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	$= 2\times10^7$	[kN/m^2]
Poisson's ratio of the tank material	ν_c	= 0.15	[-]
Unit weight of the water	γ_w	= 10	[kN/m^3]

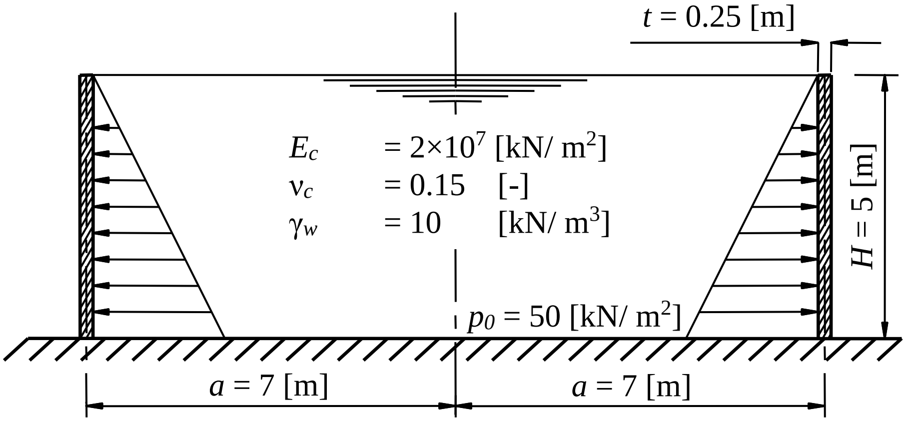

Figure 3.1 Cylindrical circular tank with dimensions

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.

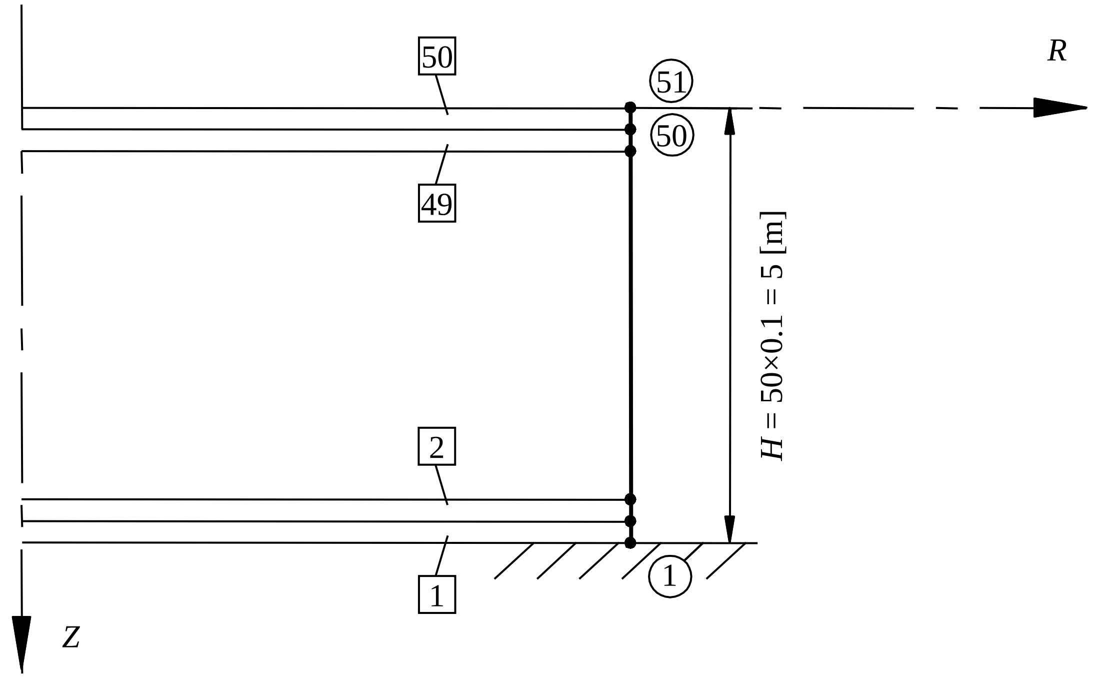

Figure 3.2 Finite element mesh of the tank

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).

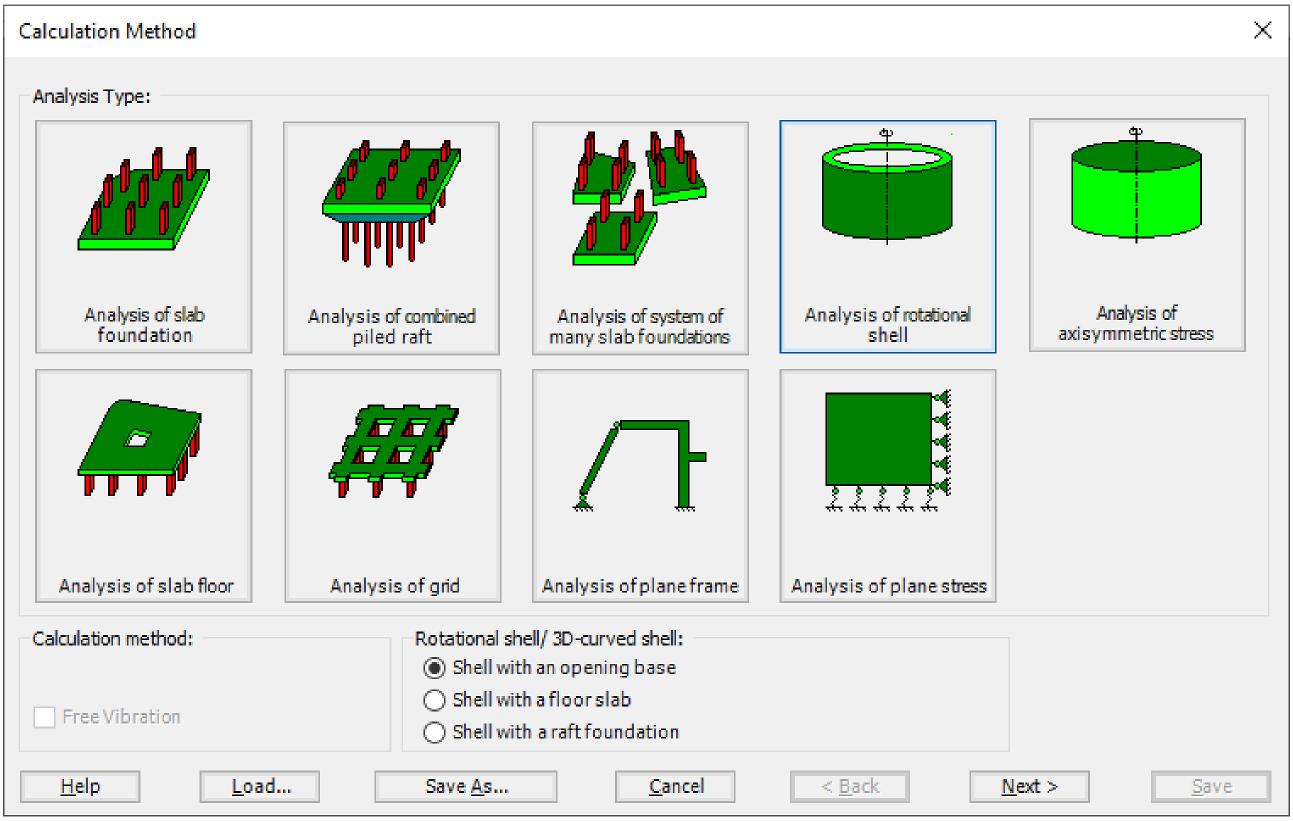

Figure 3.3 "Analysis Type" Form

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.

