# 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.

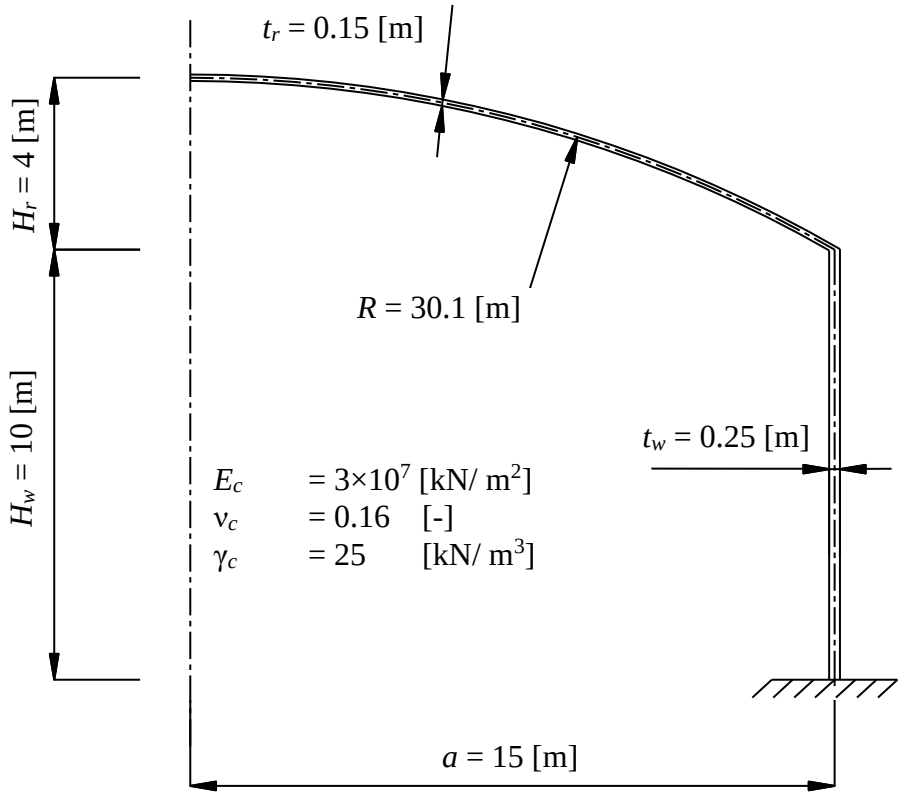


Figure 5.1 Radial section through the tank

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 | $\nu_c$ | = 0.16 | [-] |
| Unit weight of the tank material | $\gamma_c$ | = 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.

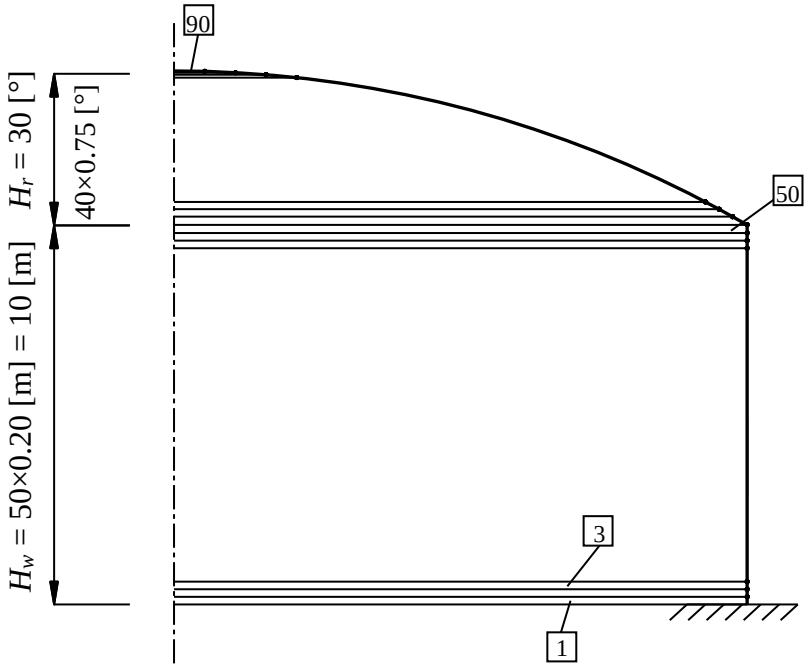


Figure 5.2 Finite element mesh of the tank

## 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).

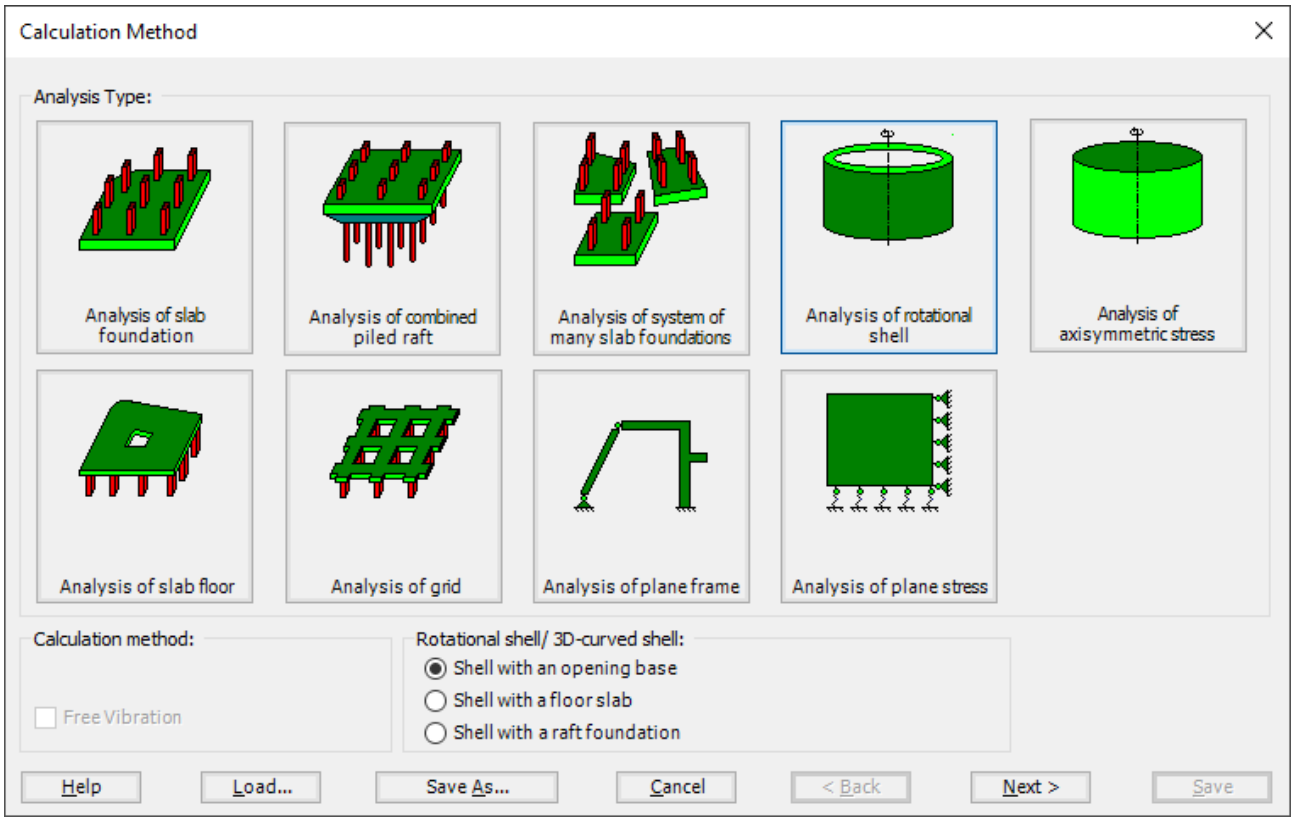


Figure 5.3 "Analysis Type" Form

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.

