Example 2

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

Contents Page

1 Description of the problem

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

2 Geometry and properties

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

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

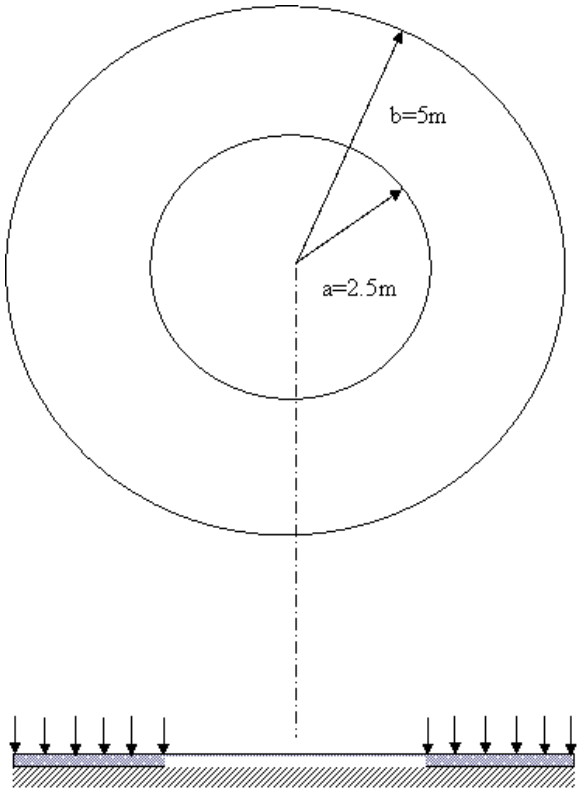

Figure 2.1 Annular plate subjected to a uniform load

3 Analysis of the plate

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

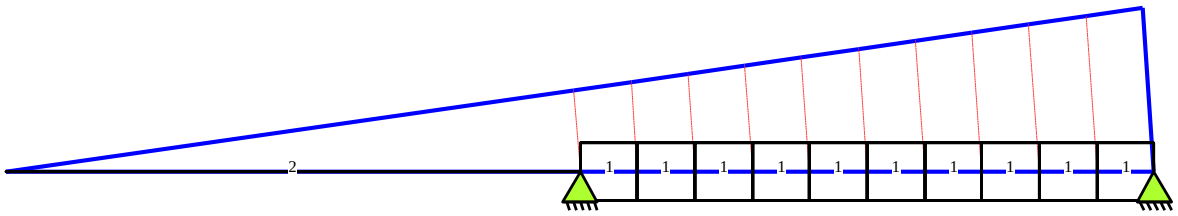

Figure 2.2 Annular plate with 10 annular regions and supports

4 Creating the project

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

4.1 Calculation method

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

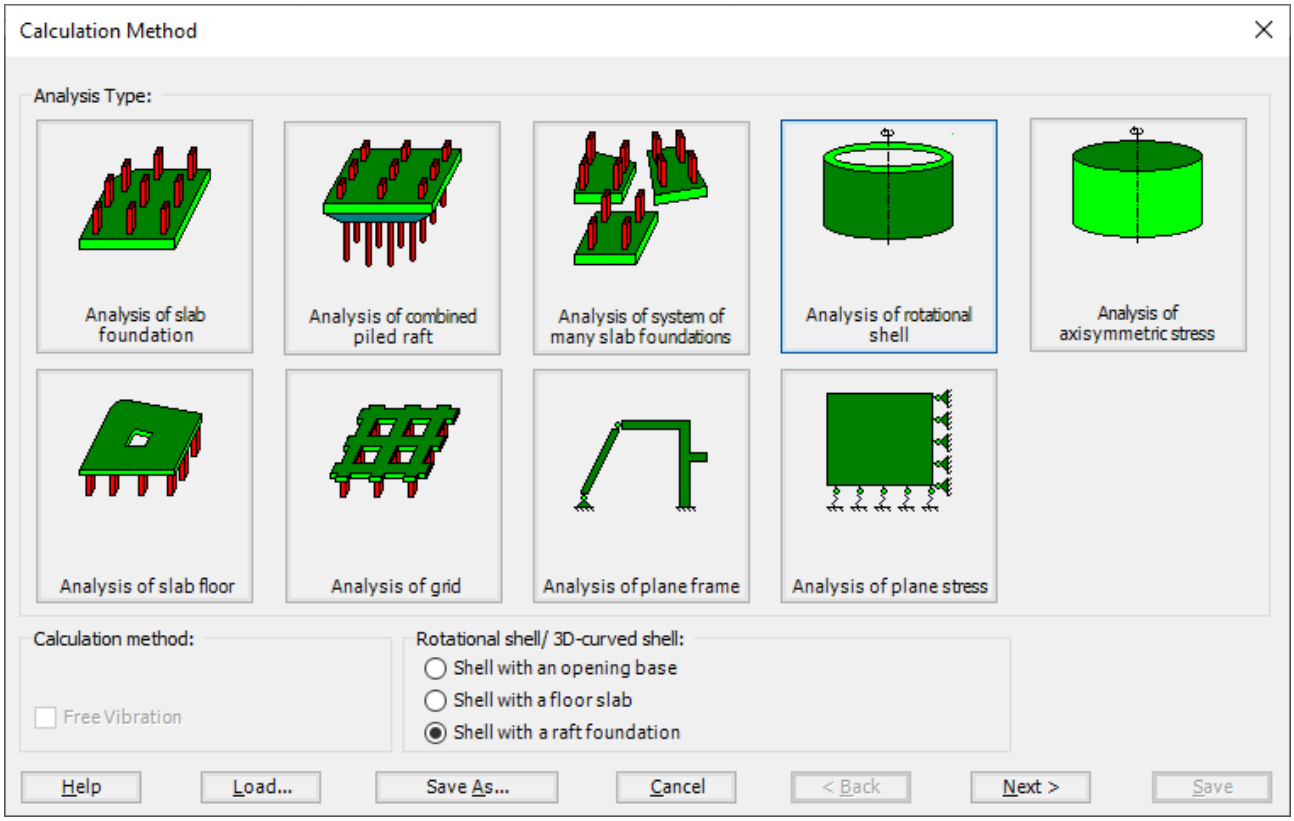

Figure 2.3 "Analysis Type" Form

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

To define the calculation method:

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

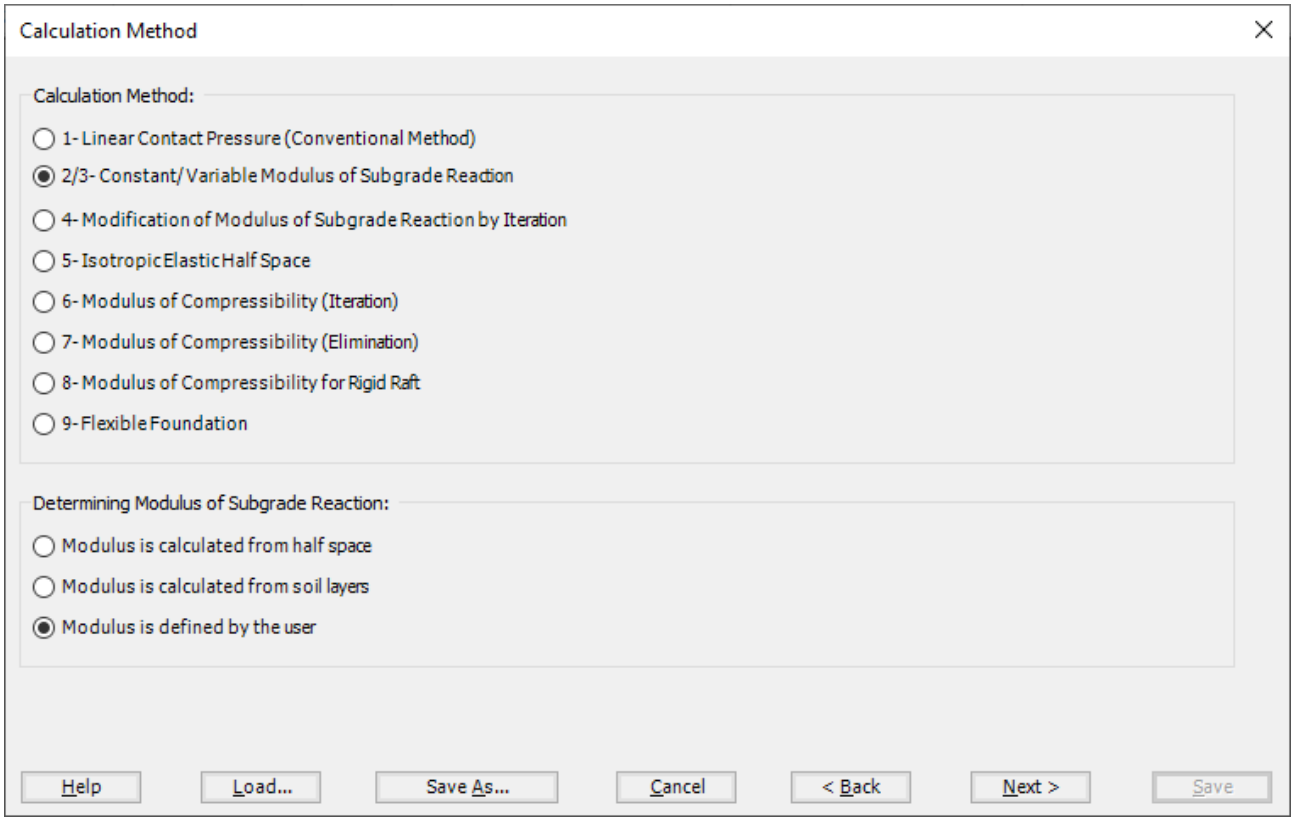

Figure 2.4 "Calculation Methods" Form

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

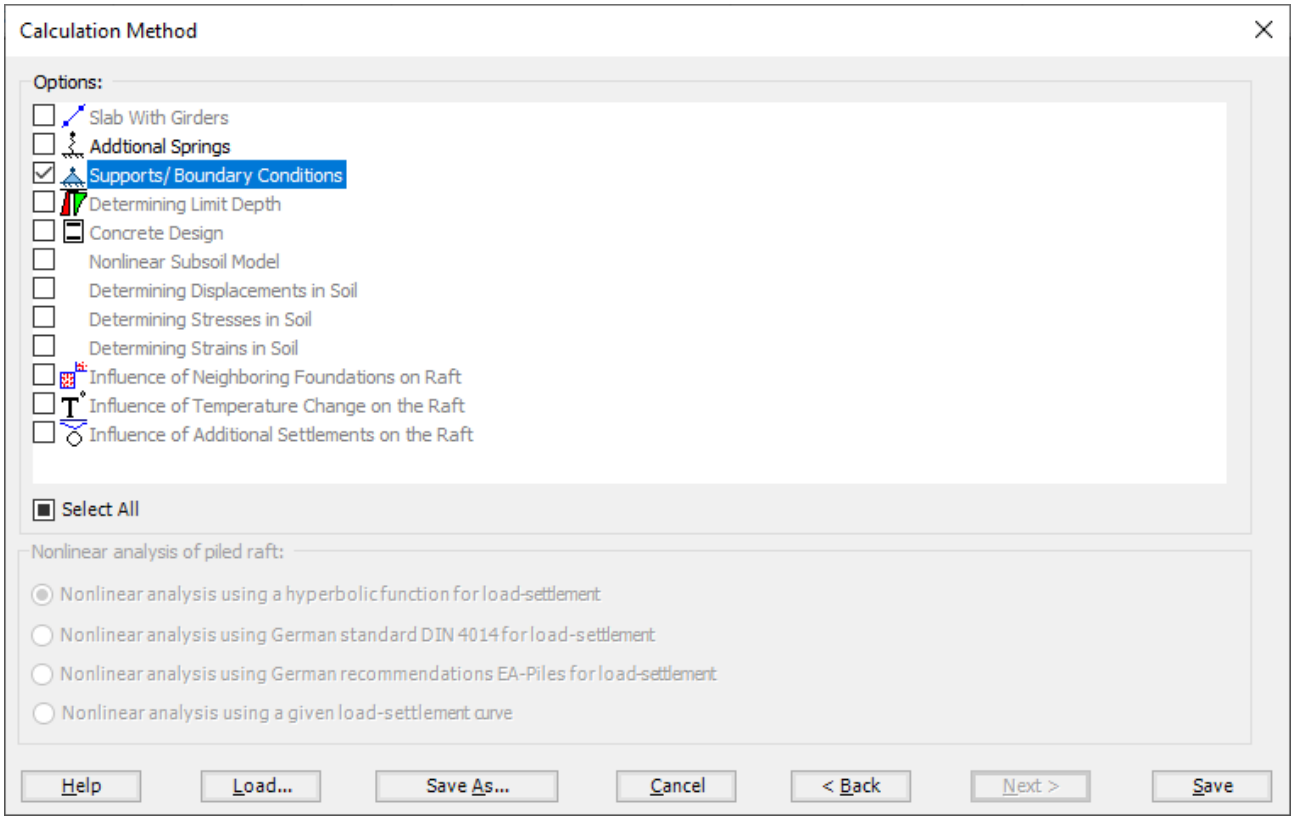

Figure 2.5 "Options" Form

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

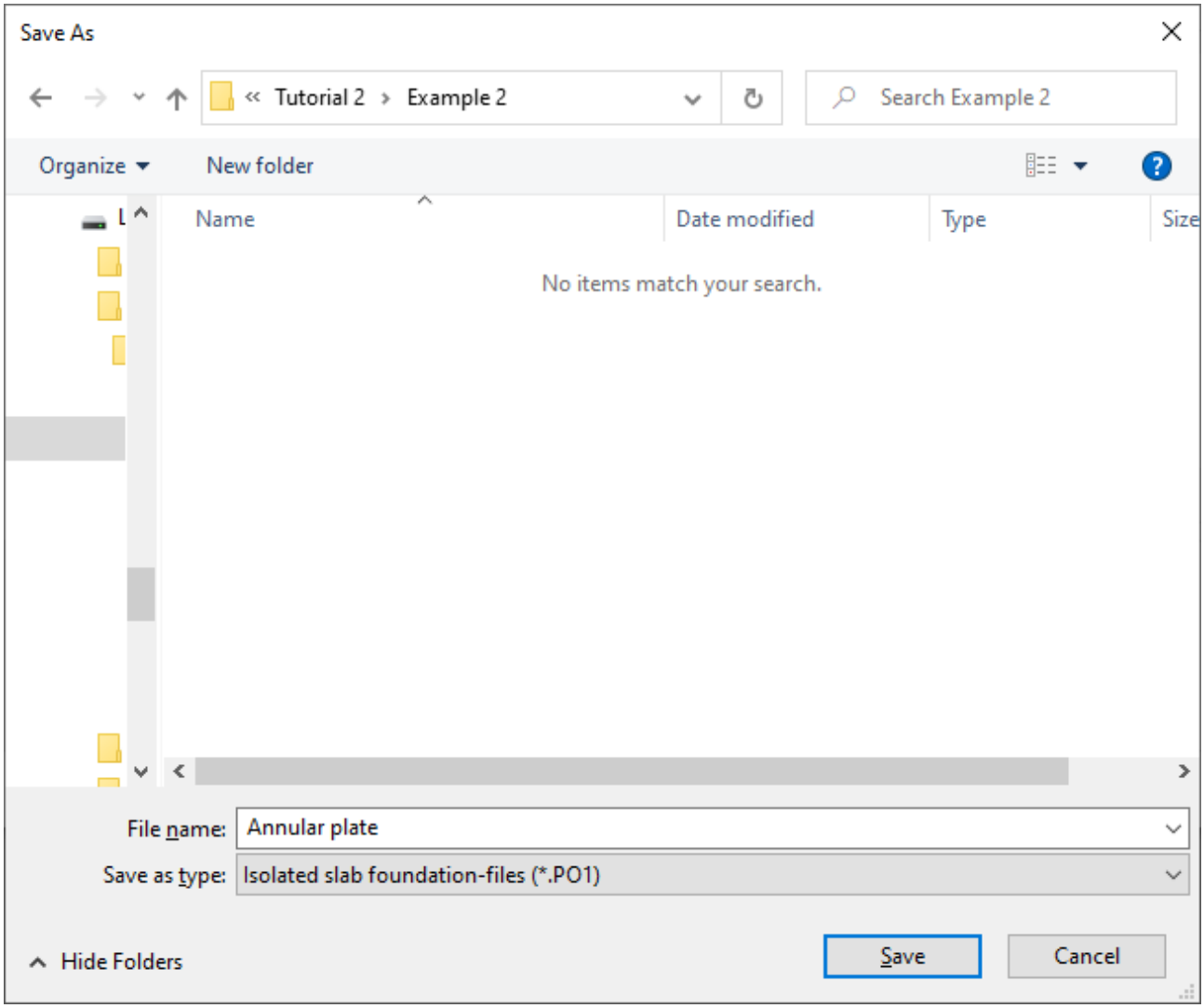