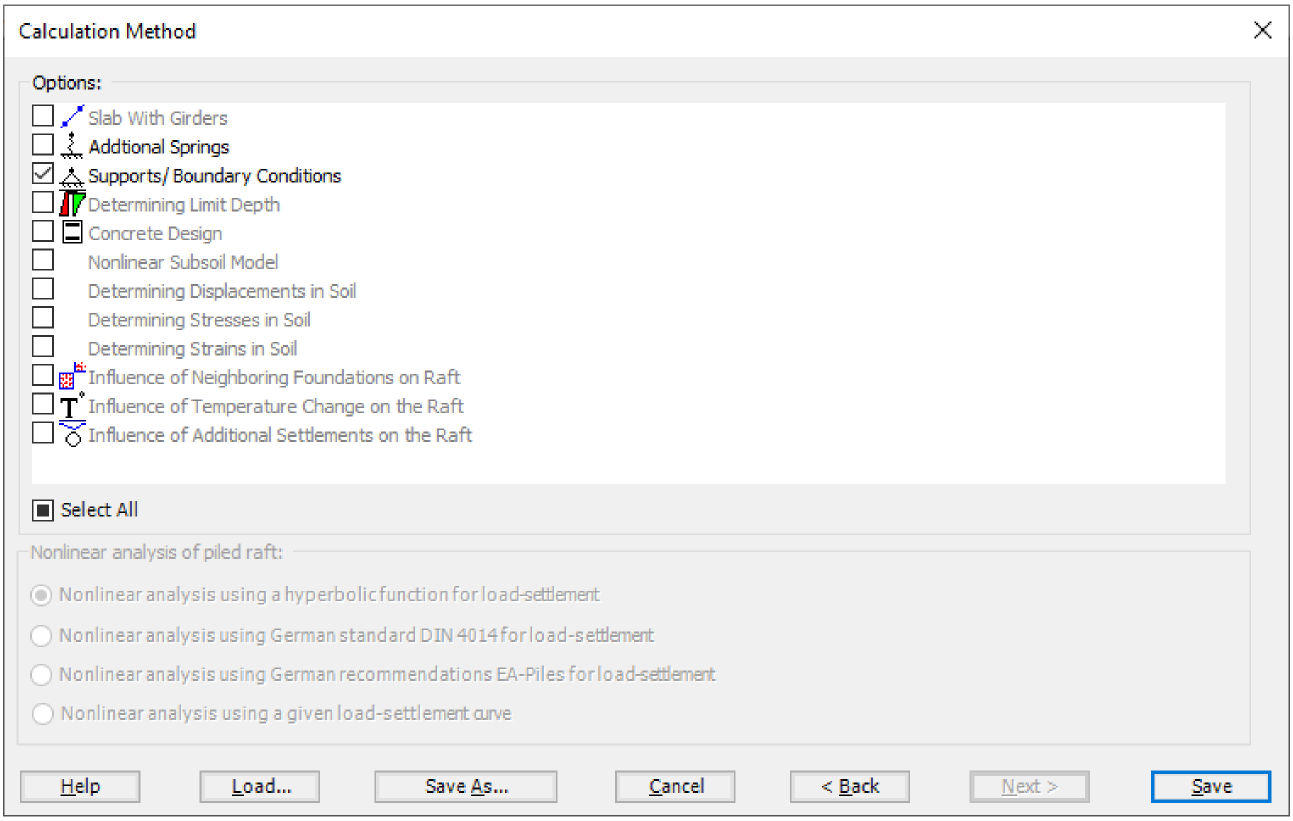

Figure 3.4 "Options" Form

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.

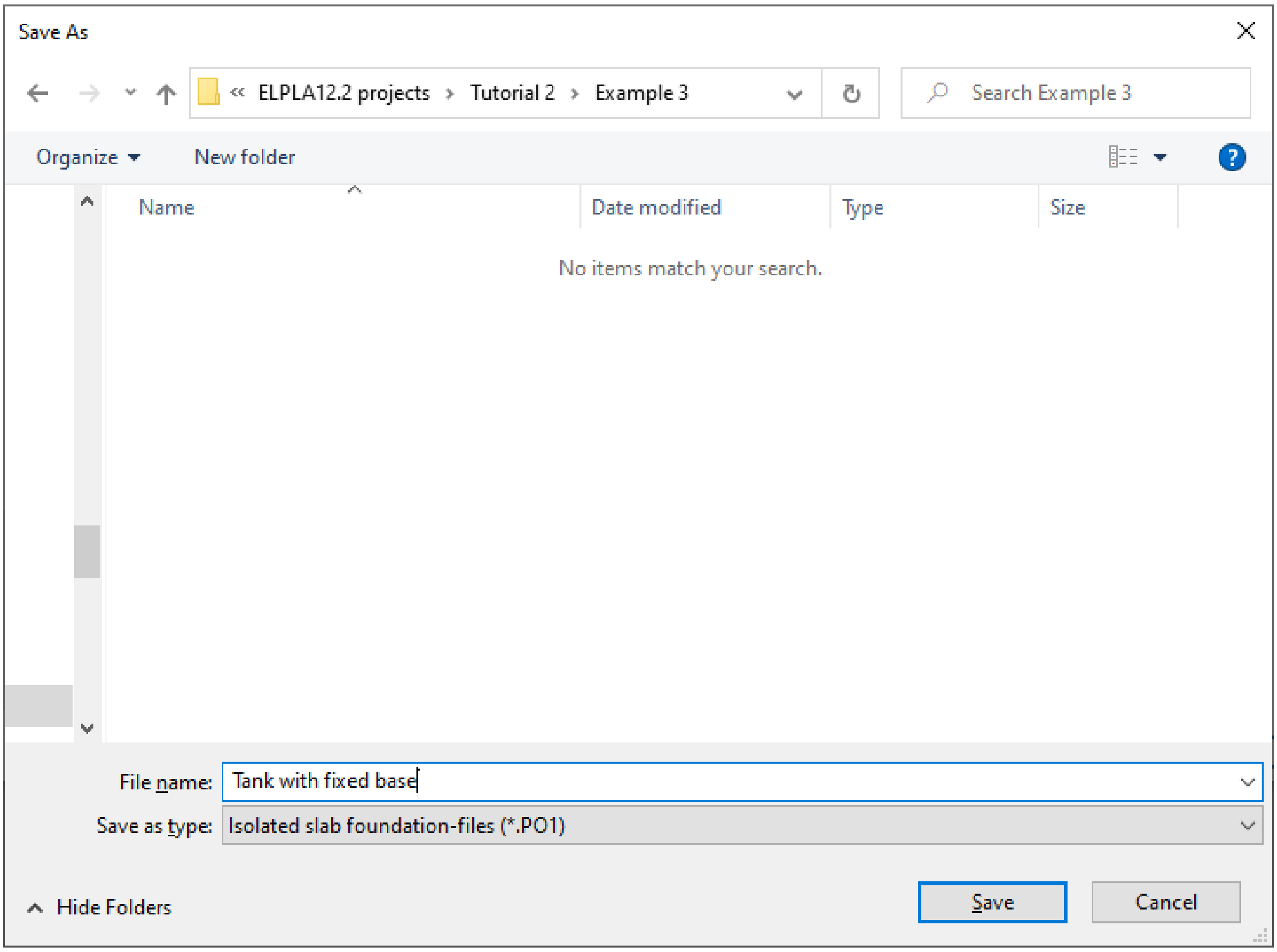

Figure 3.5 "Save as" dialog box

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

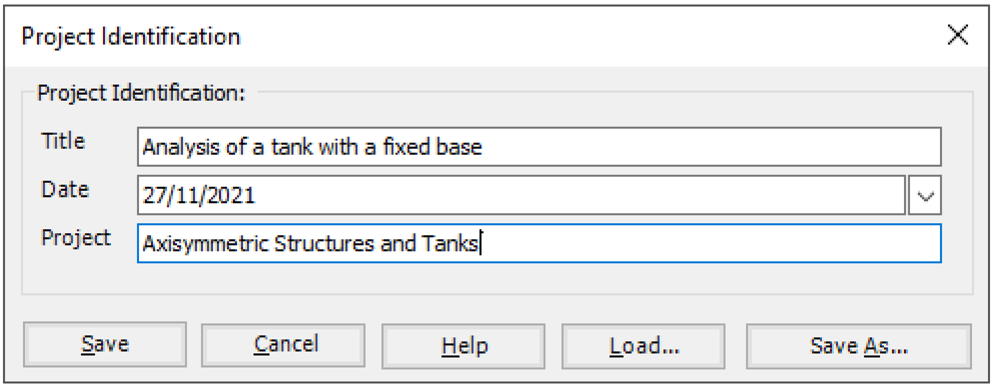

Figure 3.6 "Project Identification" dialog box

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.

