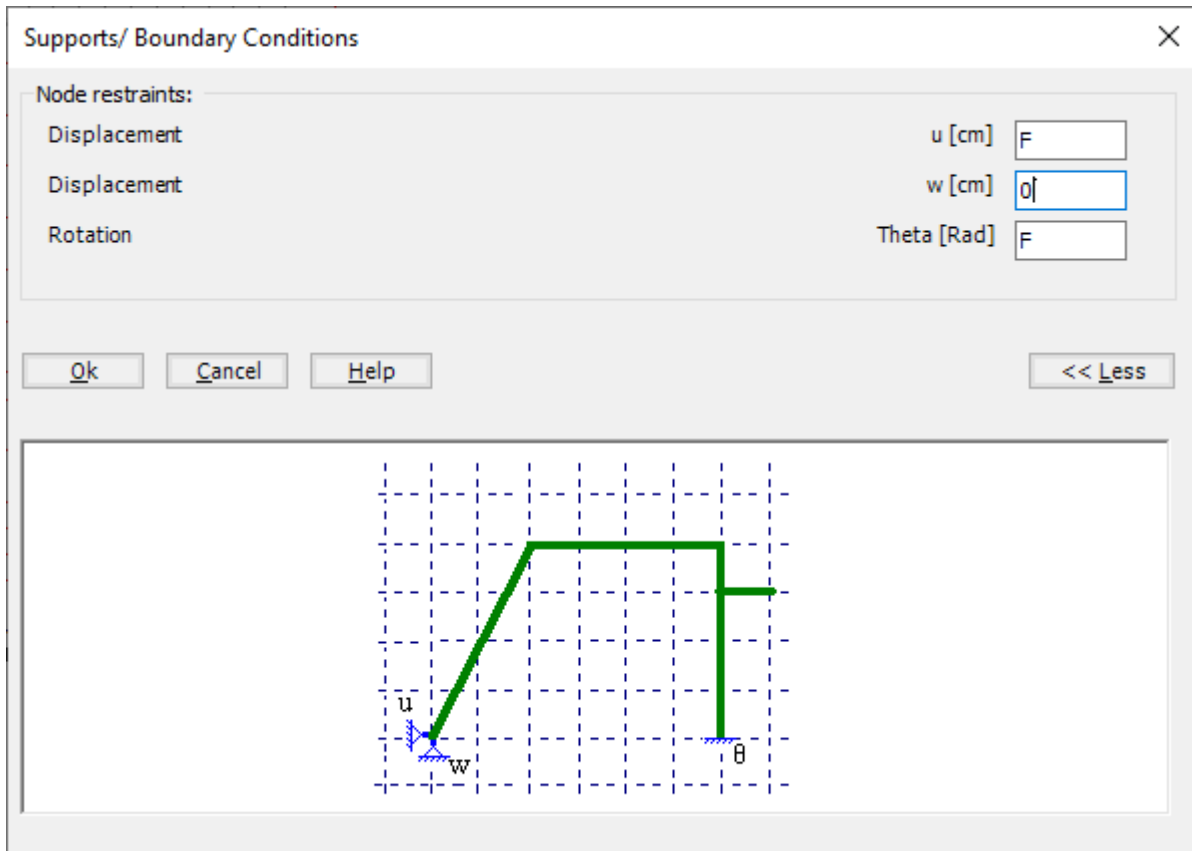


Figure 10.19 "Supports/ Boundary Conditions" dialog box

Example 10

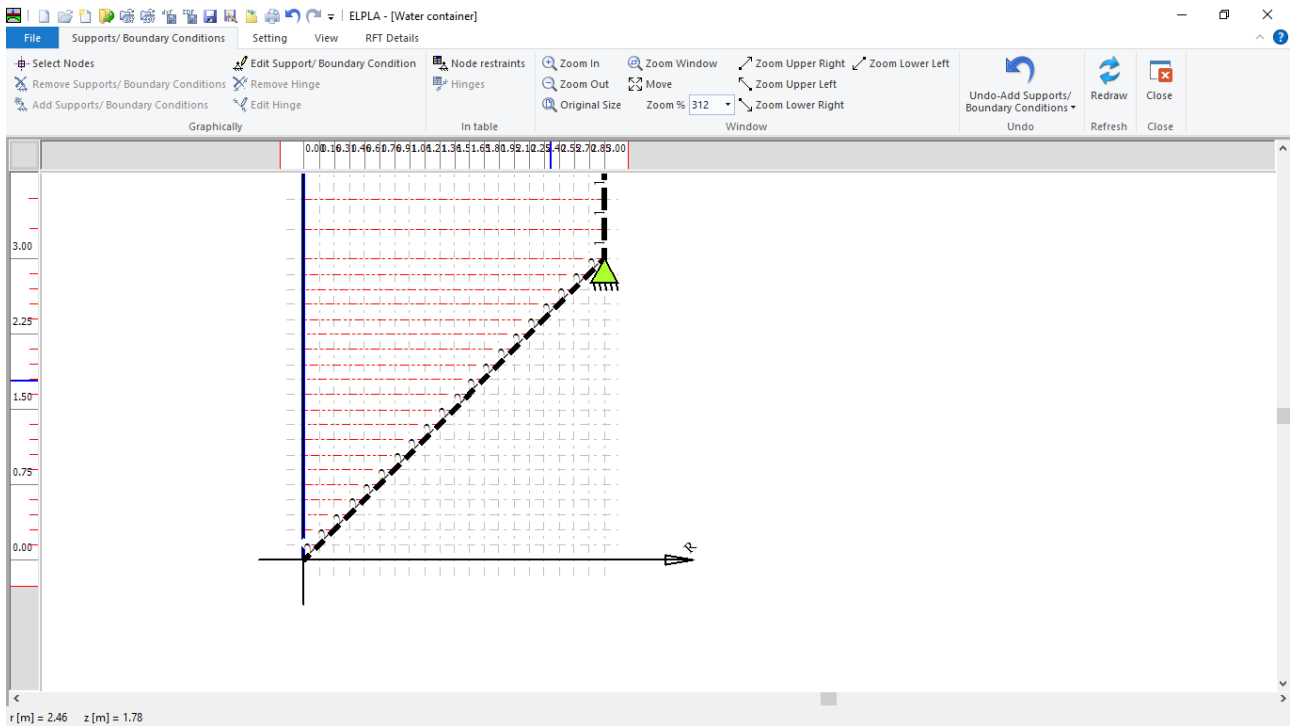


Figure 10.20 "Supports/ Boundary Conditions" window after defining the support

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.