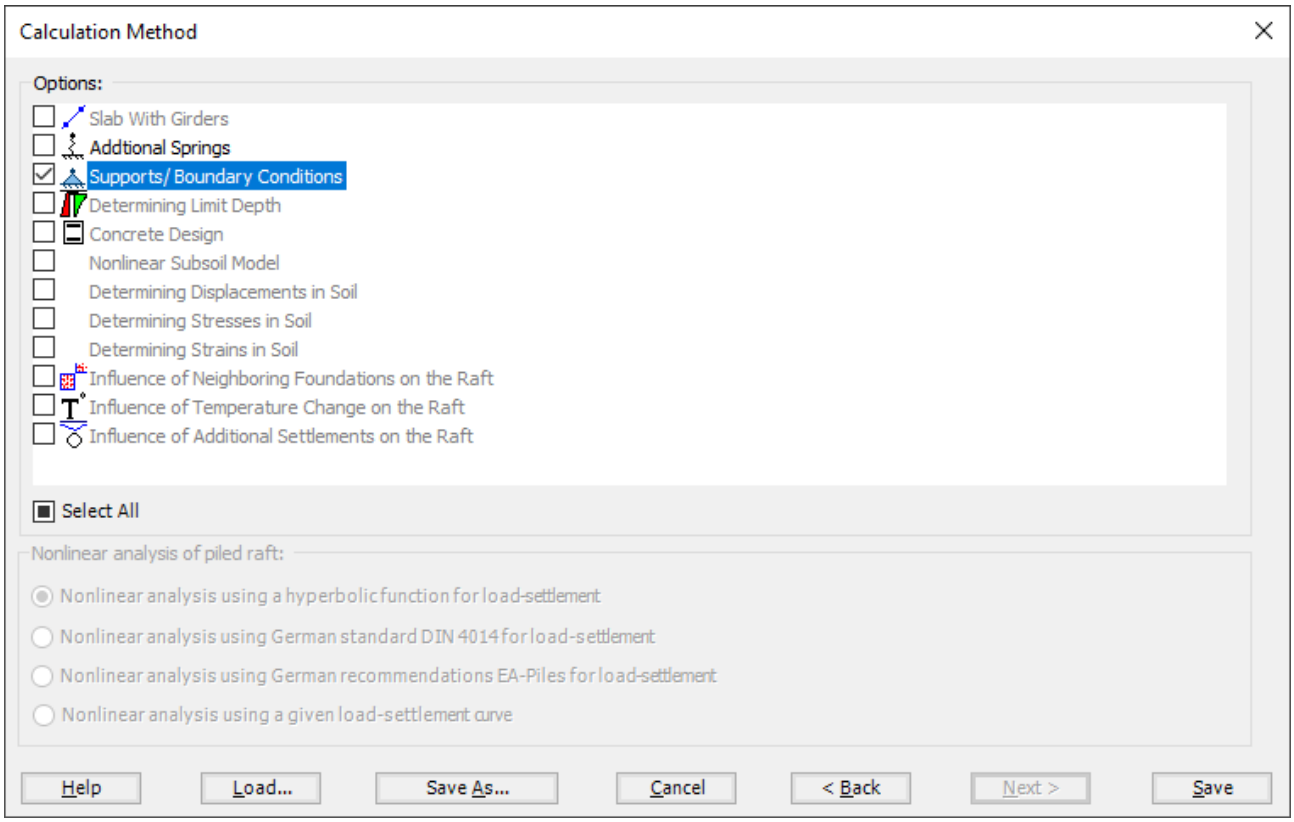


Figure 5.4 "Options" Form

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.