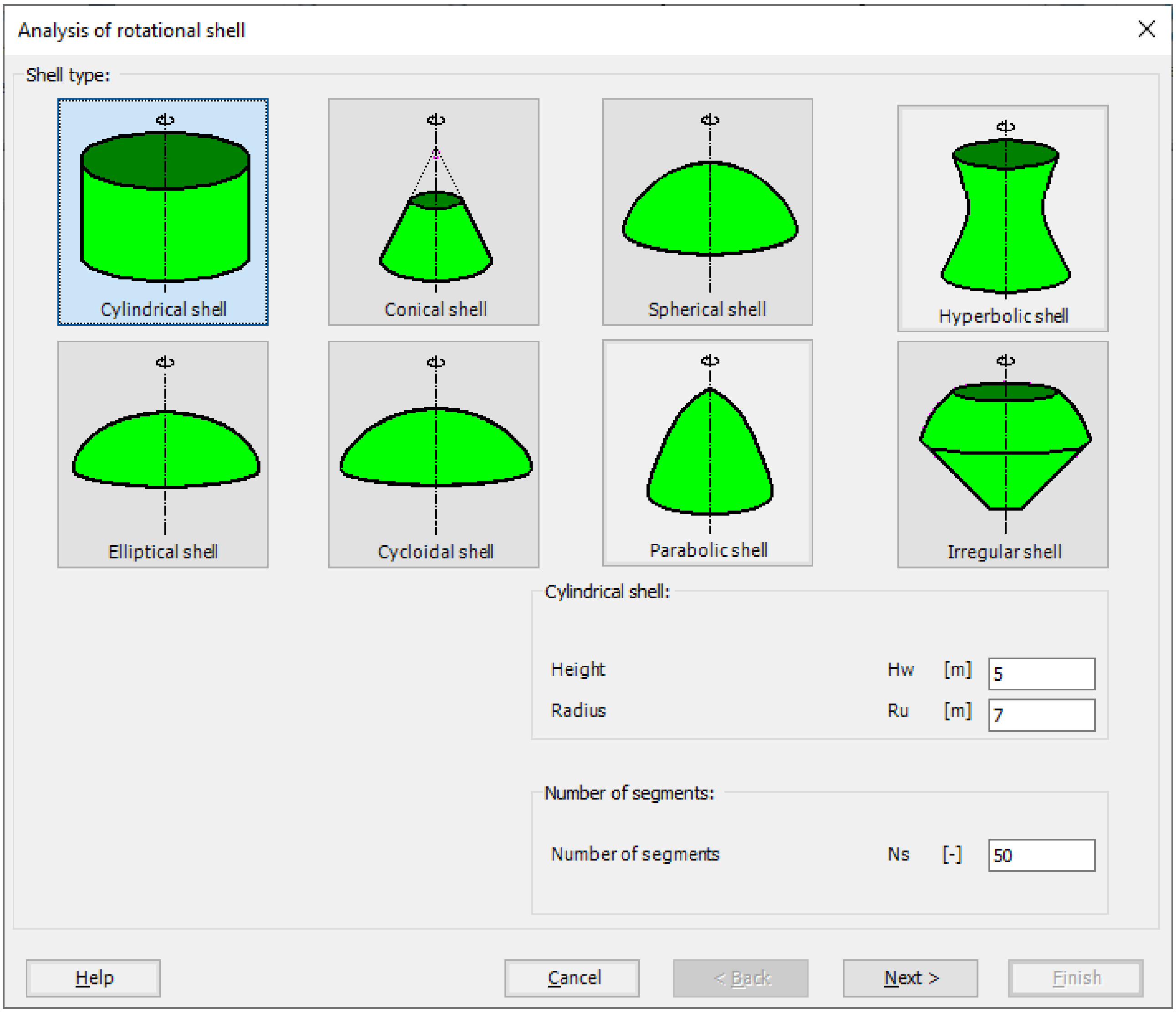

Figure 3.7 "Analysis of rotational shell" wizard with "Shell type" Form

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.