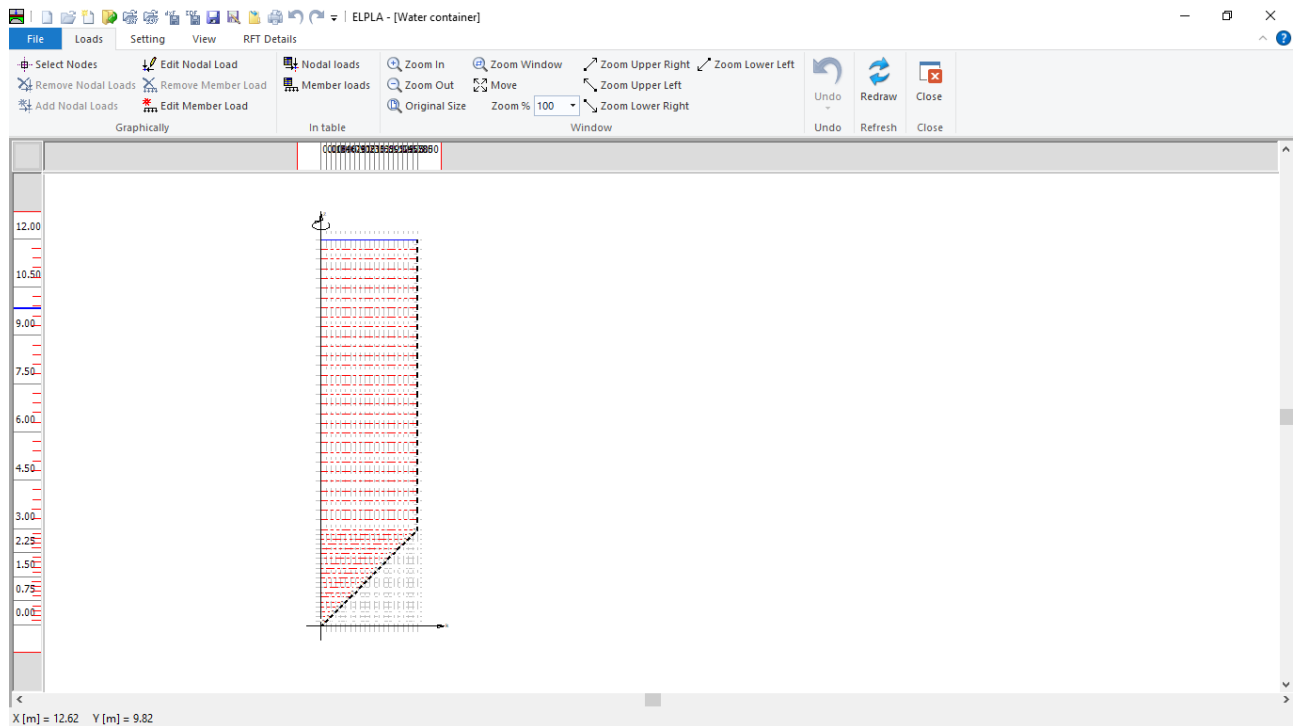


Figure 10.21 "Loads" Window

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.

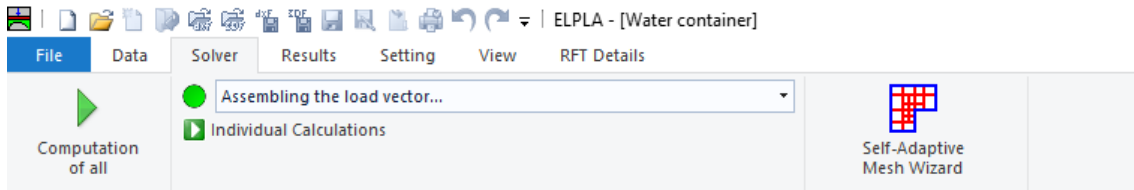


Figure 10.22 "Solver" Tab

ELPLA will 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
- 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.

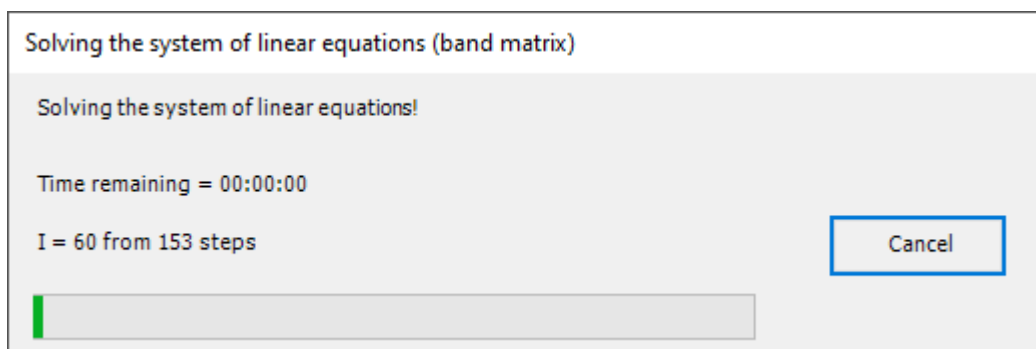


Figure 10.23 Analysis progress menu

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.