Figure 2.6 "Save as" dialog box

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

4.2 Project identification

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

In this dialog box

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

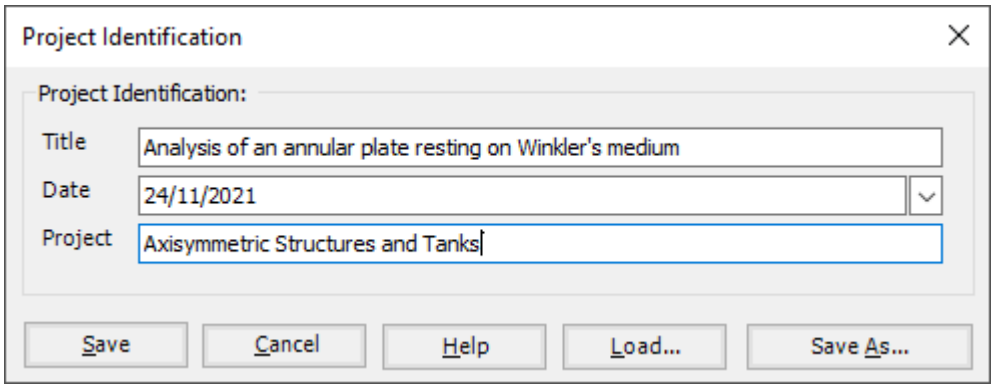

Figure 2.7 "Project Identification" dialog box