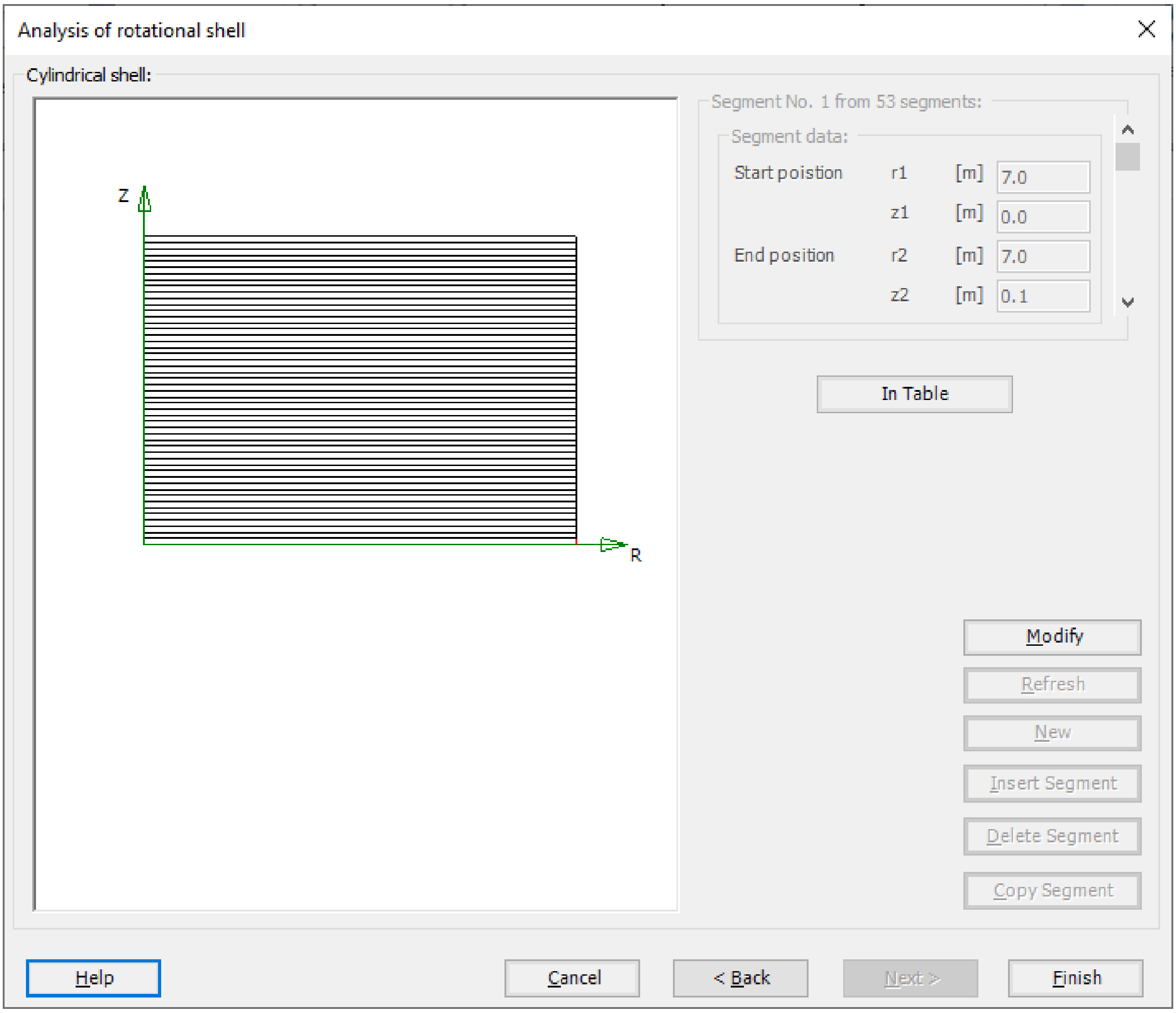

Figure 3.8 "Cylindrical shell" Form

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

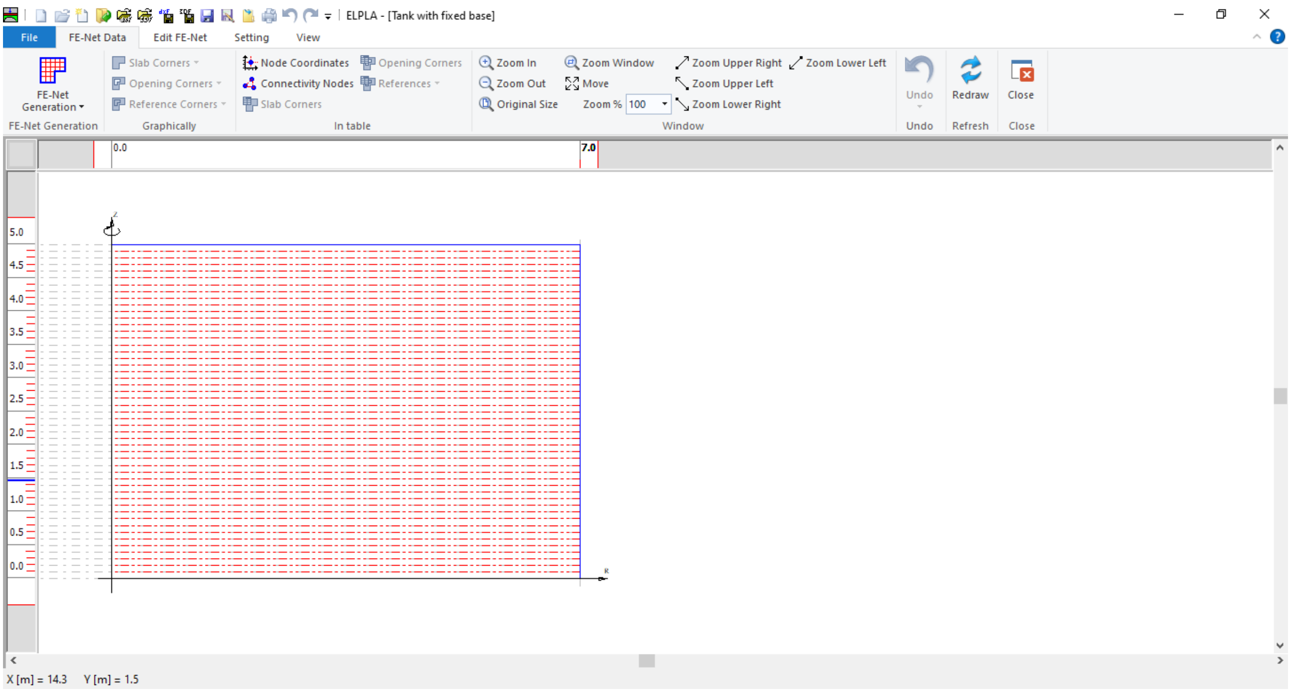

Figure 3.9 Generated FE-Net

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.

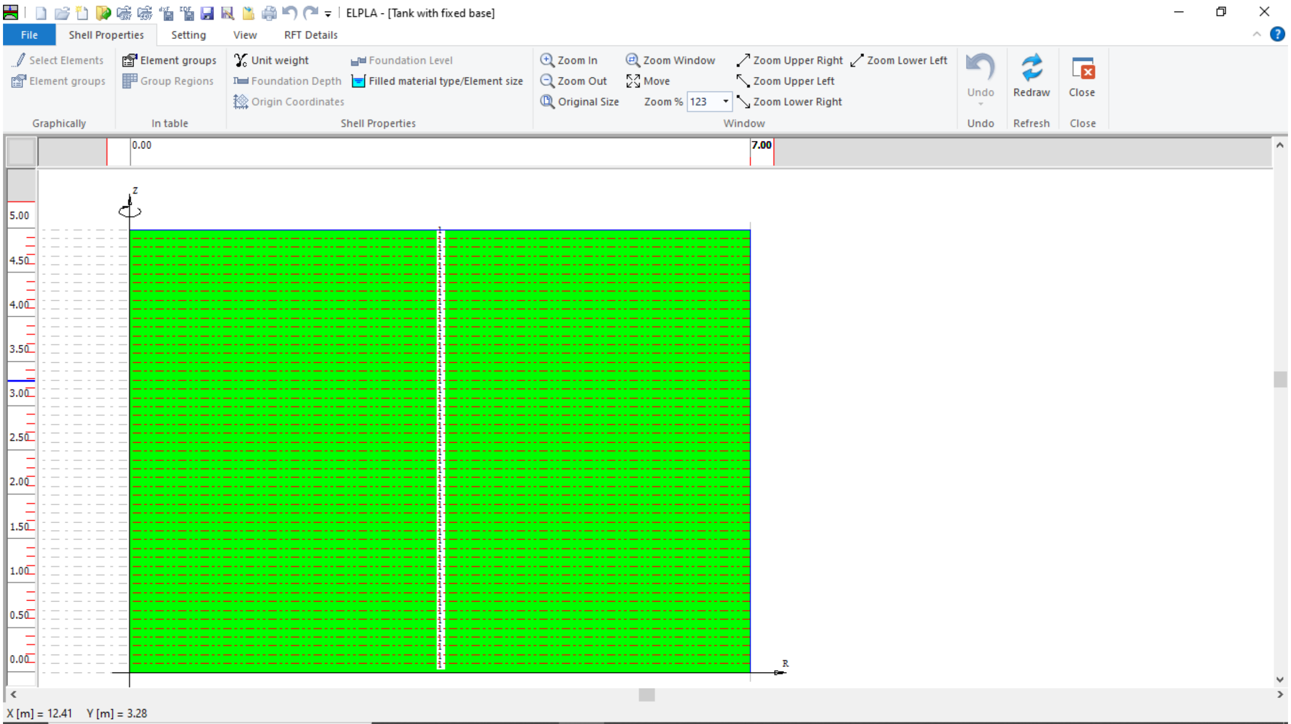

Figure 3.10 "Shell Properties" Window

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.

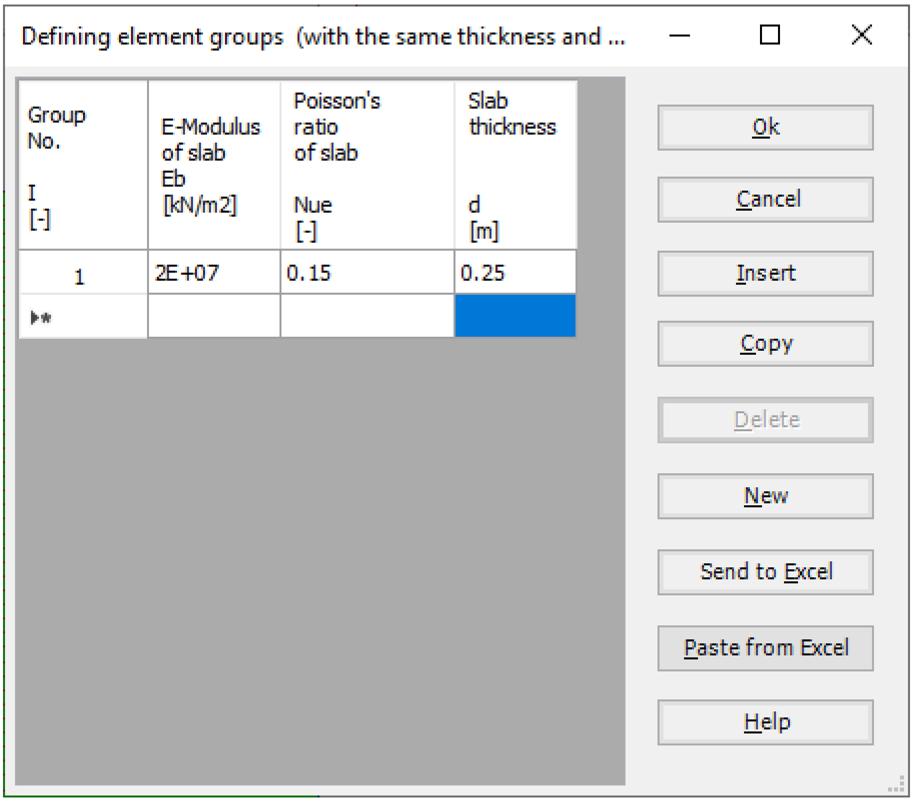

Figure 3.11 "Defining element groups" list box

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.