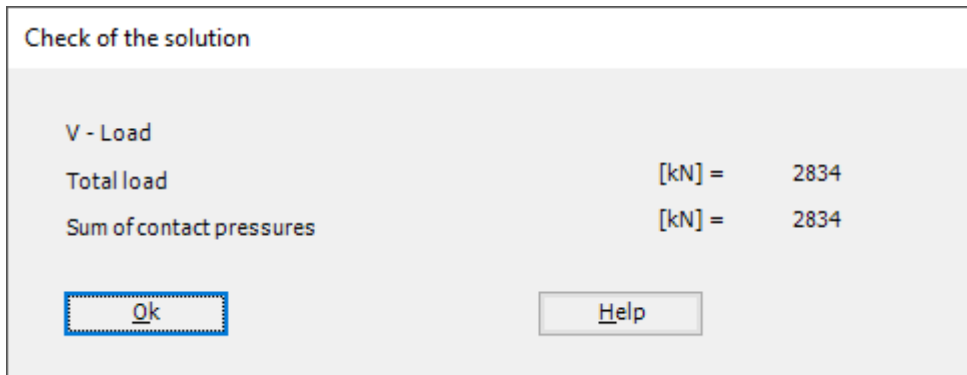


Figure 10.24 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

Example 10

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).

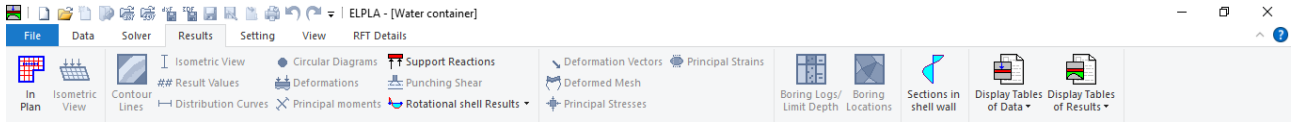


Figure 10.25 "Results" Tab

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

The Results are now displayed as shown in Figure 10.27.