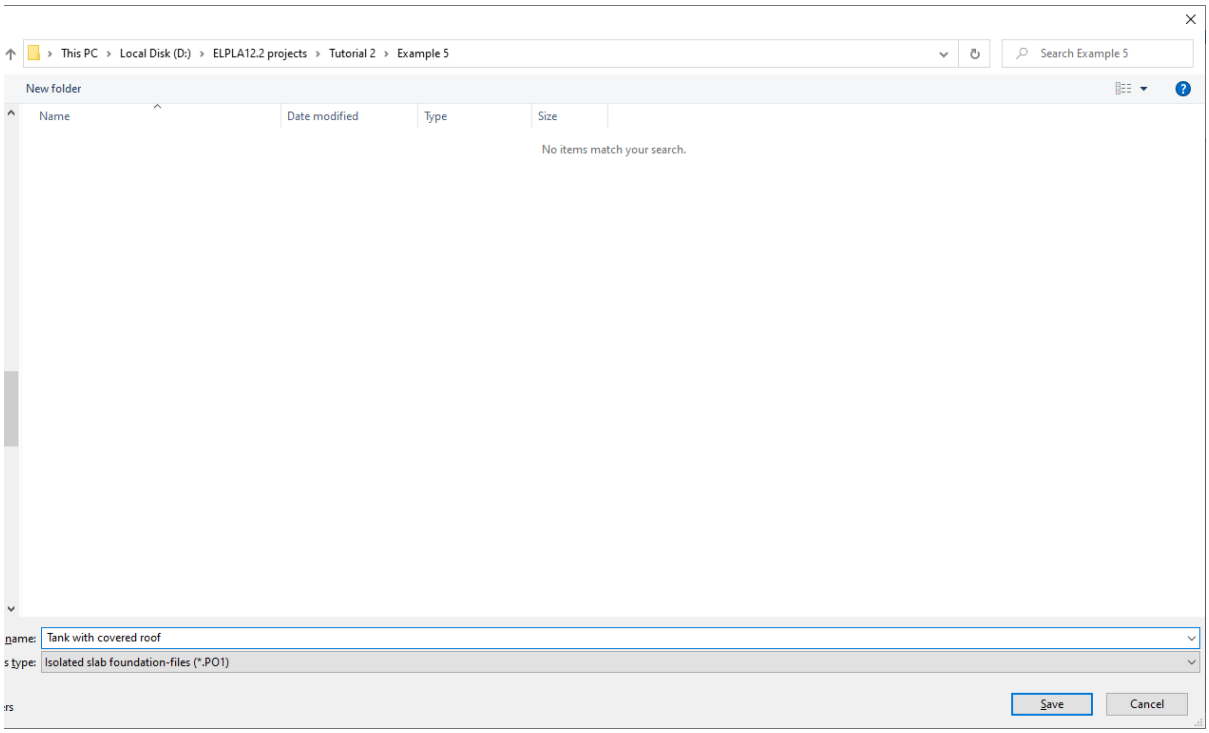


Figure 5.5 "Save as" dialog box

*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

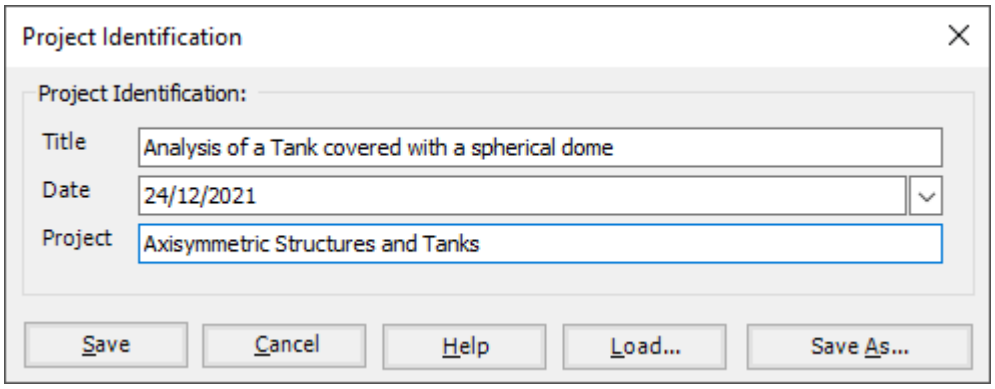


Figure 5.6 "Project Identification" dialog box

## 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.

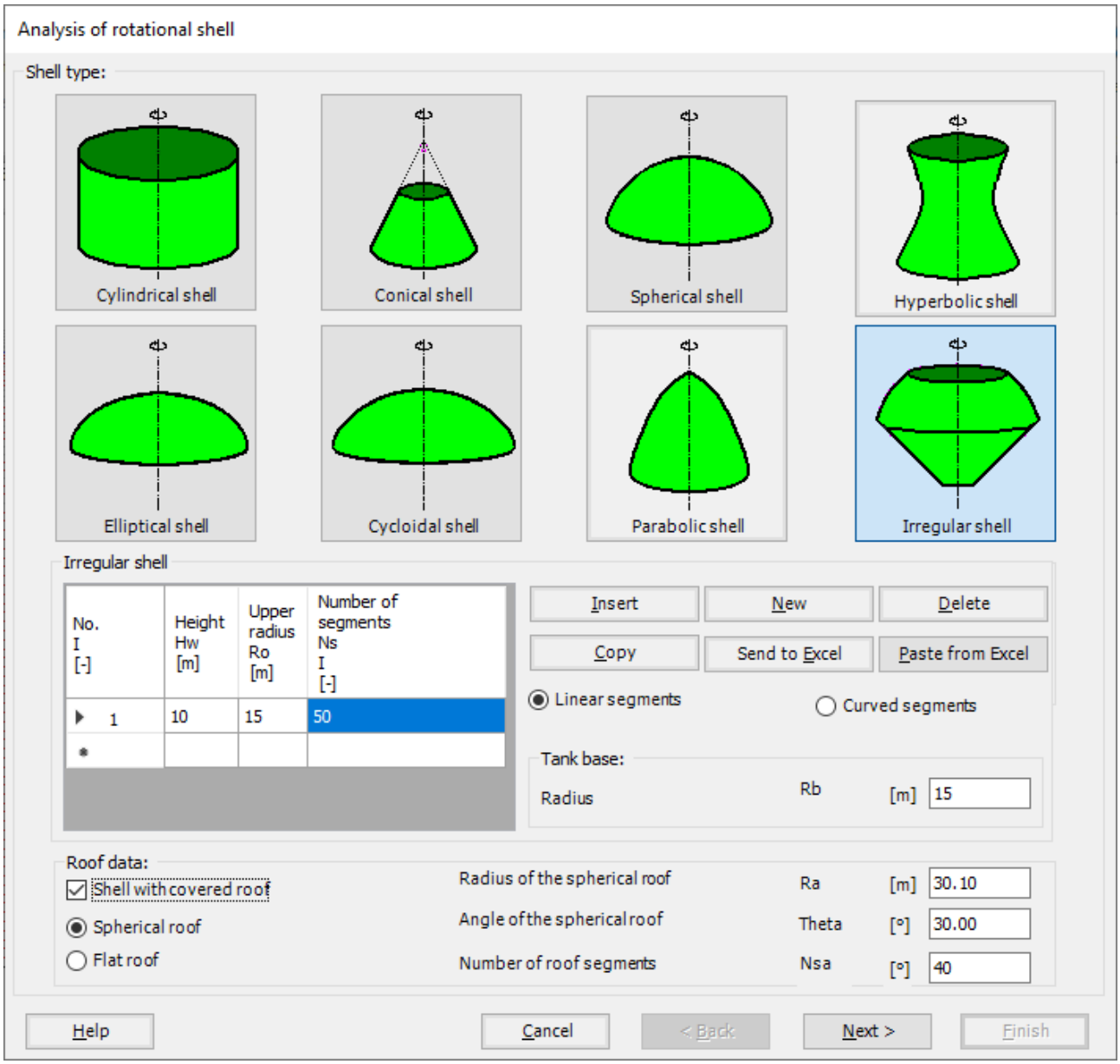


Figure 5.7 "Analysis of rotational shell" wizard with "Shell type" Form

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.

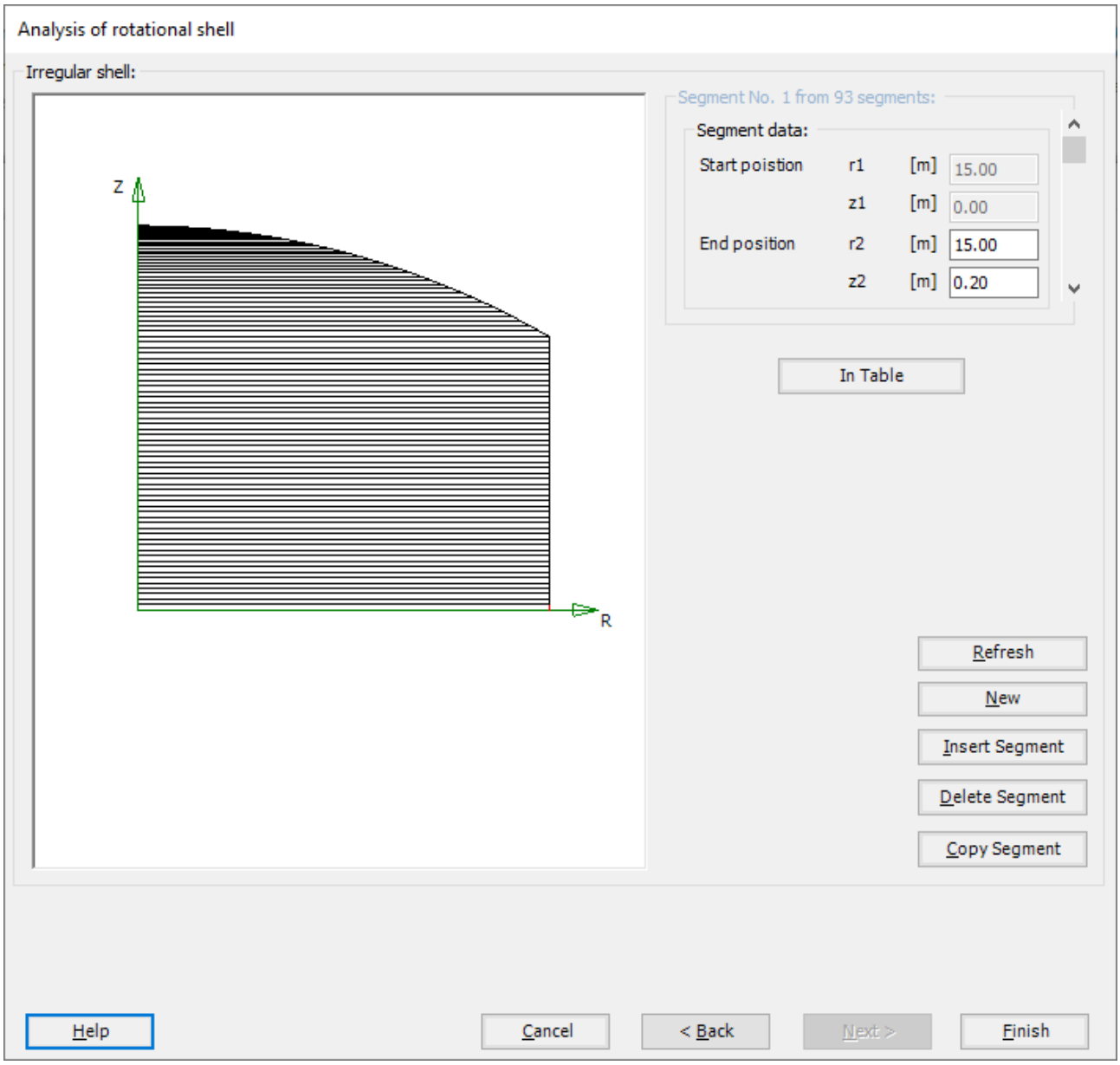


Figure 5.8 "Irregular shell" Form

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

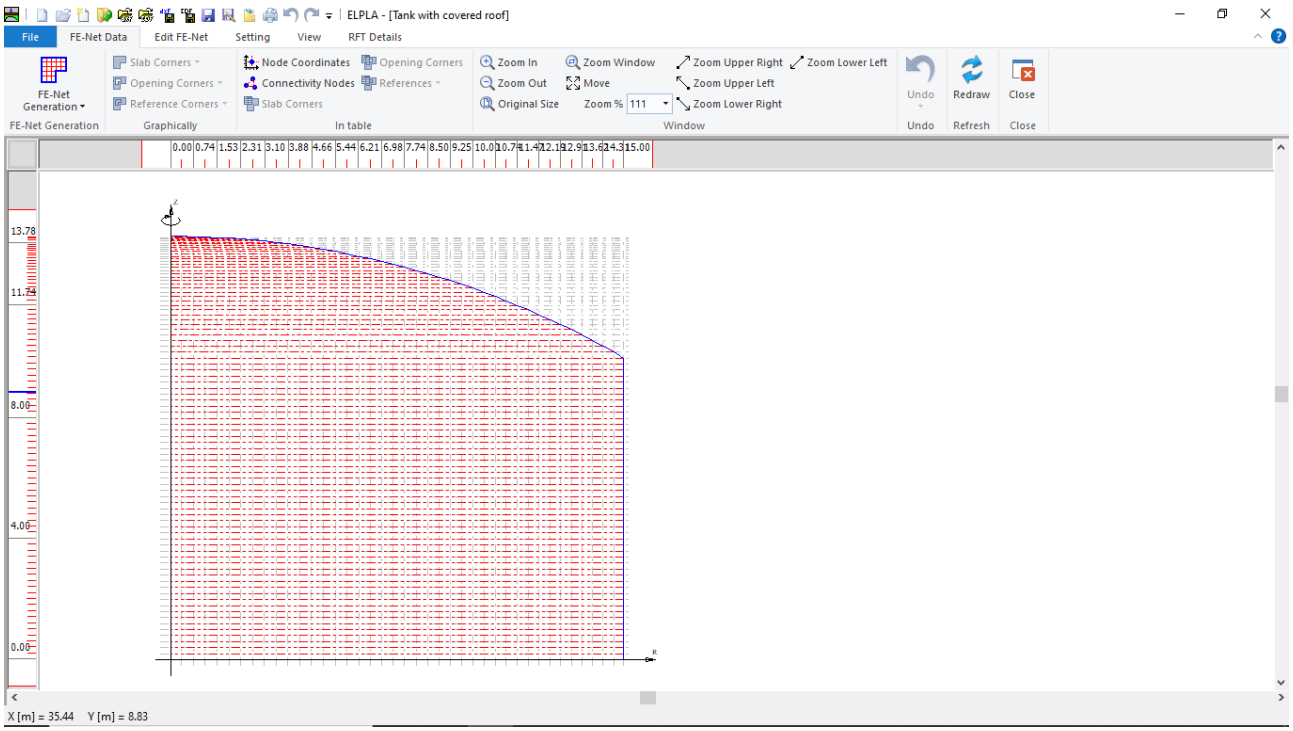


Figure 5.9 Generated FE-Net

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.