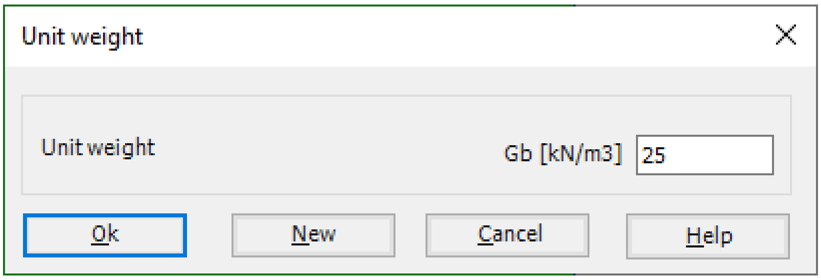

Figure 3.12 "Unit weight" dialog box

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

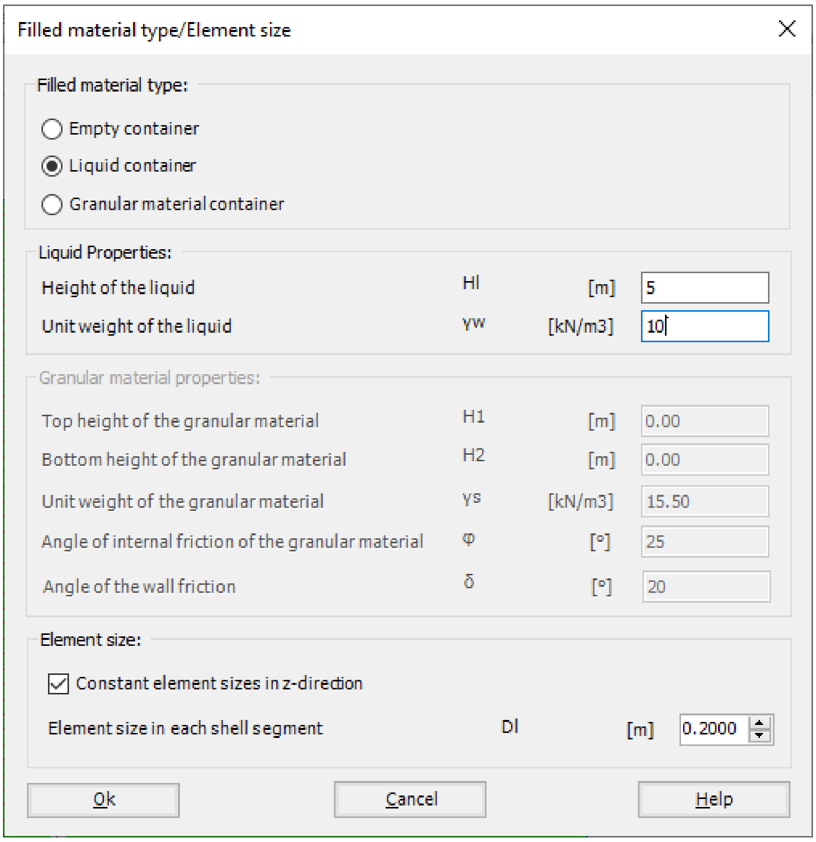

Figure 3.13 "Filled material type/Element size" Form

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.

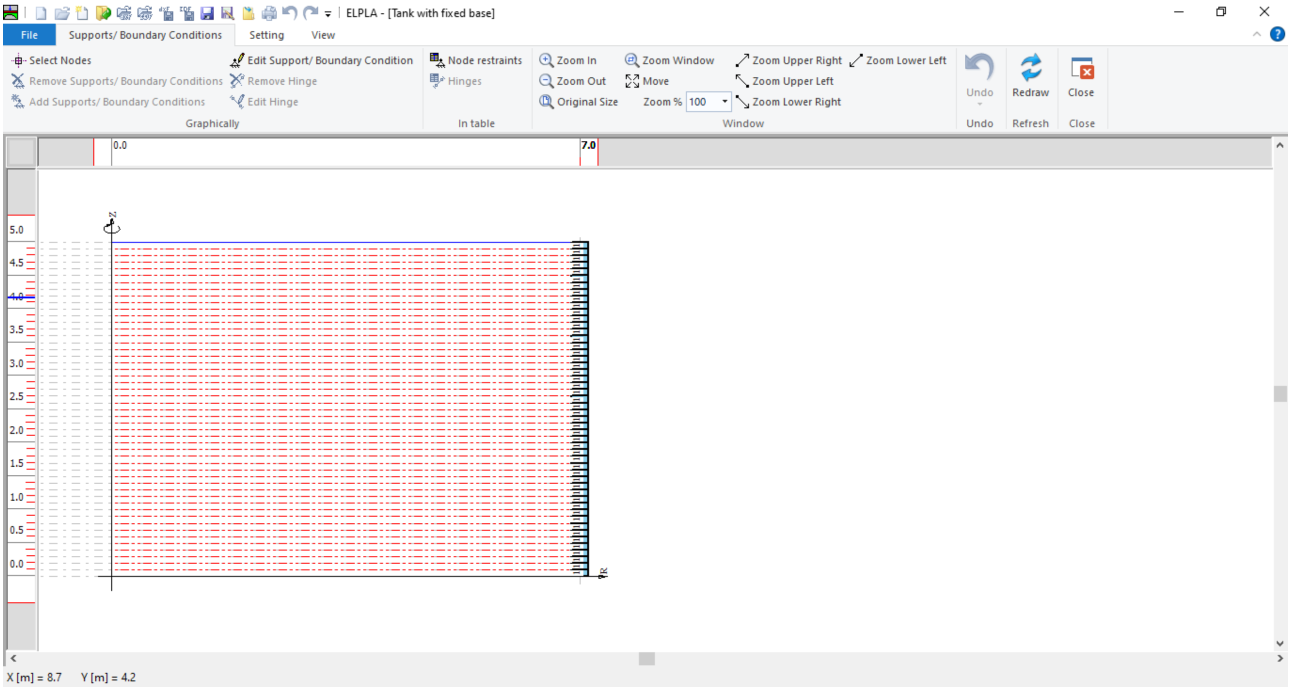

Figure 3.14 "Supports/ Boundary Conditions" Window

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.