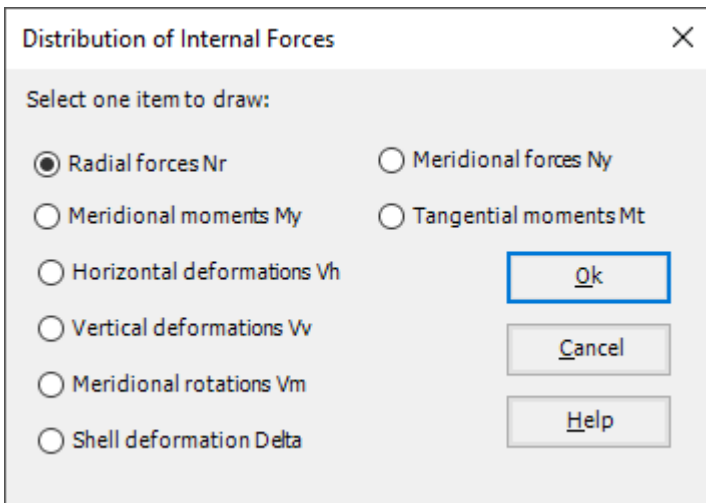


Figure 10.26 "Distribution of Internal Forces" option box

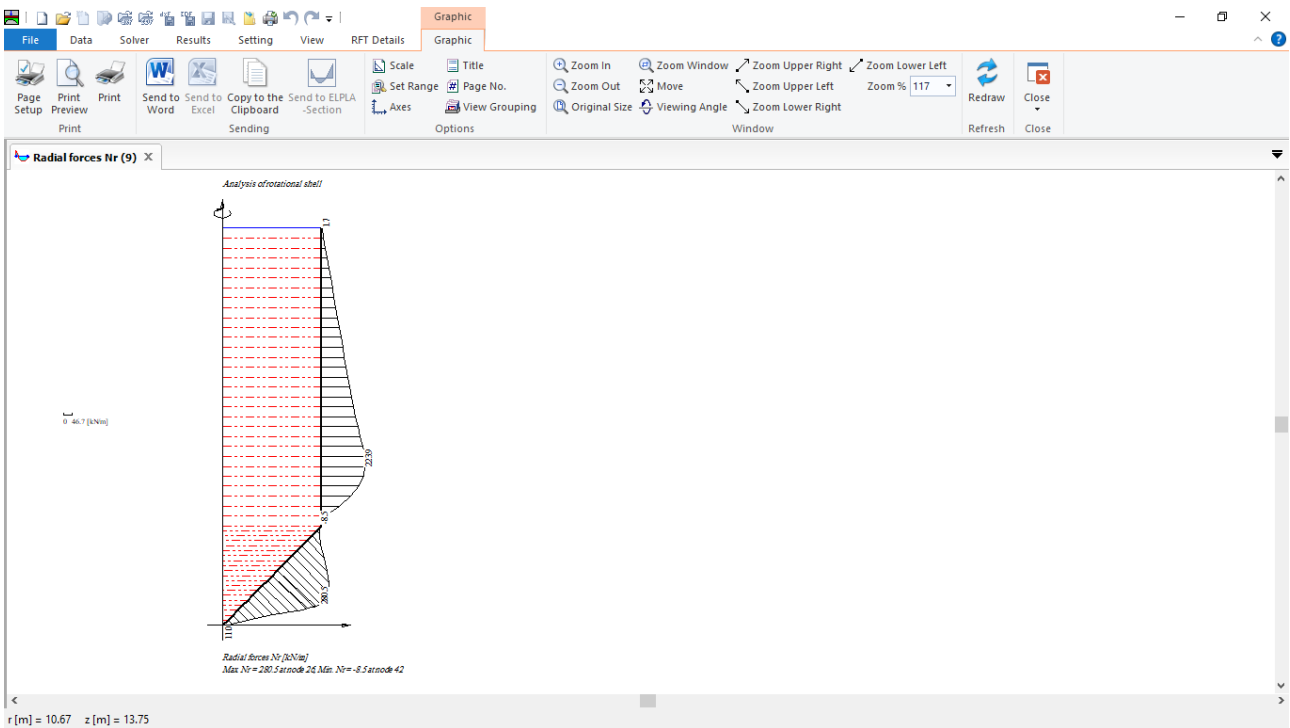


Figure 10.27 Radial forces on the shell wall

Example 10

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

The Results are now displayed as shown in Figure 10.29.