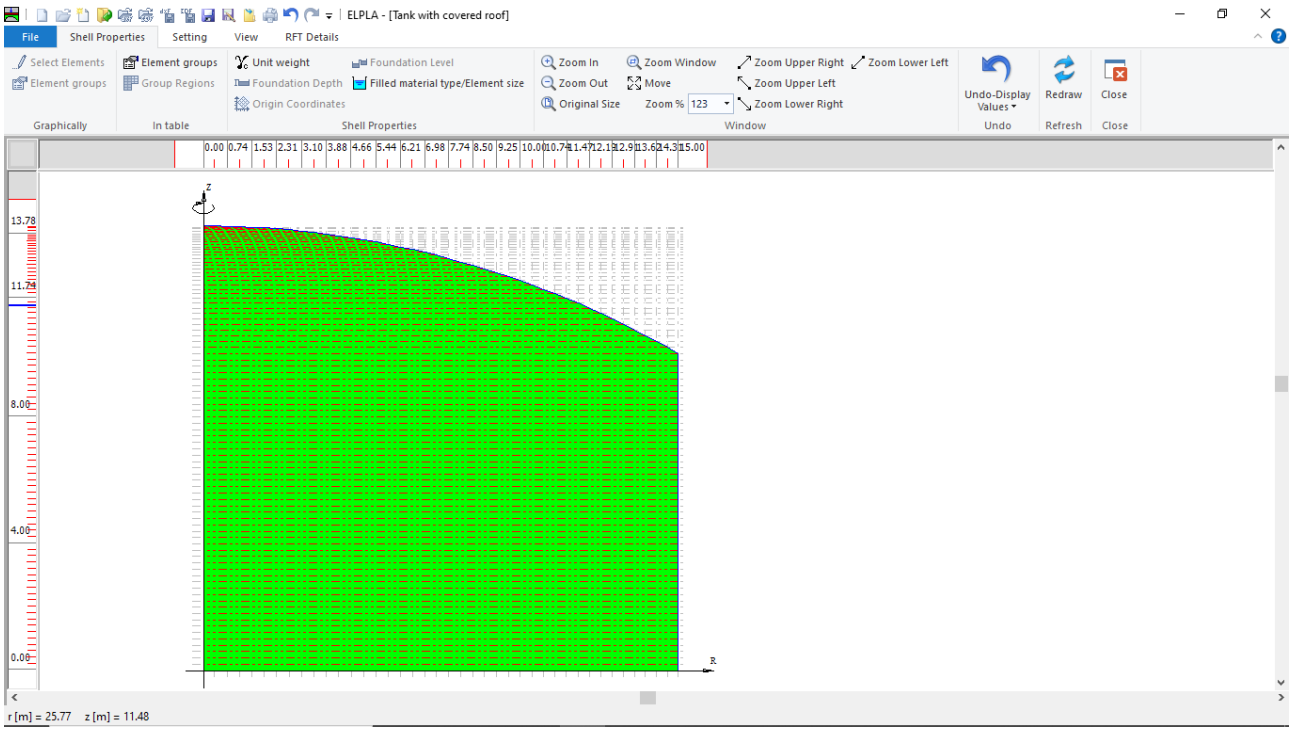

Figure 5.10 "Shell Properties" Window

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.