-

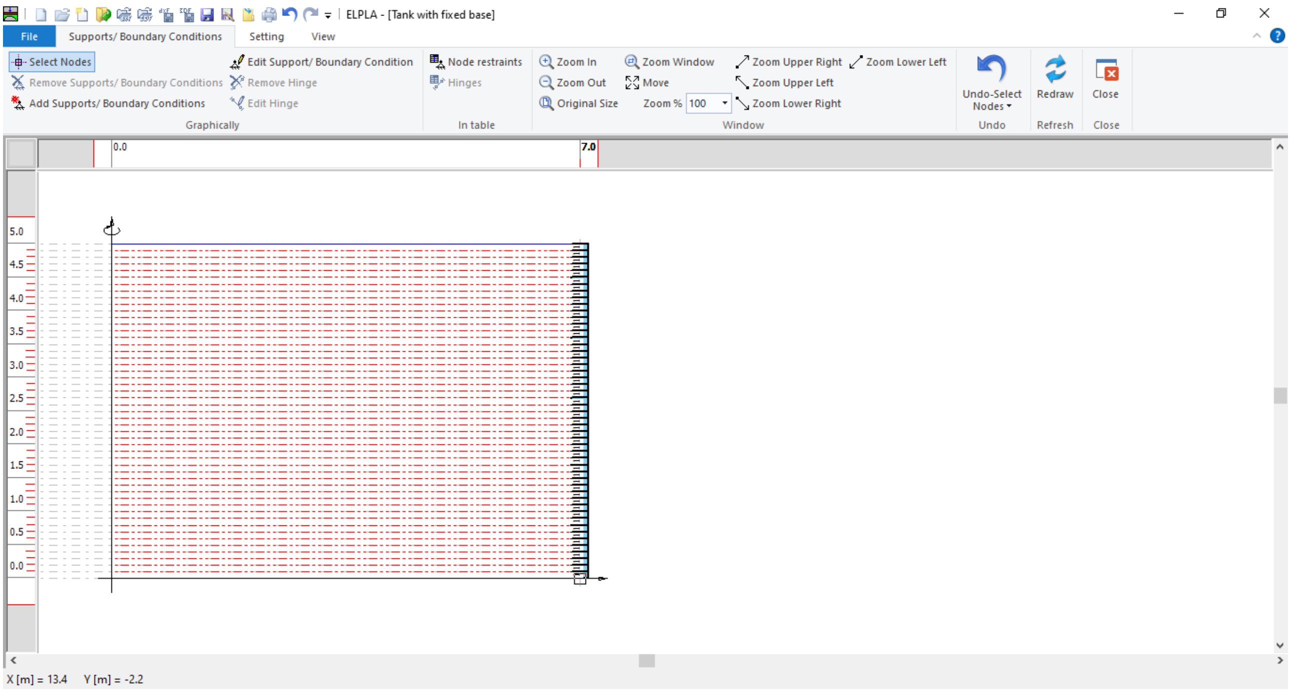

Figure 3.15 Selection of node that has a fixed support

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

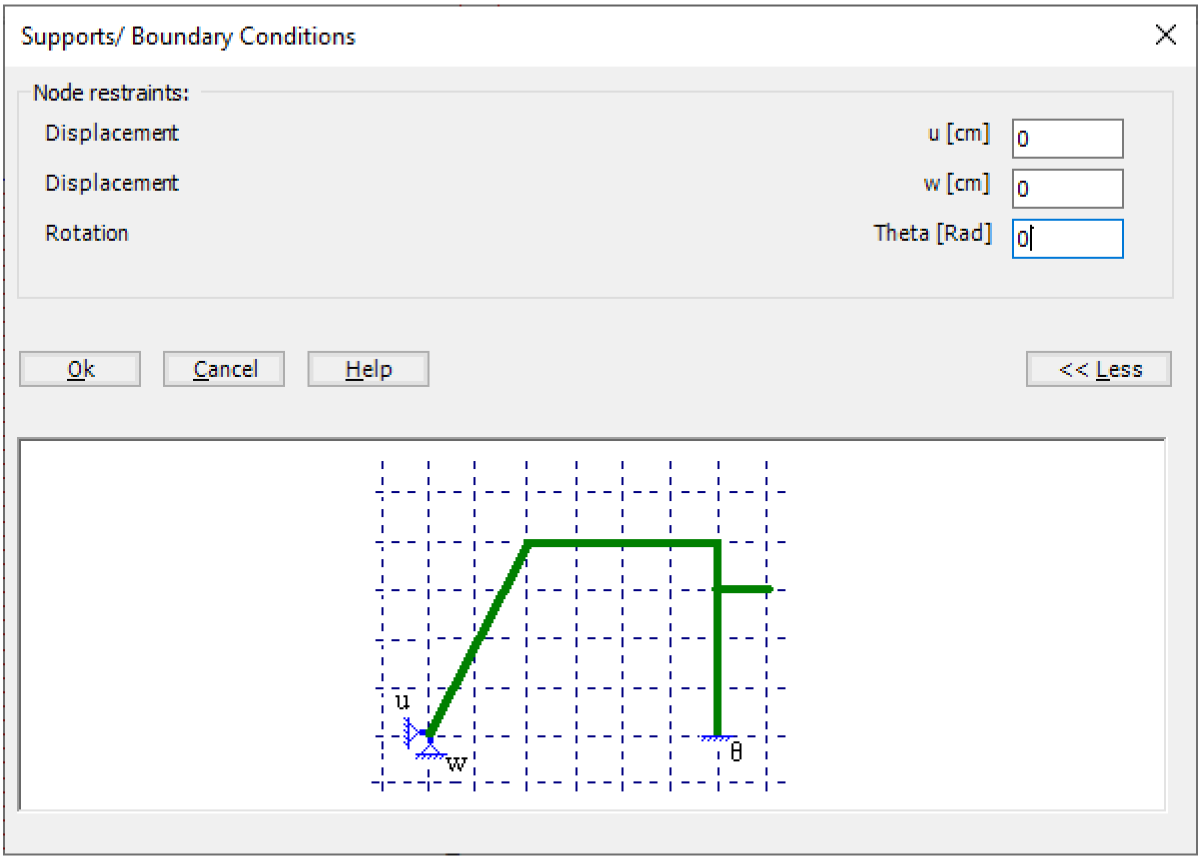

Figure 3.16 "Supports/ Boundary Conditions" dialog box

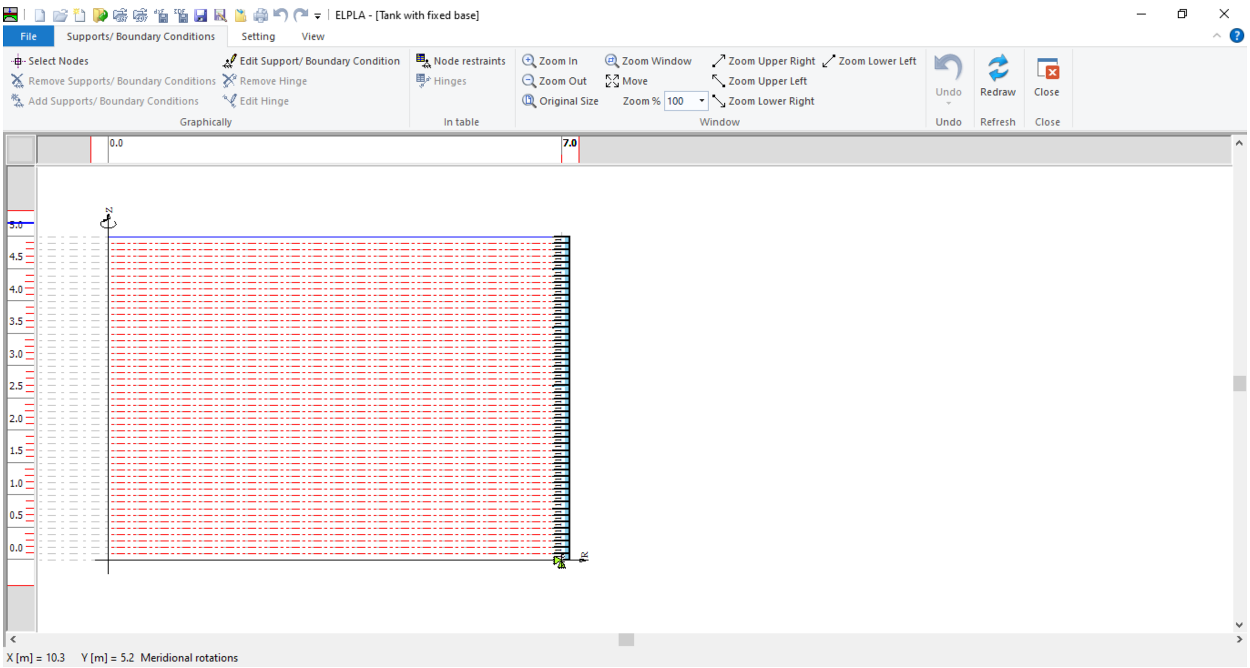

Figure 3.17 Supports on the screen

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.

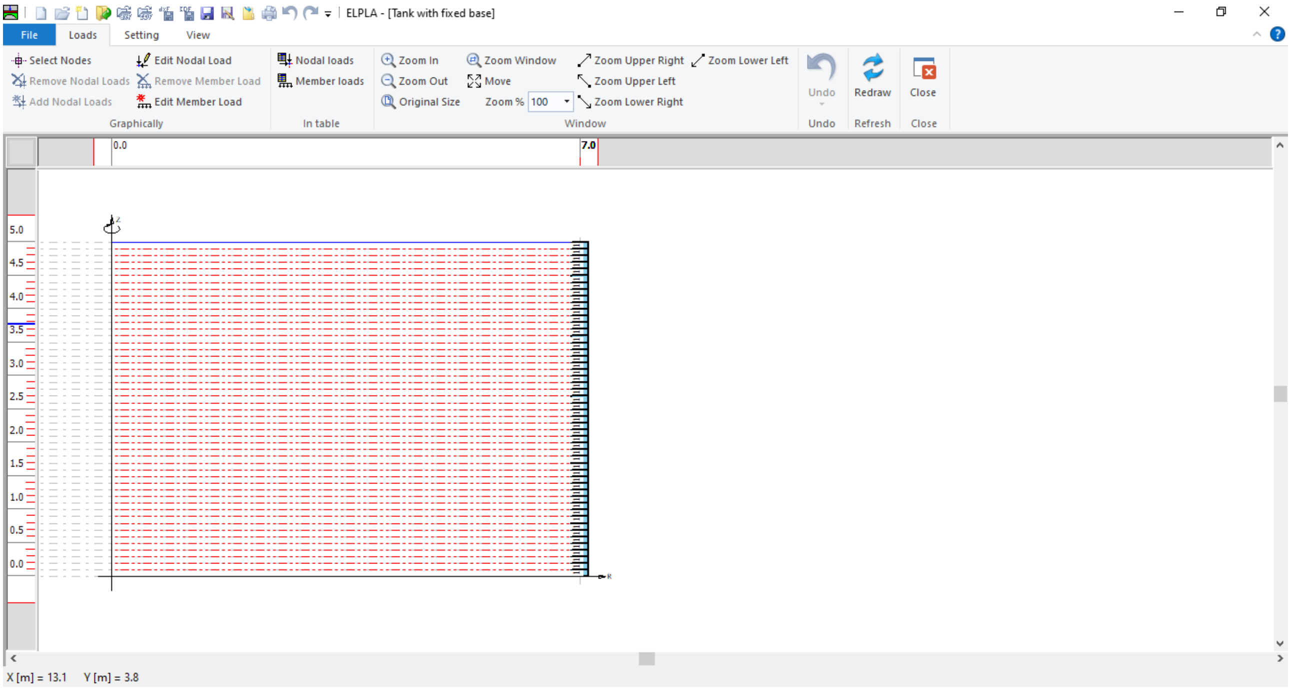

Figure 3.18 "Loads" Window

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.

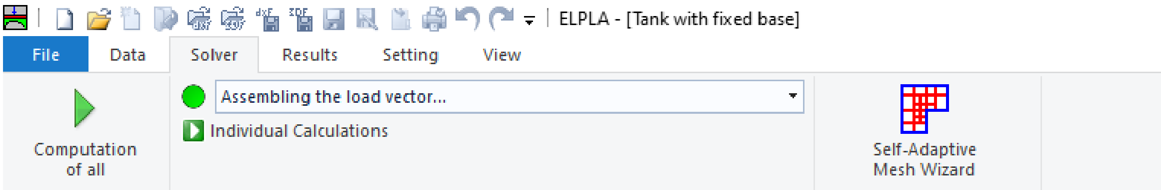

Figure 3.19 "Solver" Tab

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.