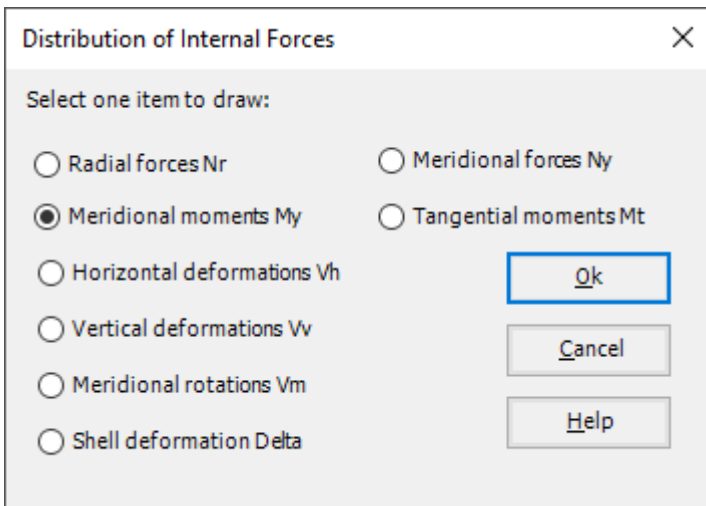


Figure 10.28 "Distribution of Internal Forces" option box

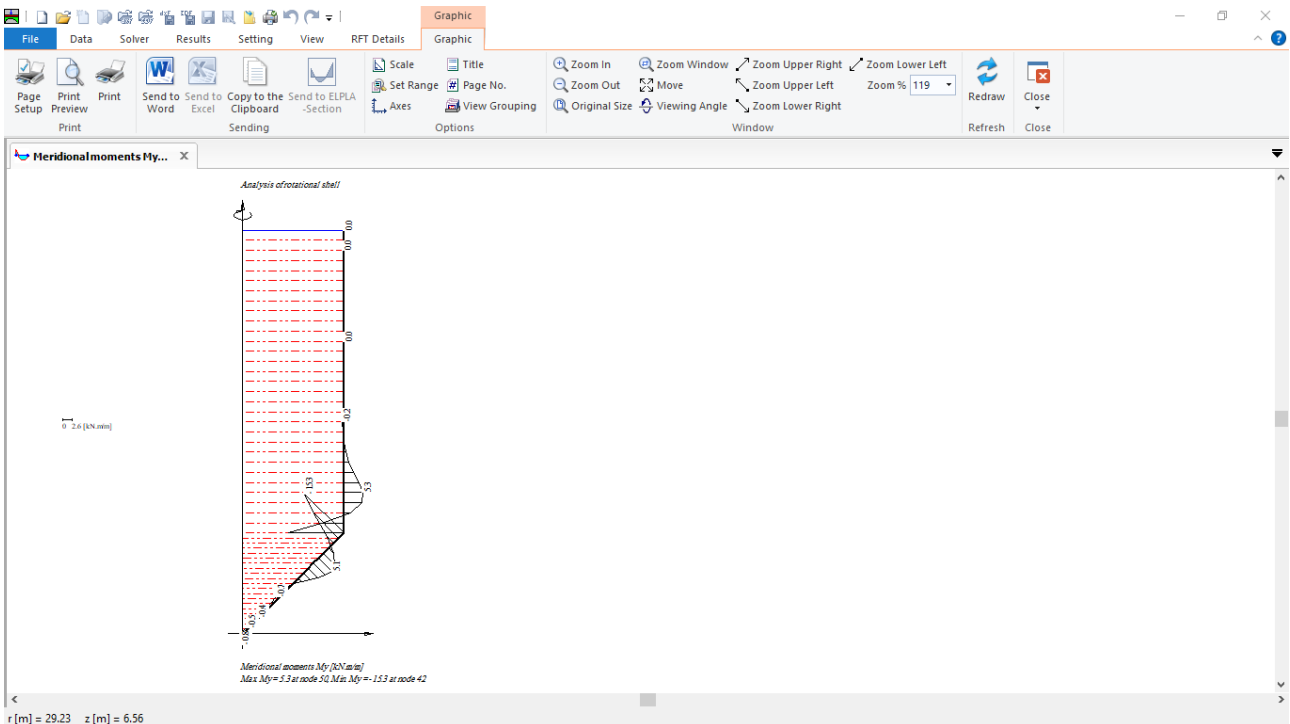


Figure 10.29 Meridional moments on the shell wall

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

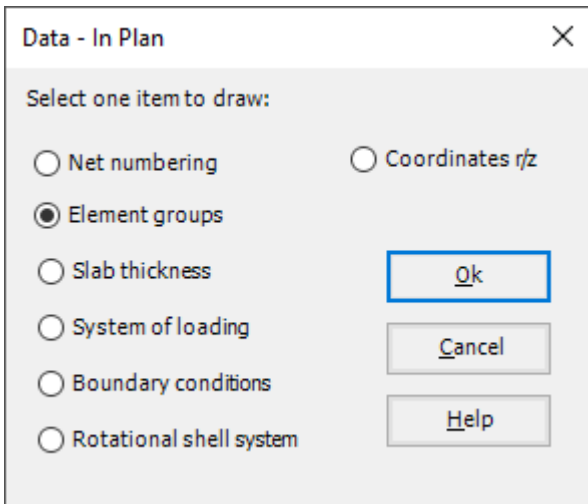