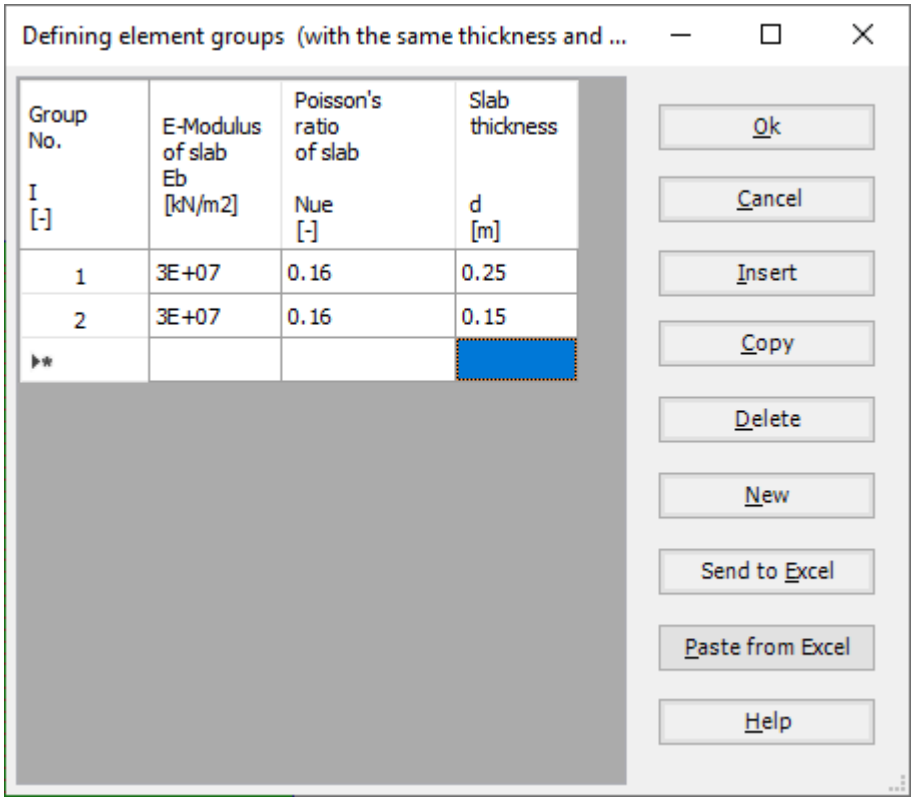


Figure 5.11 "Defining element groups" list box

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

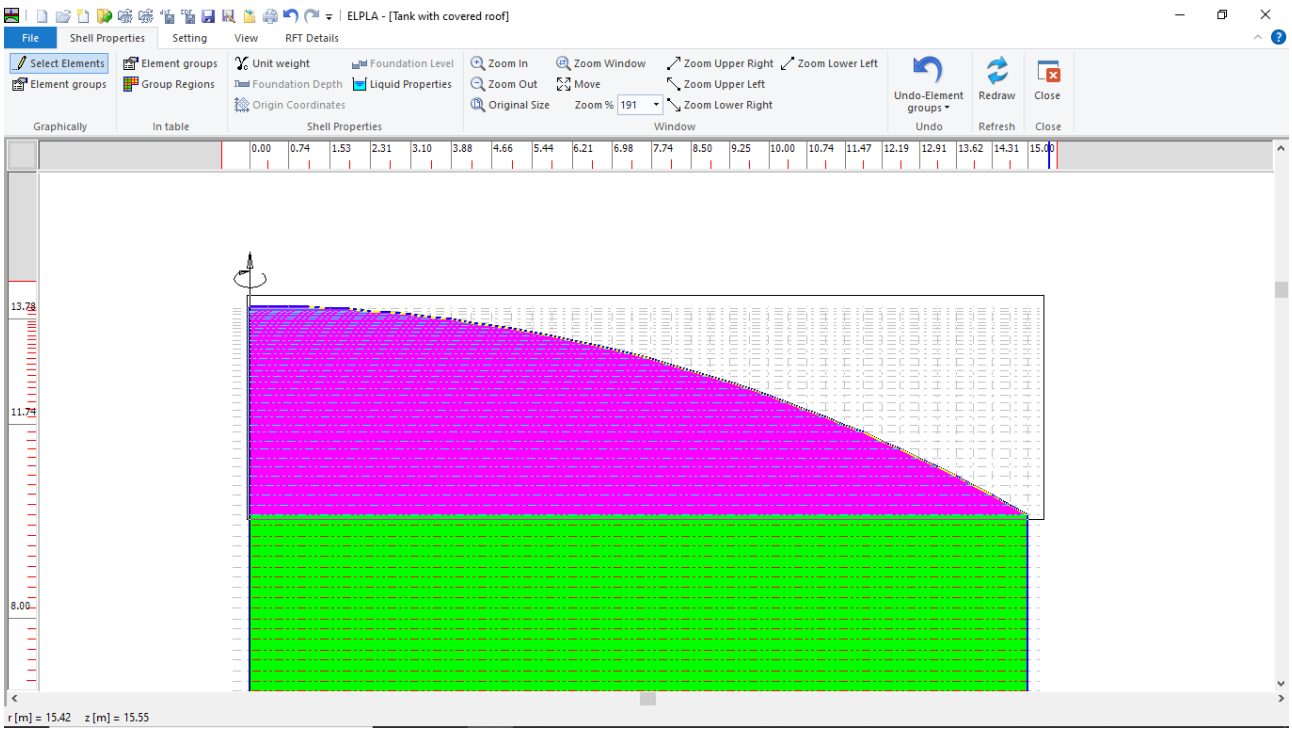


Figure 5.12 Selecting the nodes that include the tank roof


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.

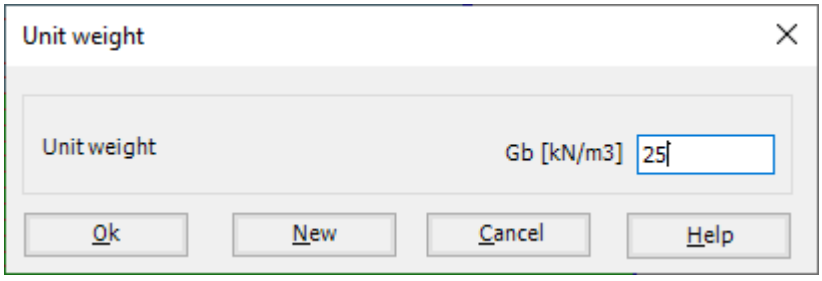


Figure 5.14 "Unit weight" dialog box

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

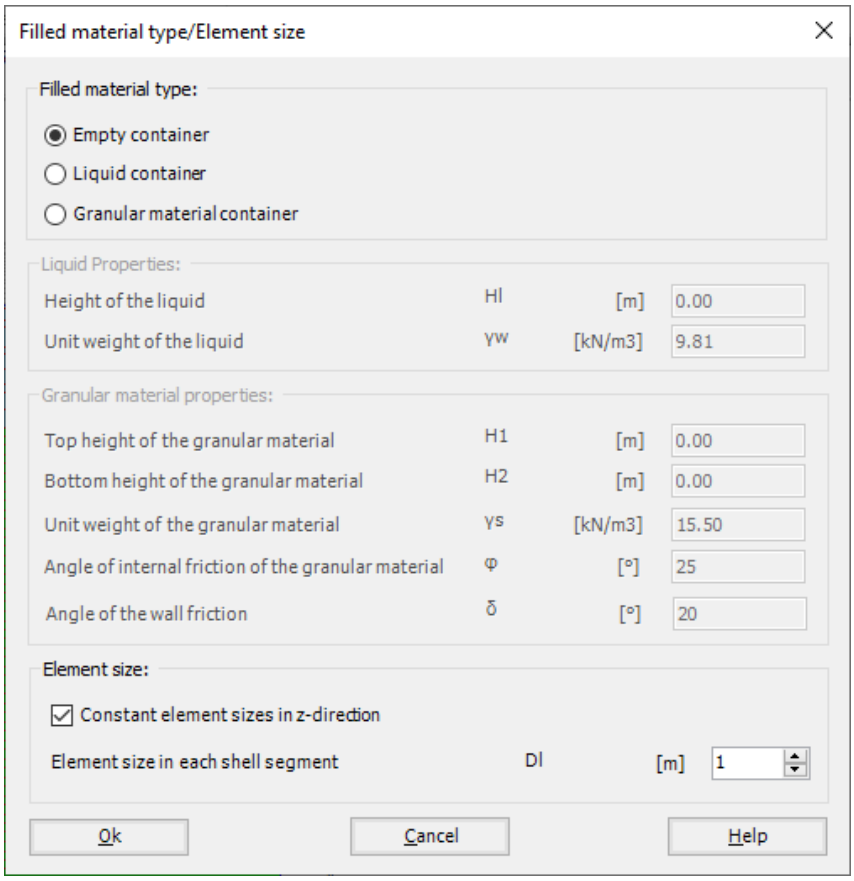


Figure 5.15 "Liquid Properties/Element size" dialog box

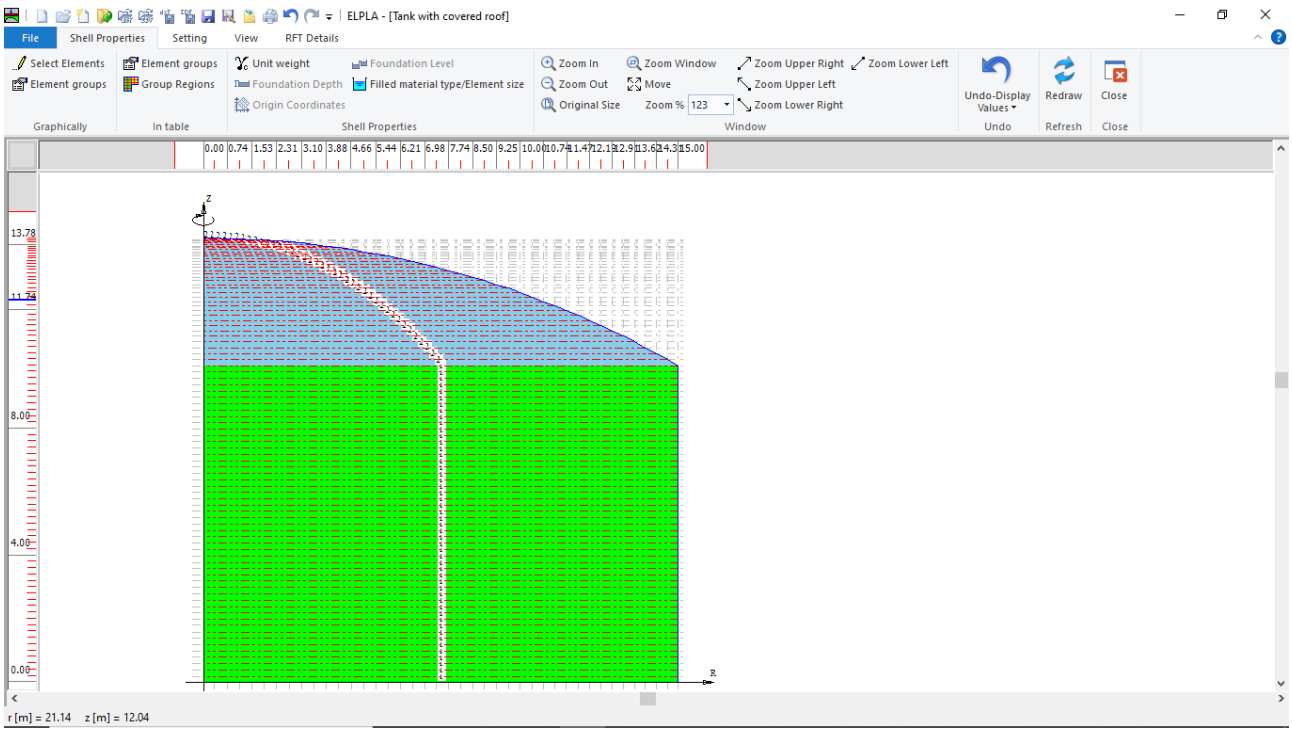


Figure 5.16 "Shell Properties" window after defining the shell data

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.