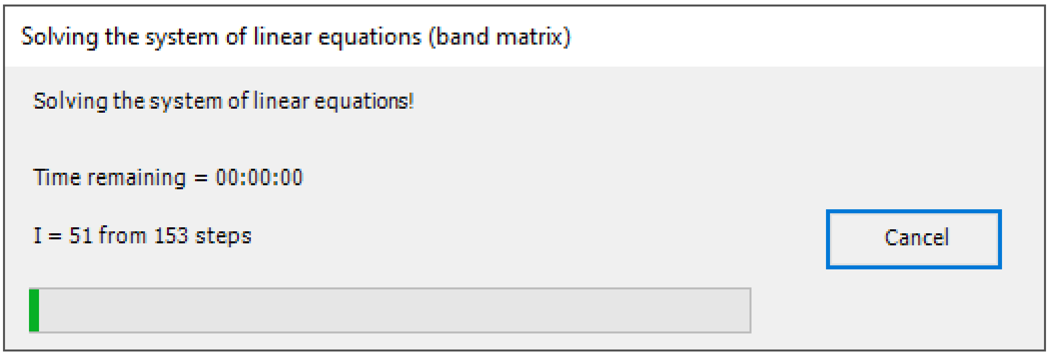

Figure 3.20 Analysis progress menu

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.

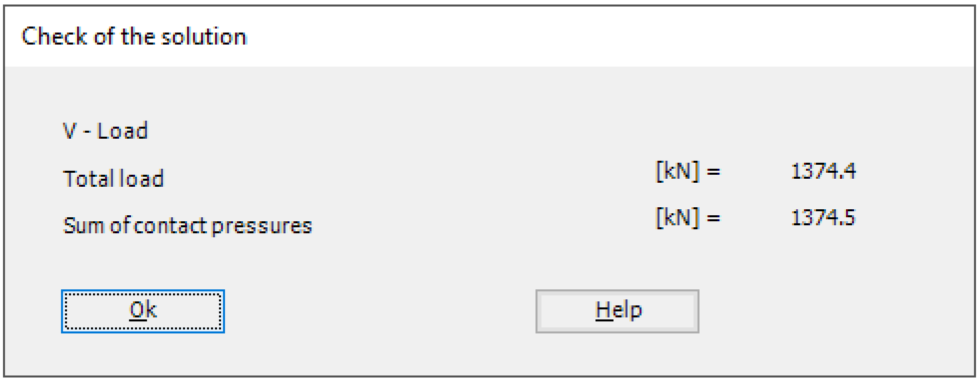

Figure 3.21 Menu "Check of the solution"

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).

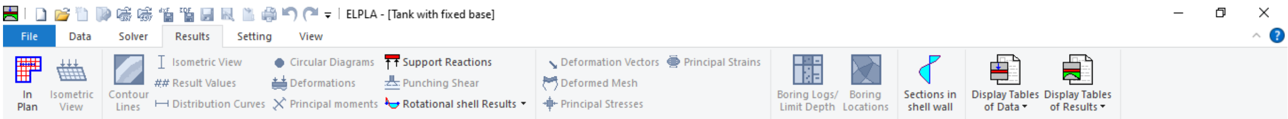

Figure 3.22 "Results" Tab

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.

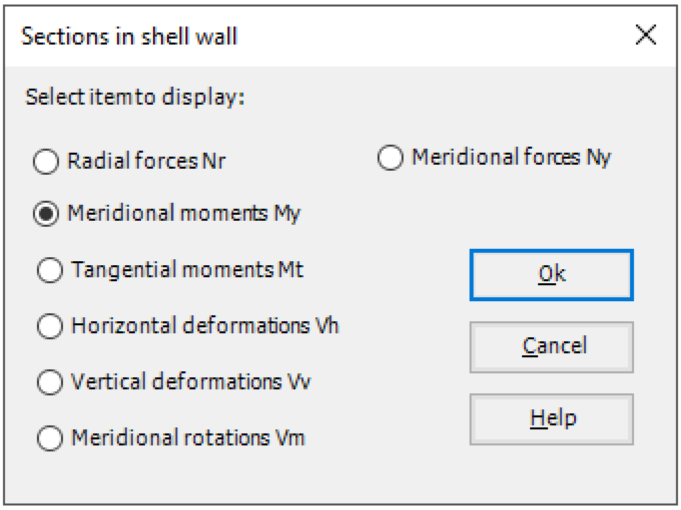

Figure 3.23 "Sections in shell wall" option box

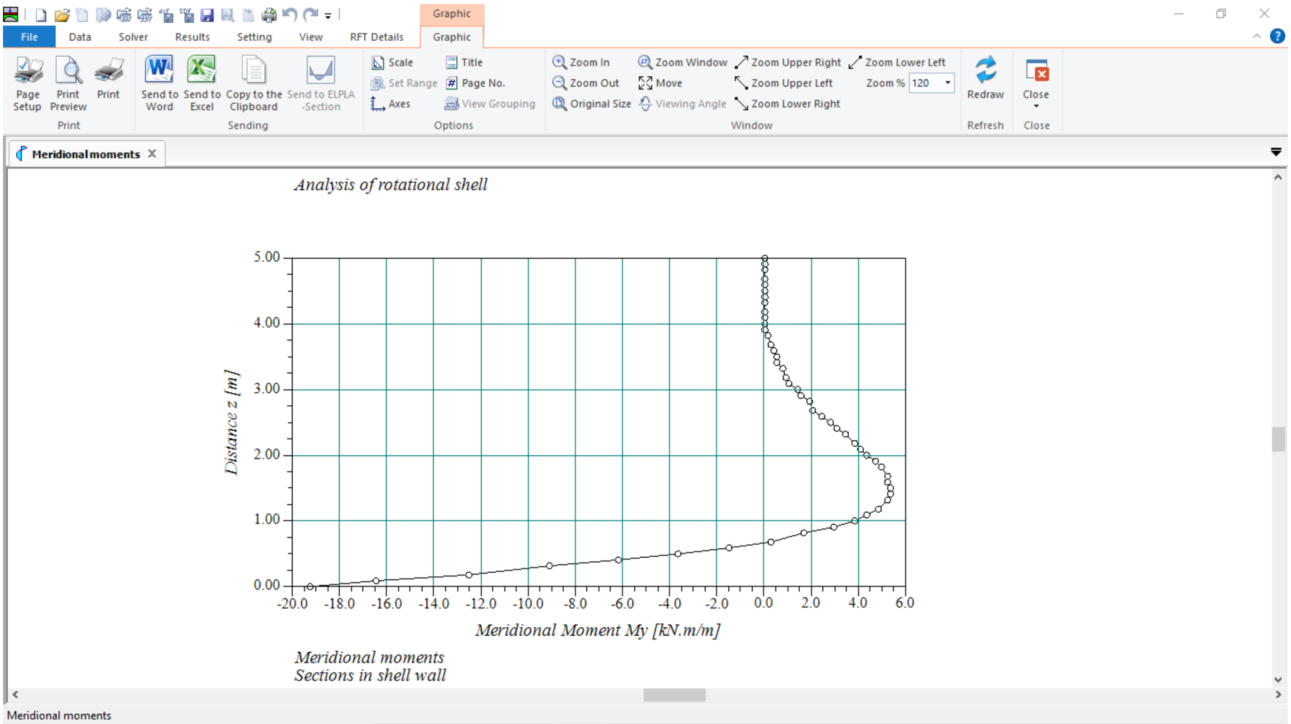

Figure 3.24 Meridional moments sections in shell wall

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.