Figure 10.30 "Data – In Plan" option box

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

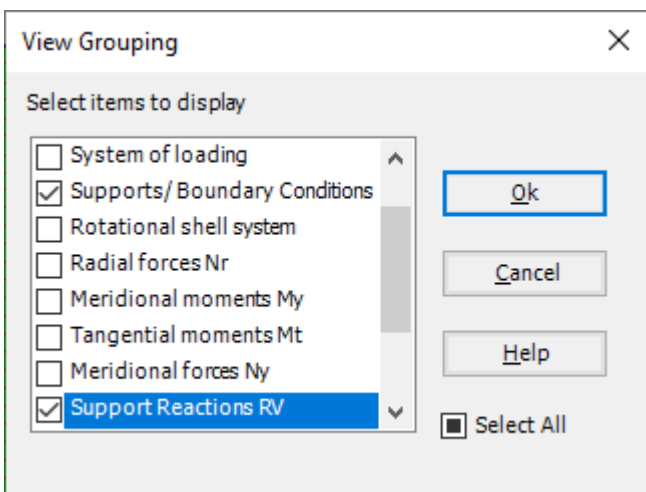


Figure 10.31 "View Grouping" check group box

Example 10

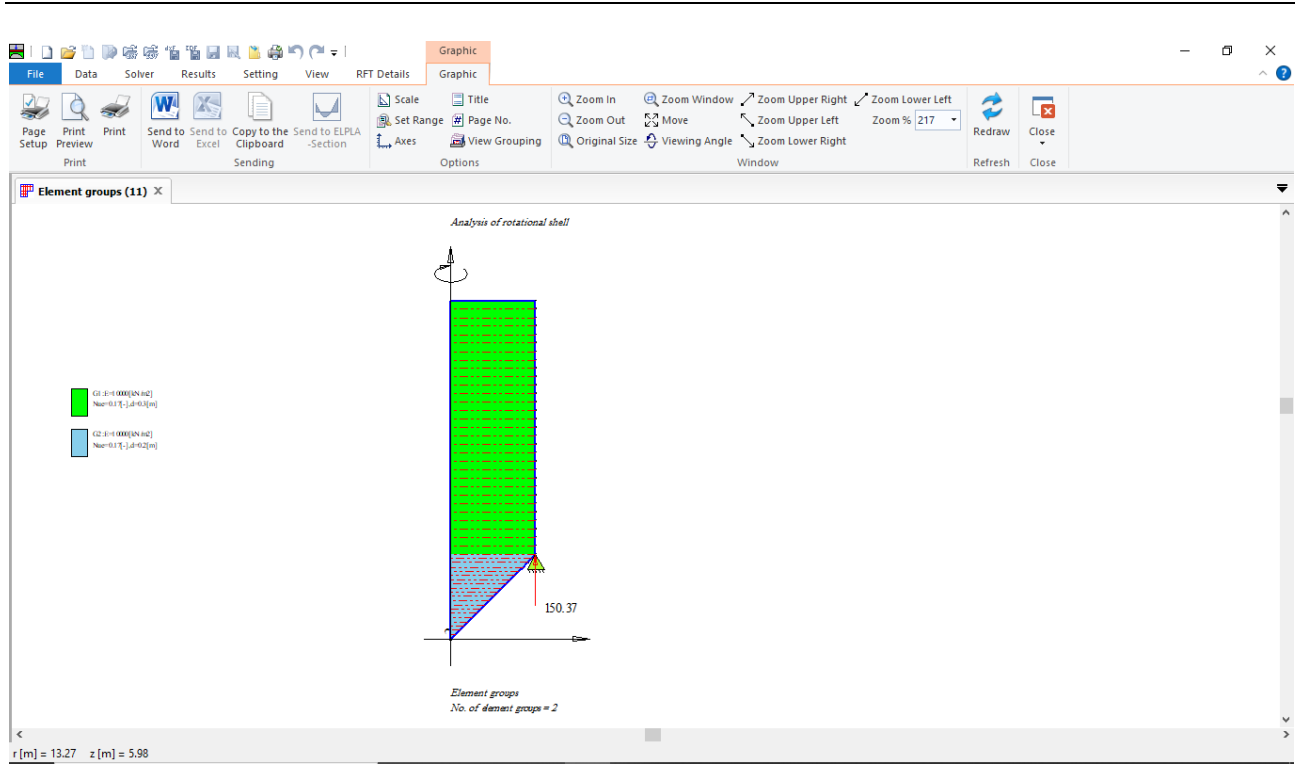


Figure 10.32 Element groups of the tank