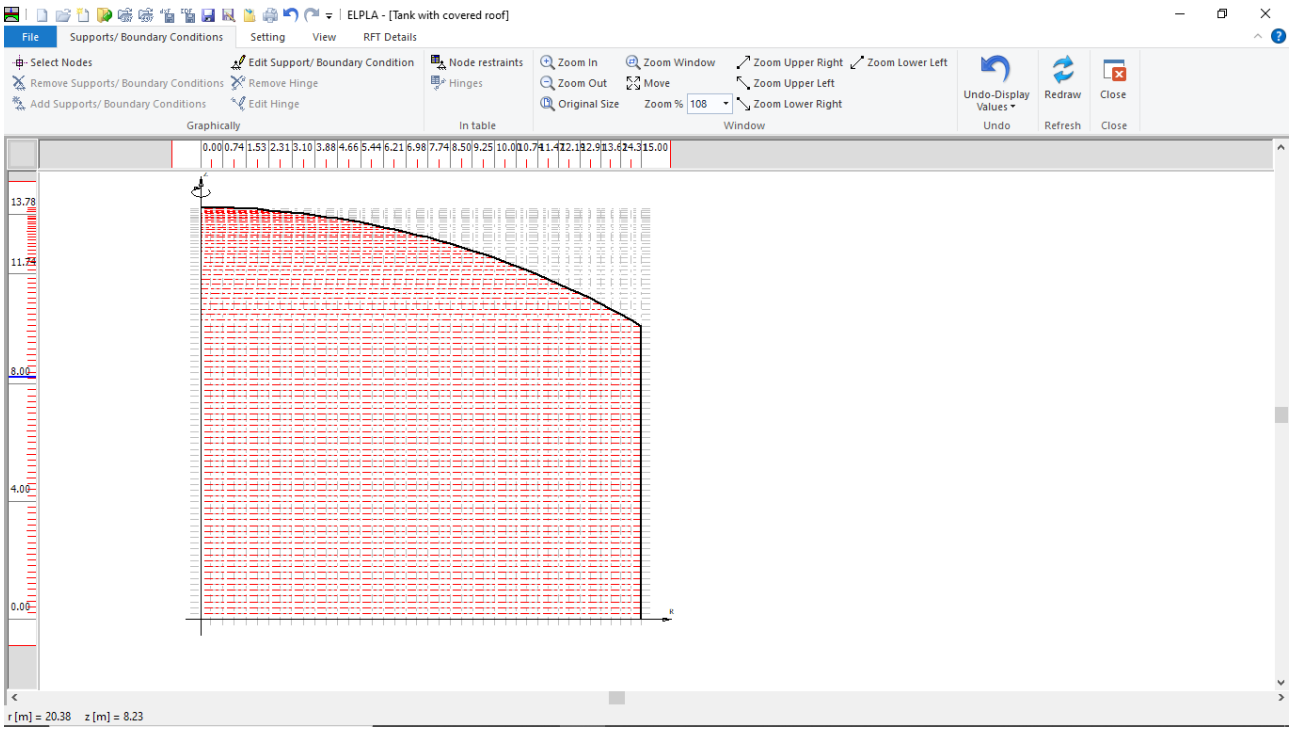


Figure 5.17 "Supports/ Boundary Conditions" Window

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.