4.3 FE-Net data

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

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

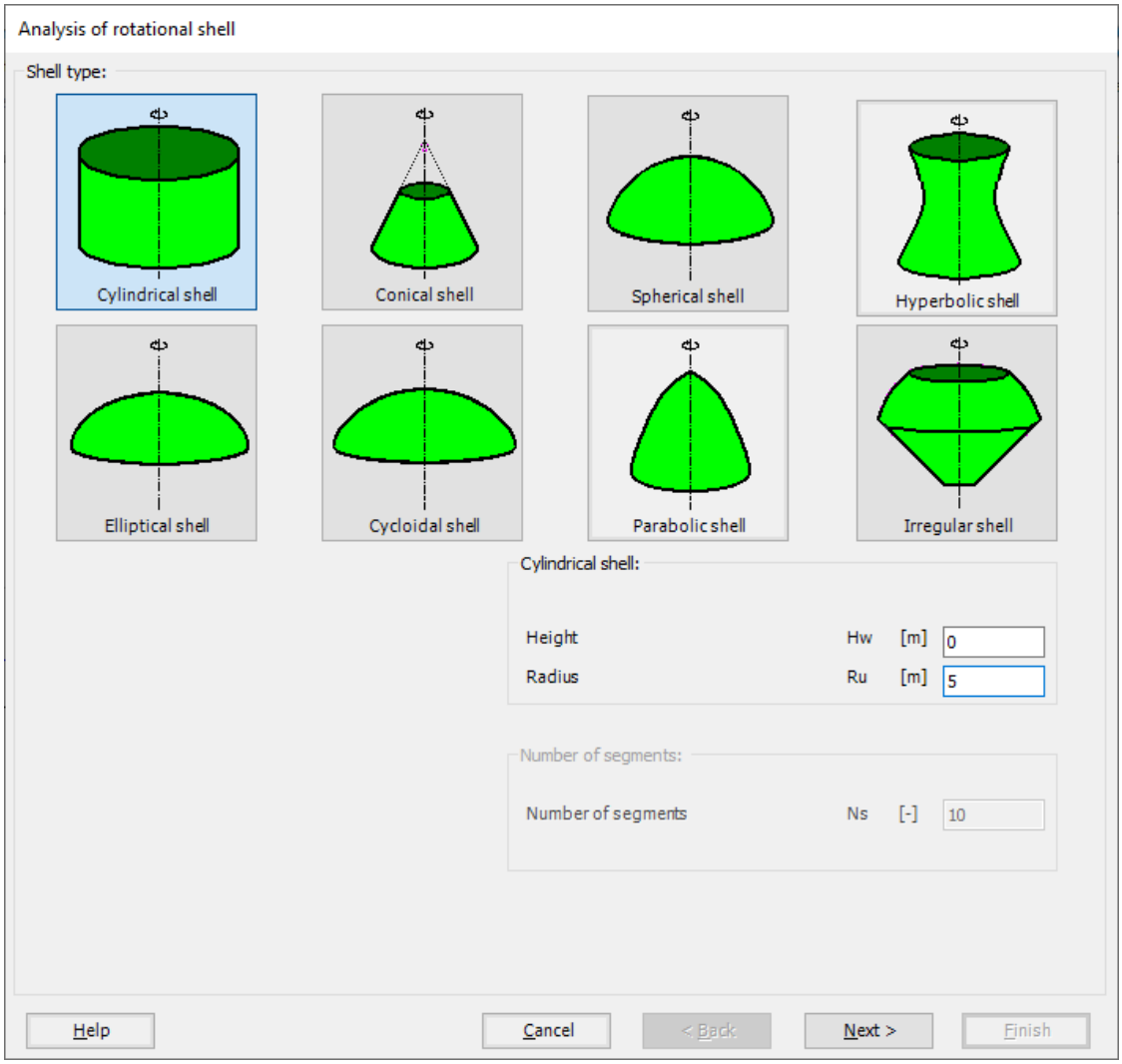

Figure 2.8 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

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

-

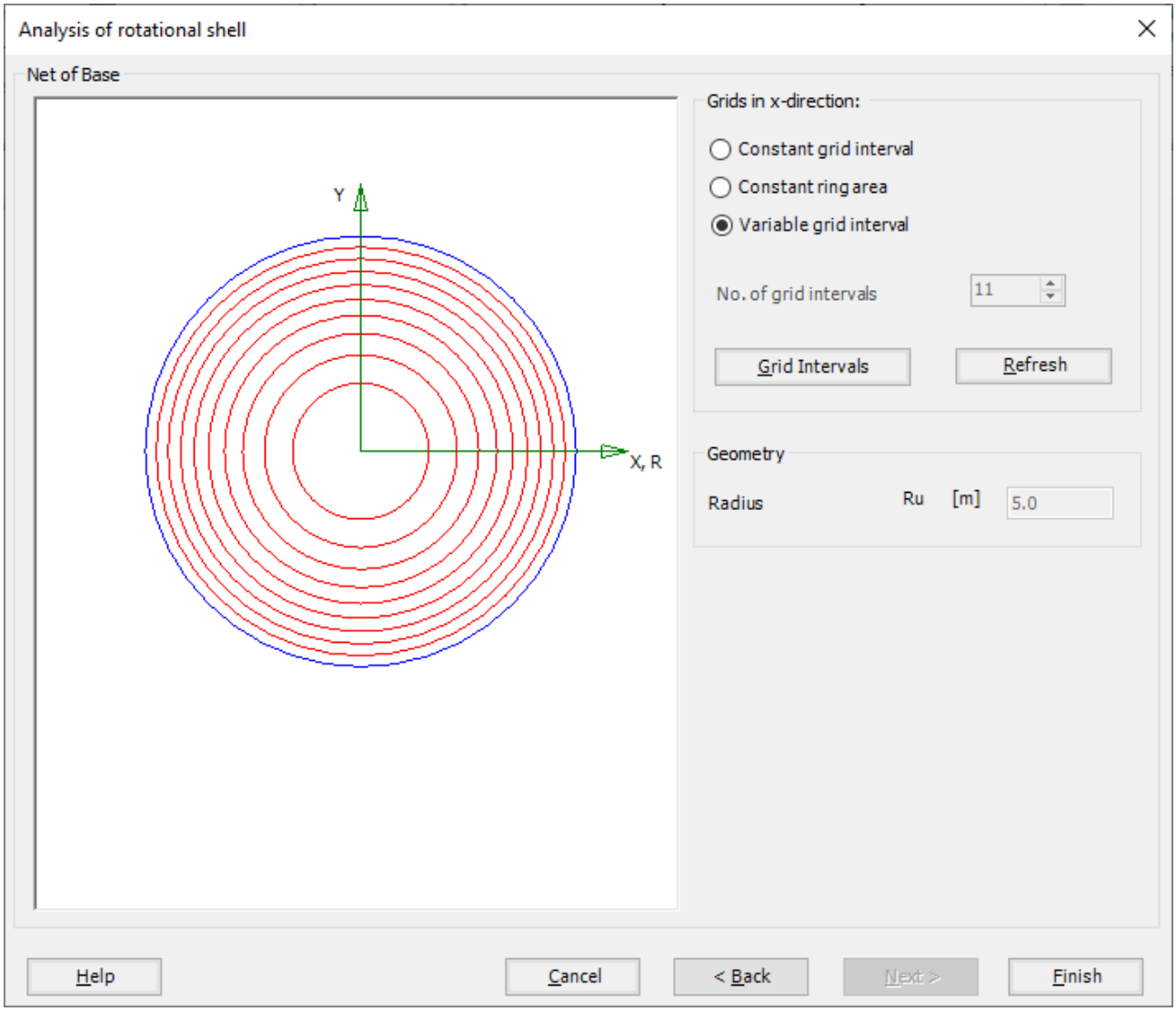