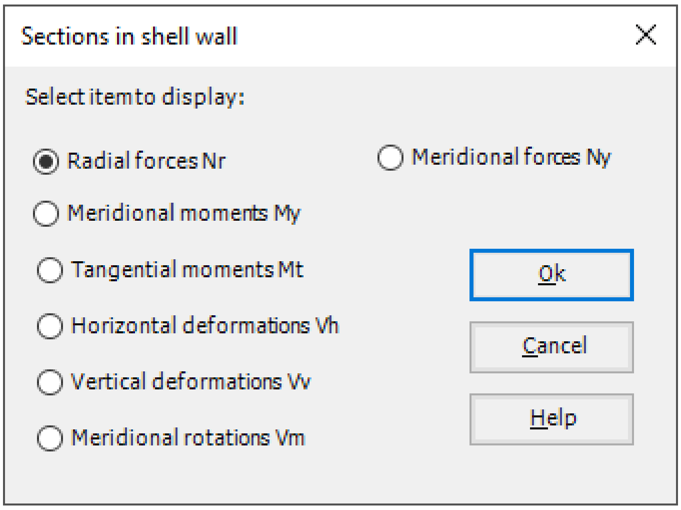

Figure 3.25 "Distribution of Internal Forces" option box

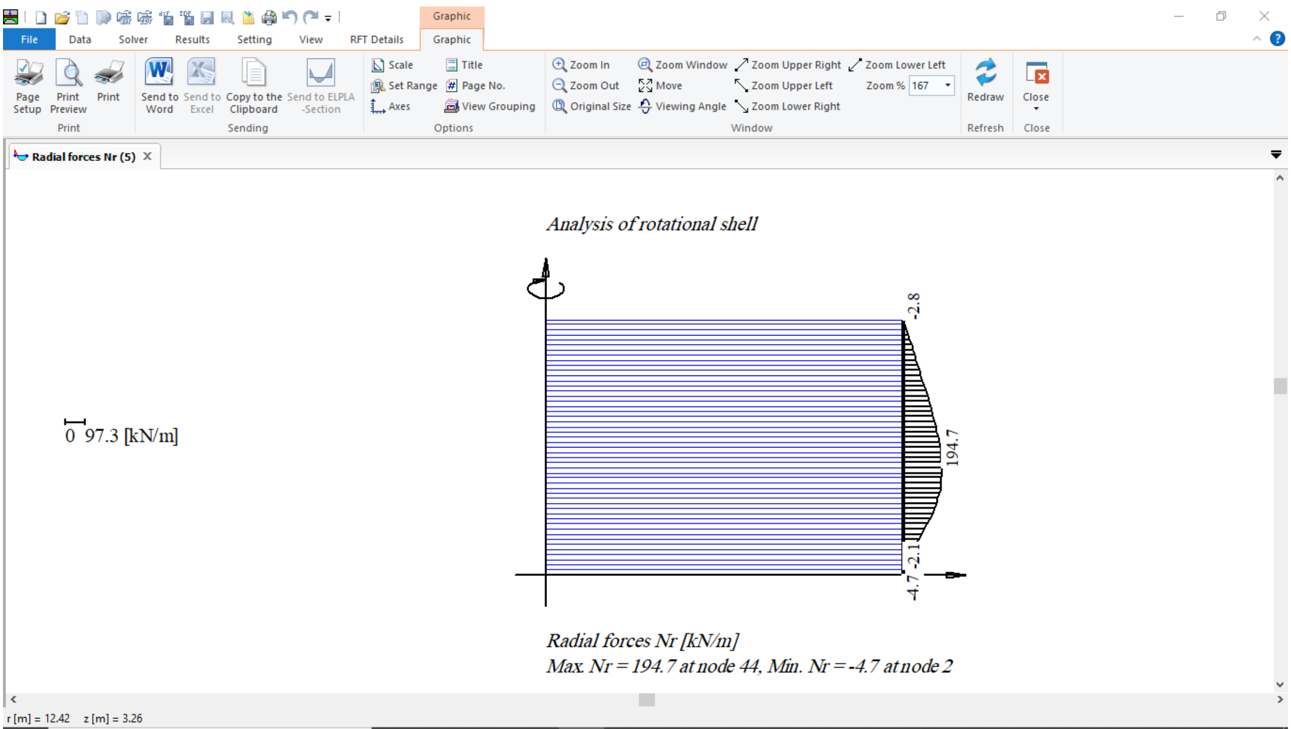

Figure 3.26 Radial forces on the shell wall

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

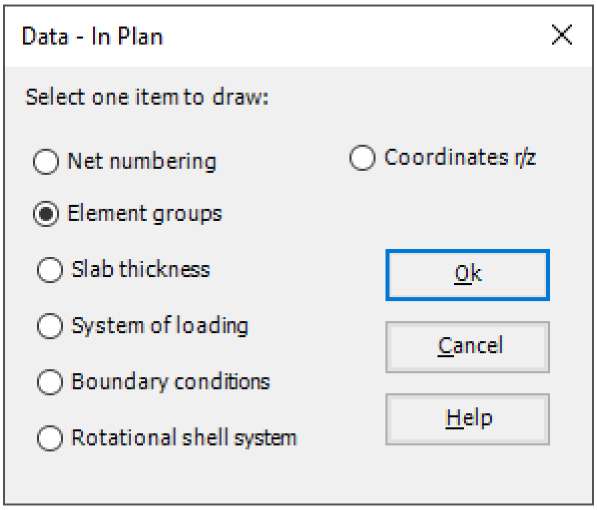

Figure 3.27 "Data – In Plan" option box

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

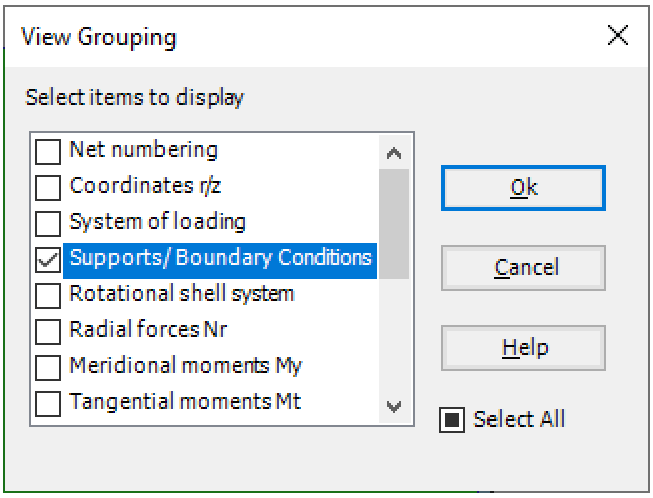

Figure 3.28 "View Grouping" check group box

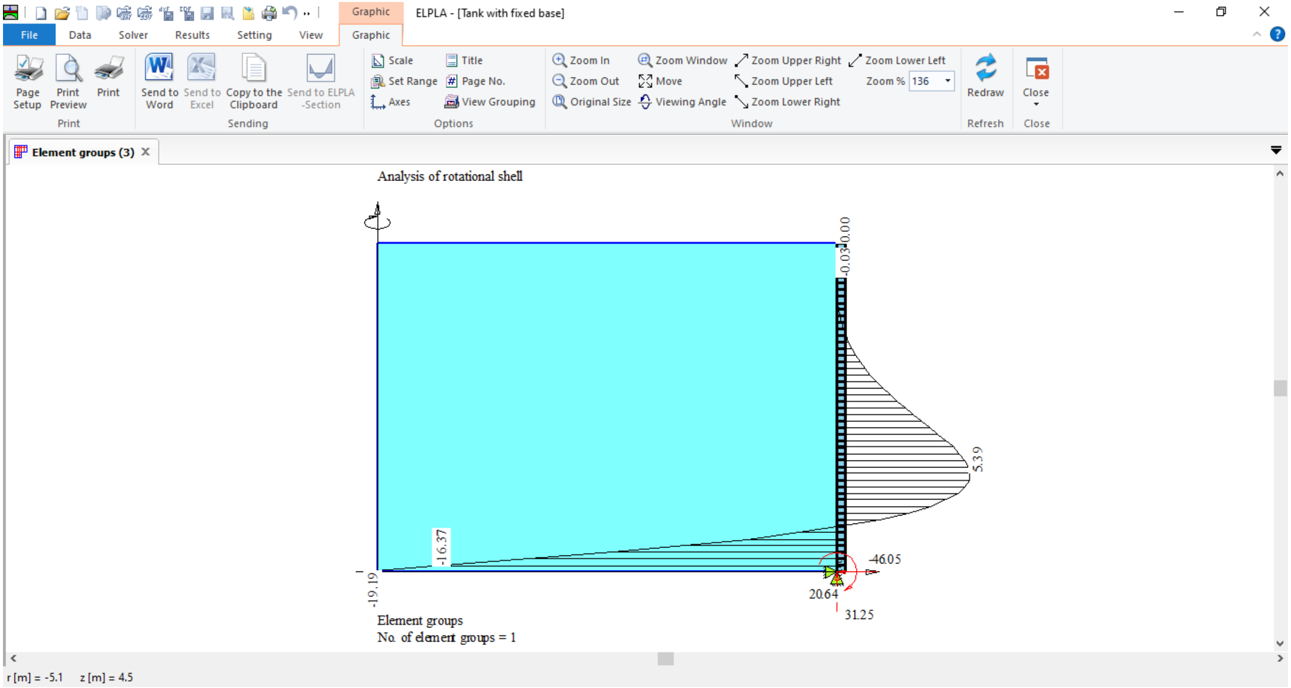

Figure 3.29 Element groups