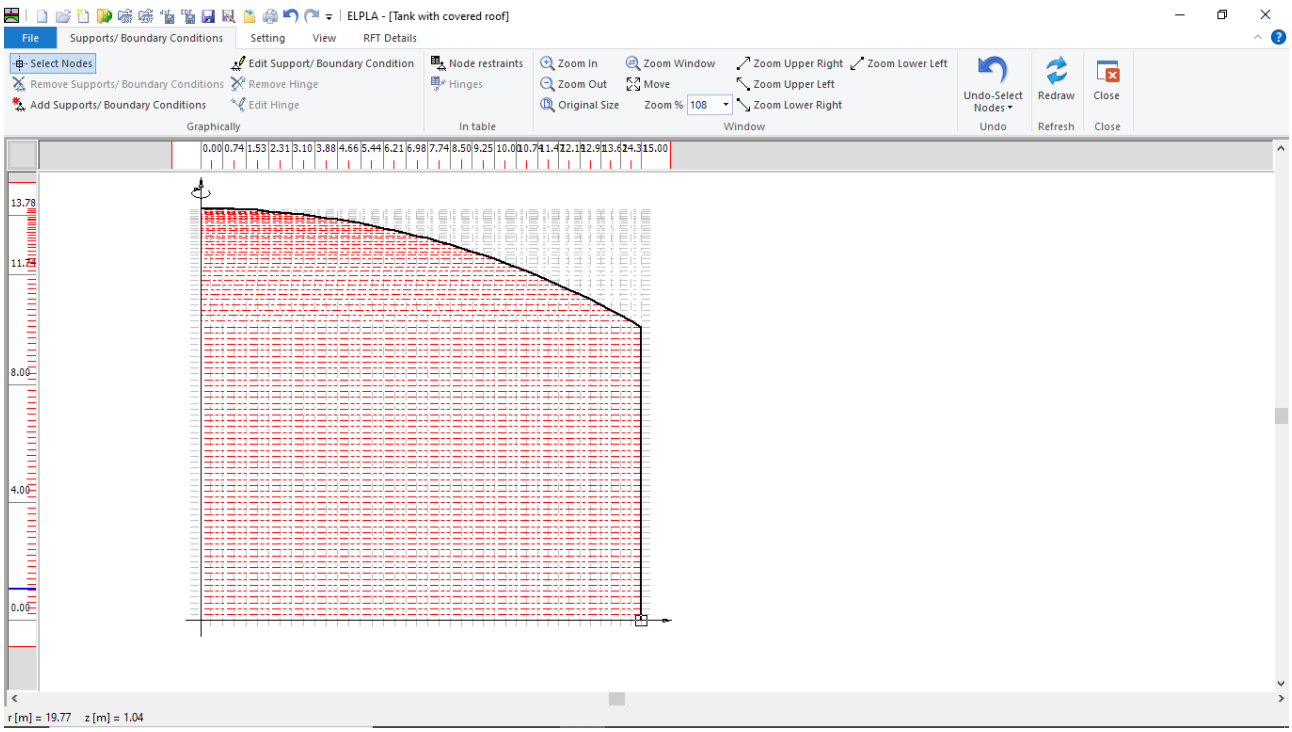


Figure 5.18 Selection of the node that has the fixed support

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

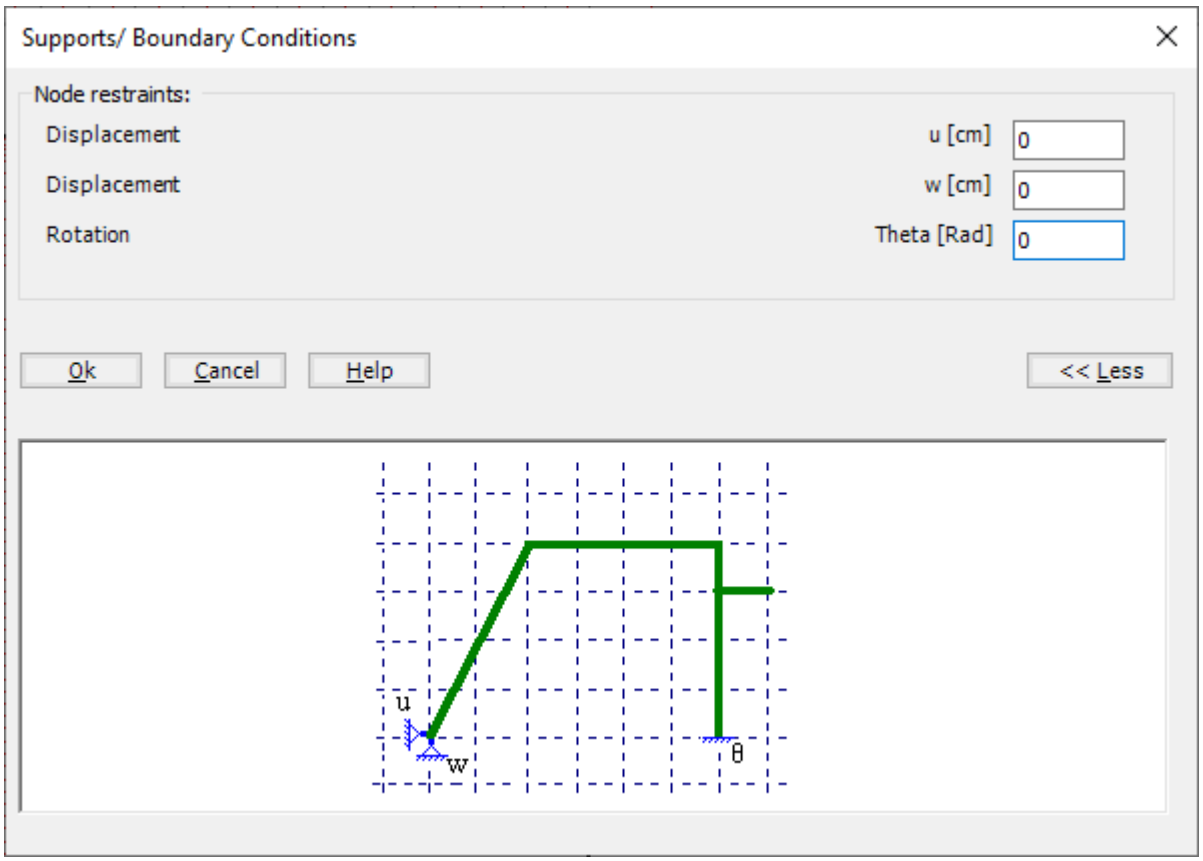


Figure 5.19 "Supports/ Boundary Conditions" dialog box

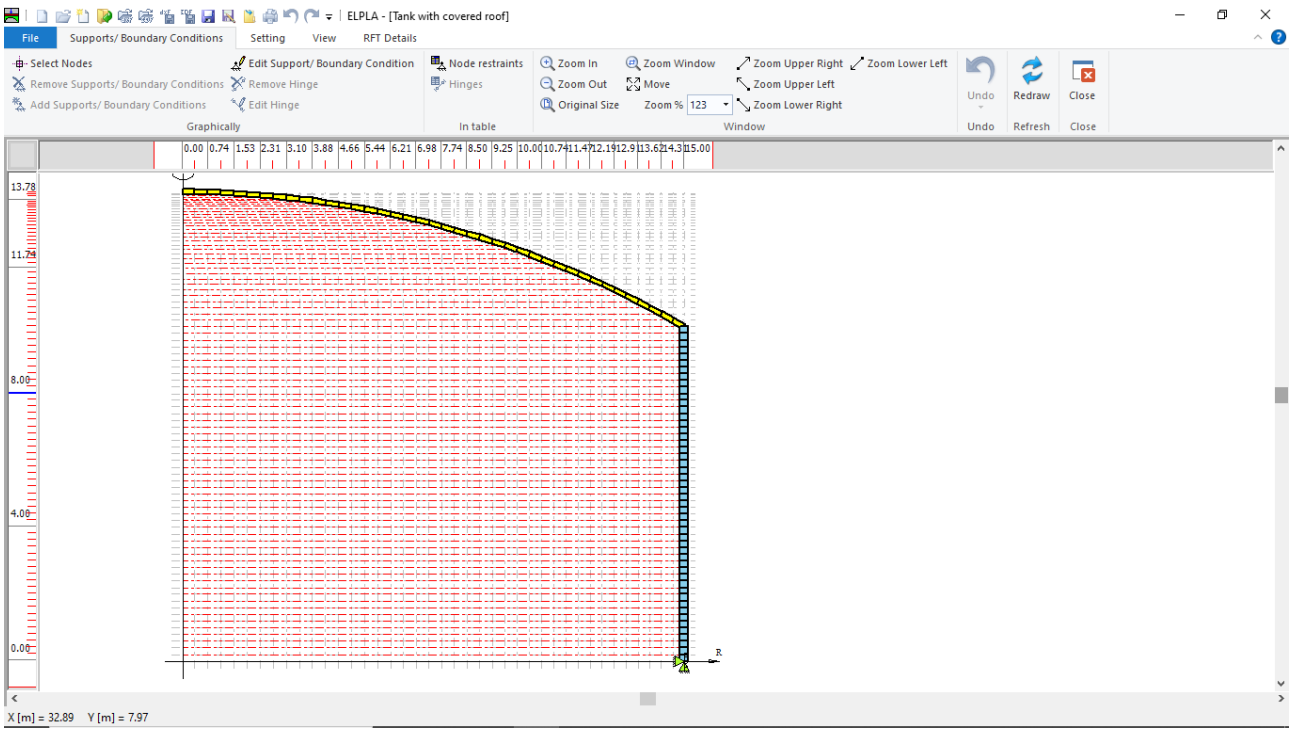


Figure 5.20 "Supports/ Boundary Conditions" window after defining the support

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.

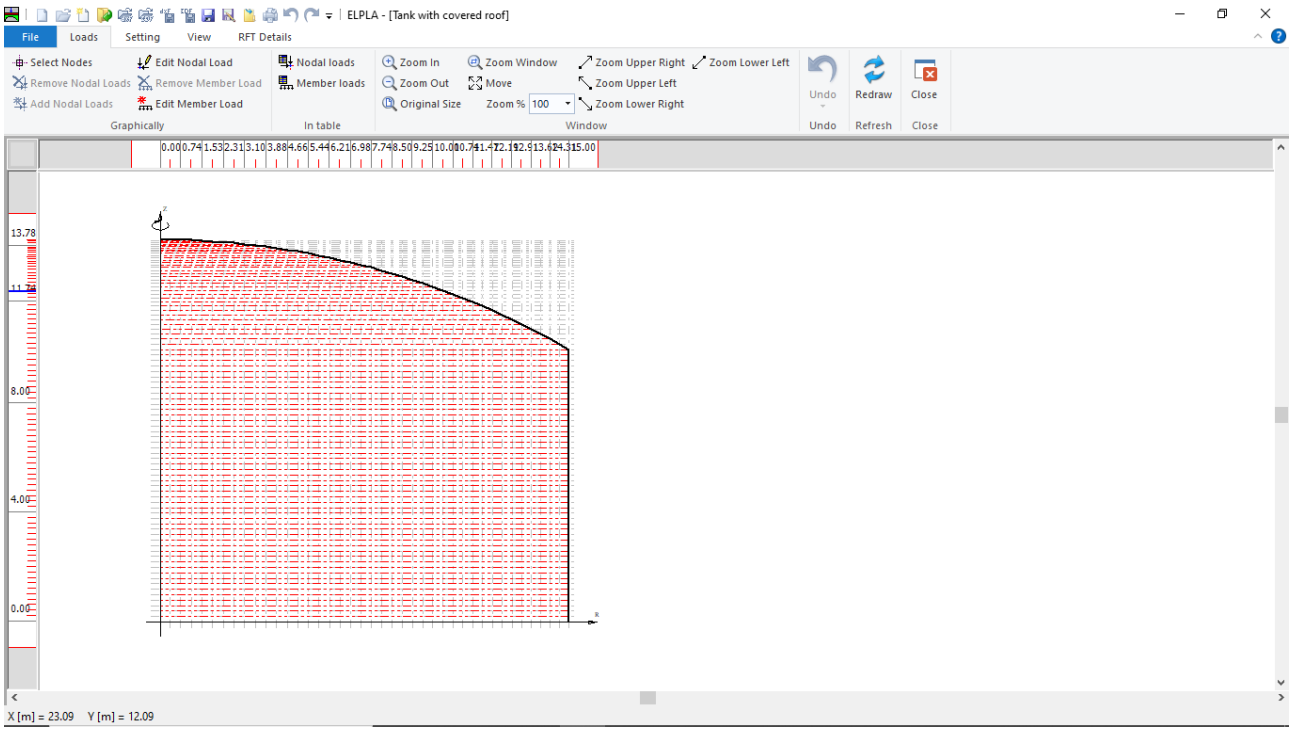


Figure 5.21 "Loads" Window

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.

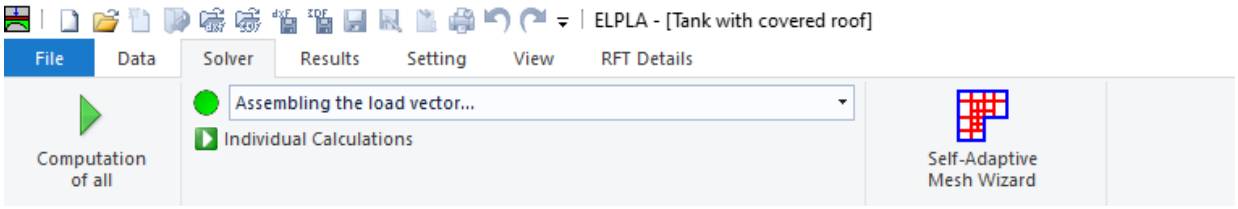


Figure 5.22 "Solver" Tab

*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.