Figure 2.9 "Net of Base" Form

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

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

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

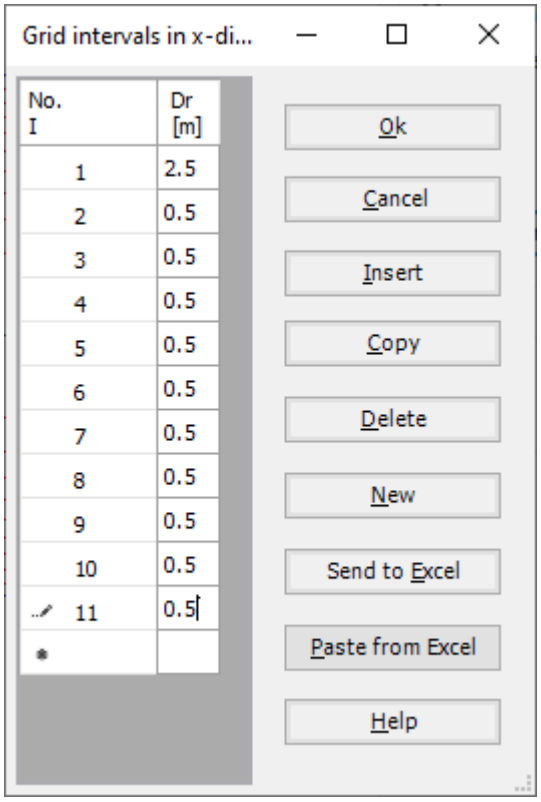

Figure 2.10 "Grid intervals in *x*-direction" Form

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

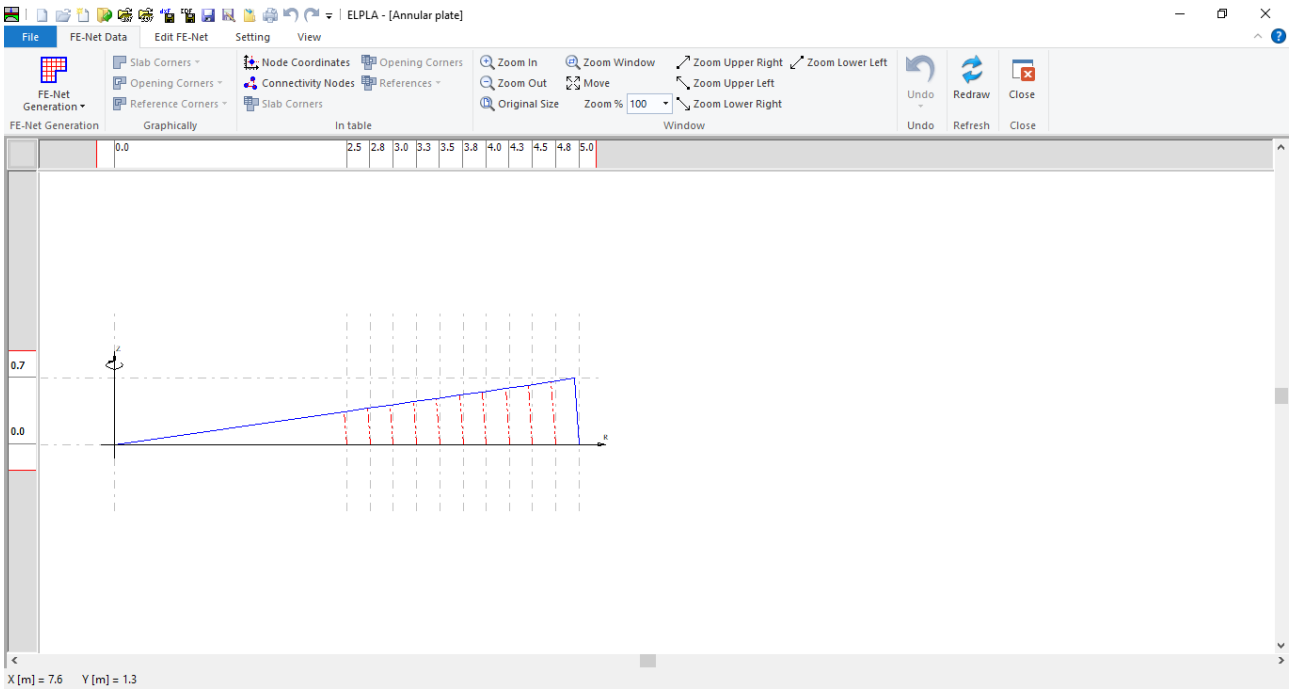

Figure 2.11 Generated FE-Net

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

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

4.4 Soil Properties

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

Soil data

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