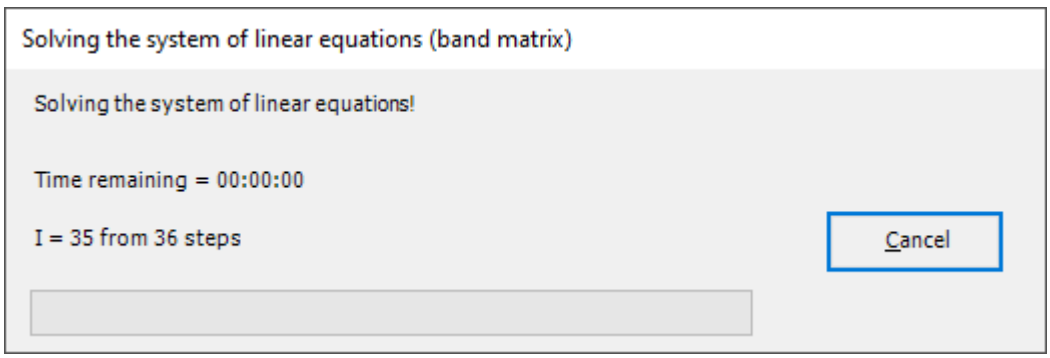


Figure 5.23 Analysis progress menu

**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.

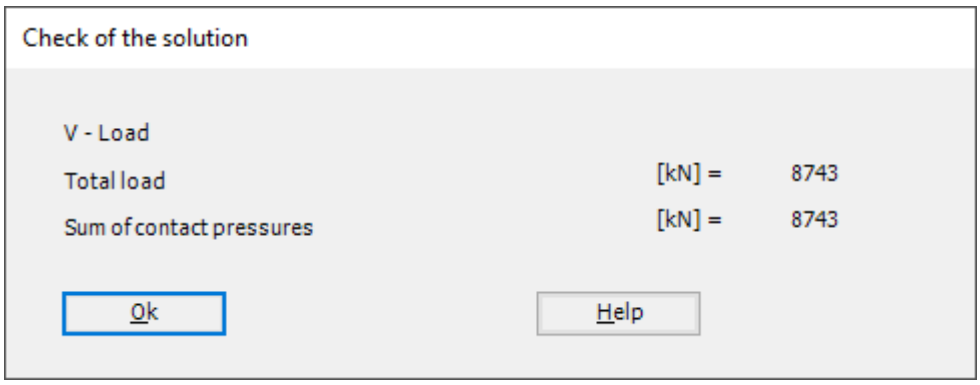


Figure 5.24 Menu "Check of the solution"

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).

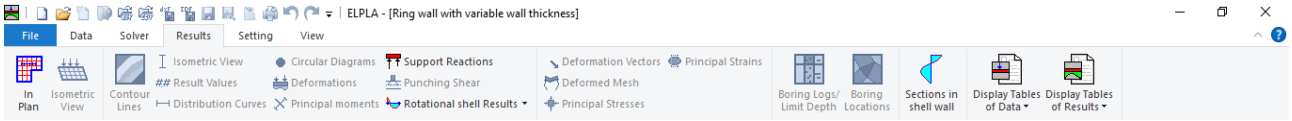


Figure 5.25 "Results" Tab

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

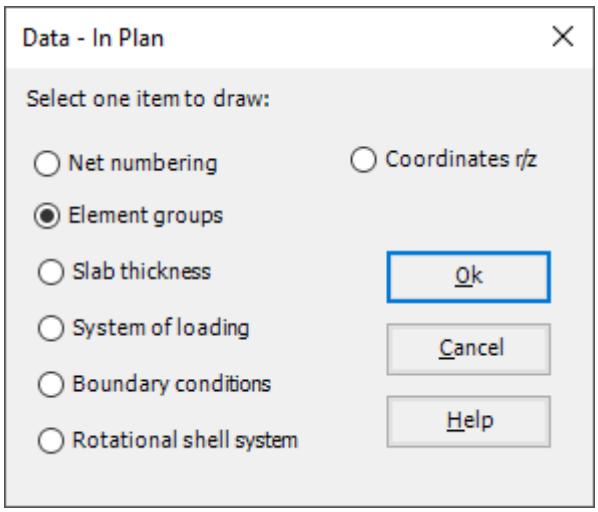


Figure 5.26 "Data – In Plan" option box

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

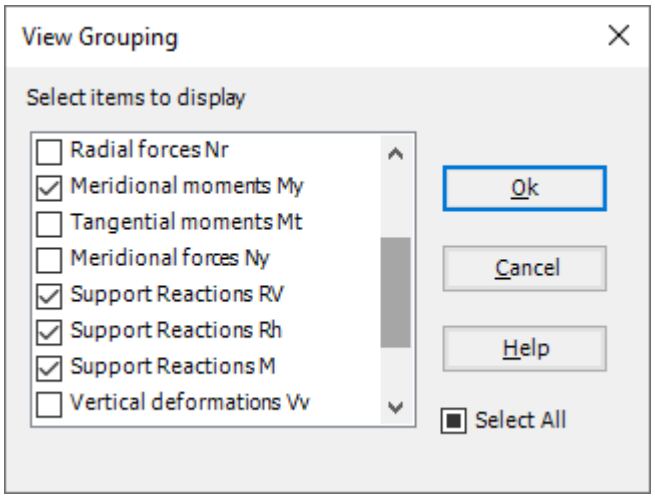


Figure 5.27 "View Grouping" check group box

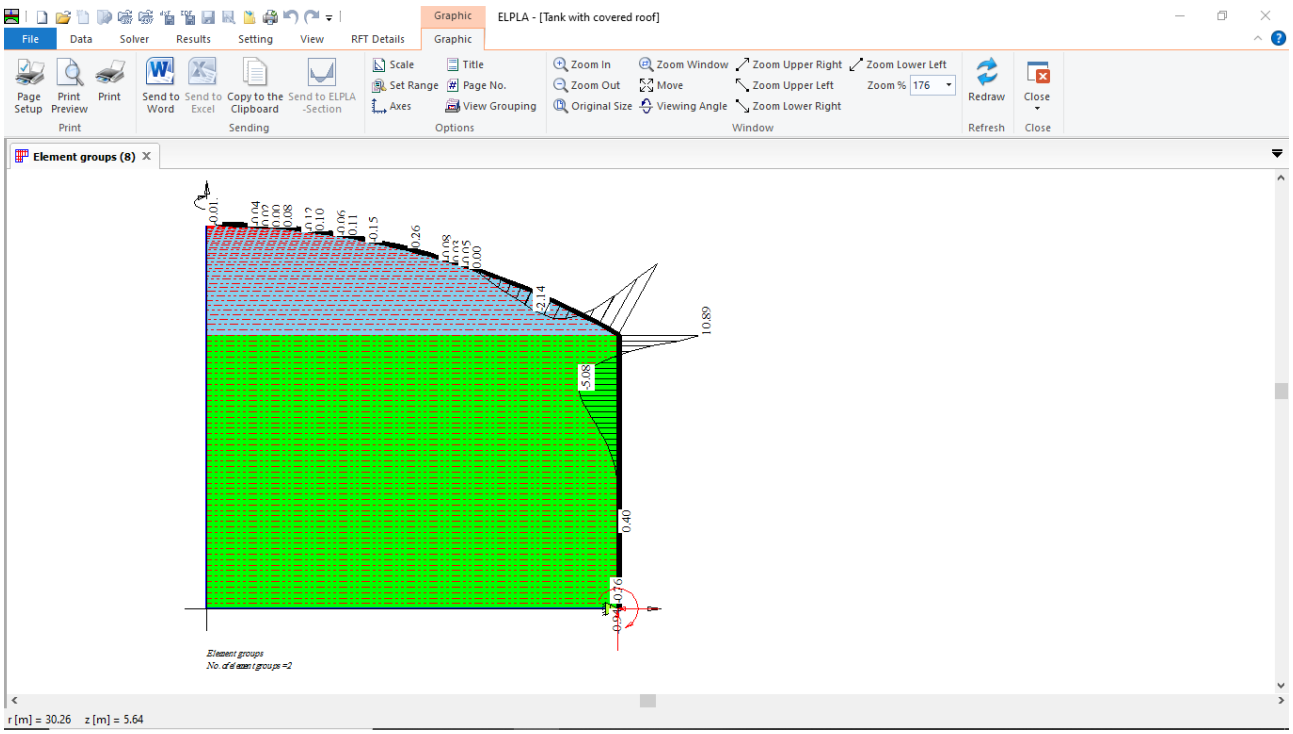


Figure 5.28 Element groups of the tank