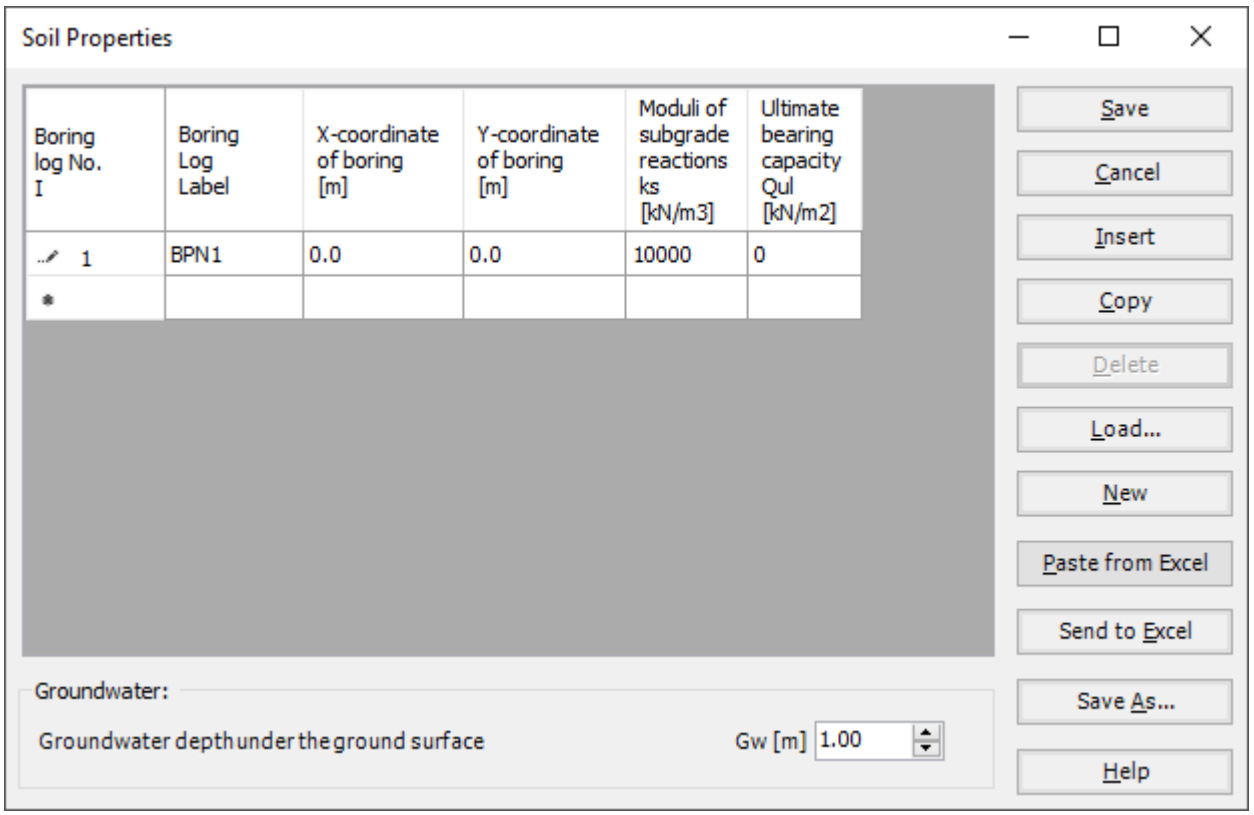

Figure 2.12 "Soil Properties" form

4.5 Shell properties

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

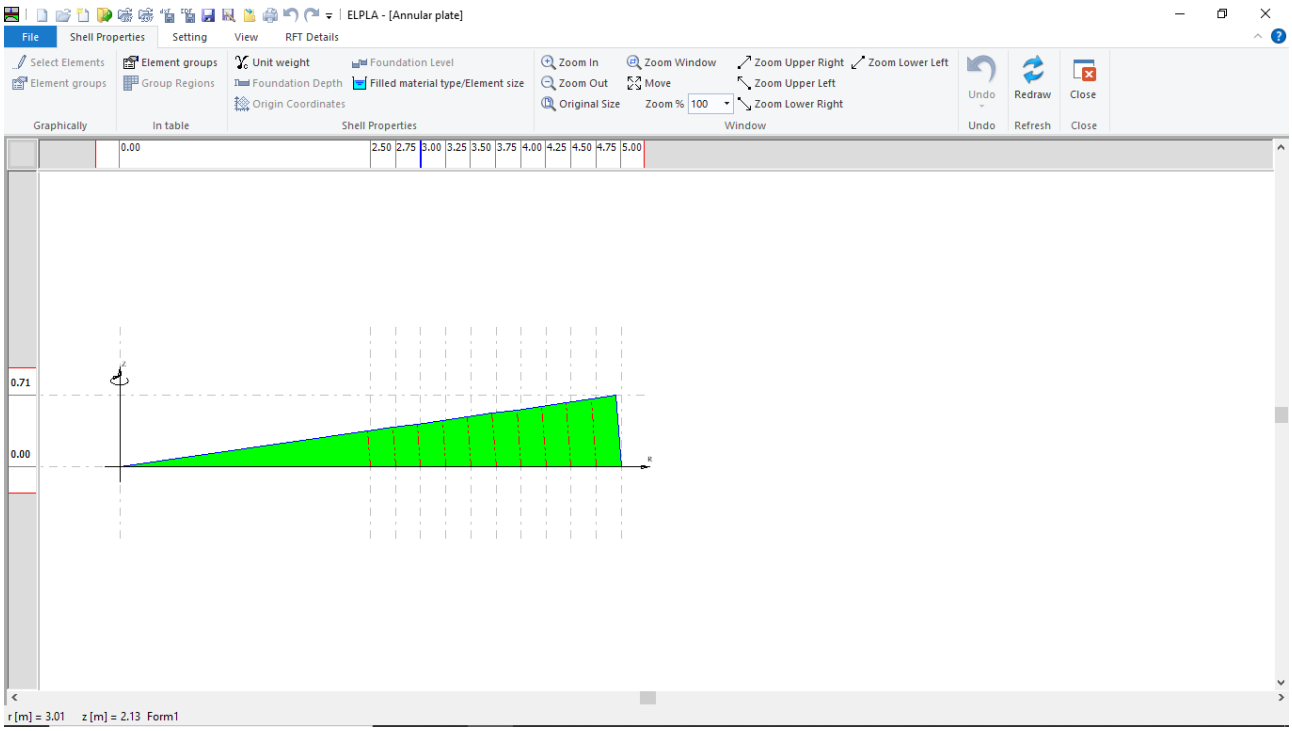

Figure 2.13 "Shell Properties" Window

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

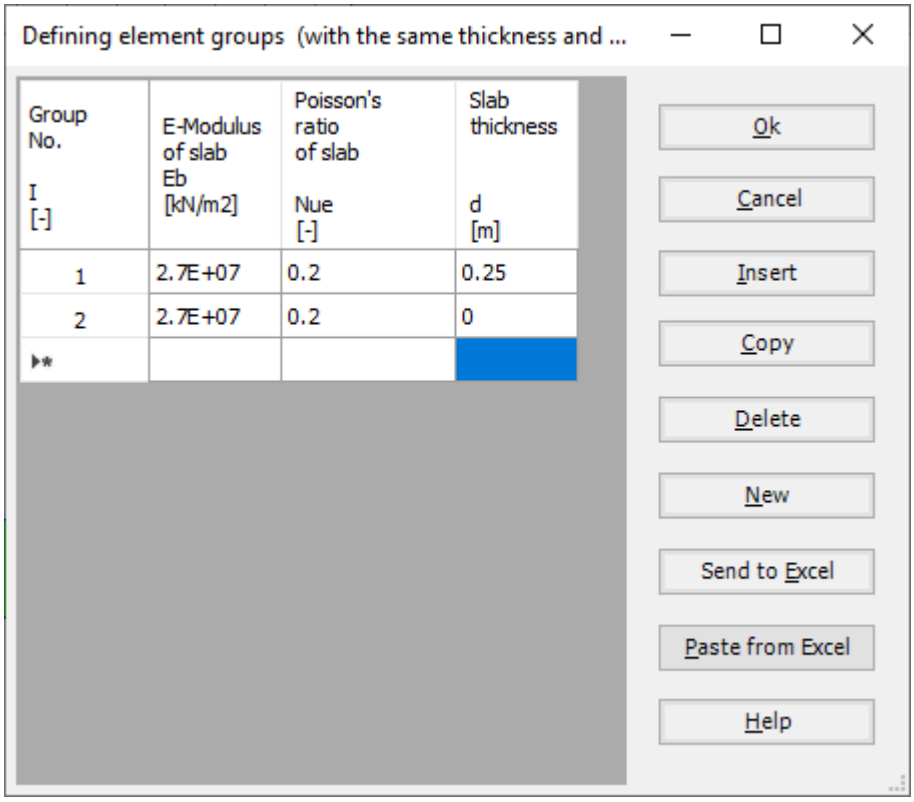

Figure 2.14 "Defining element groups" list box

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

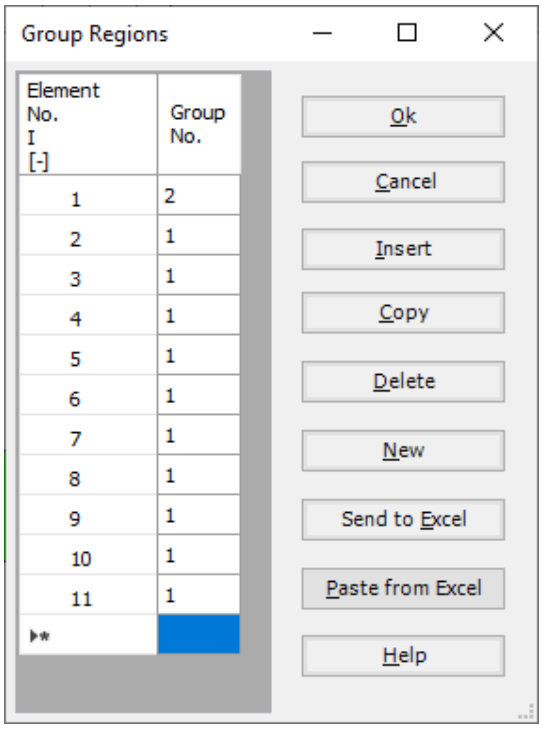

Figure 2.15 "Group Regions" Form

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

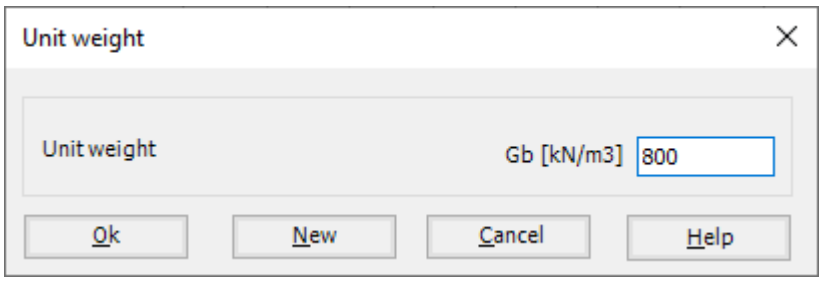

Figure 2.16 "Unit weight" dialog box

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

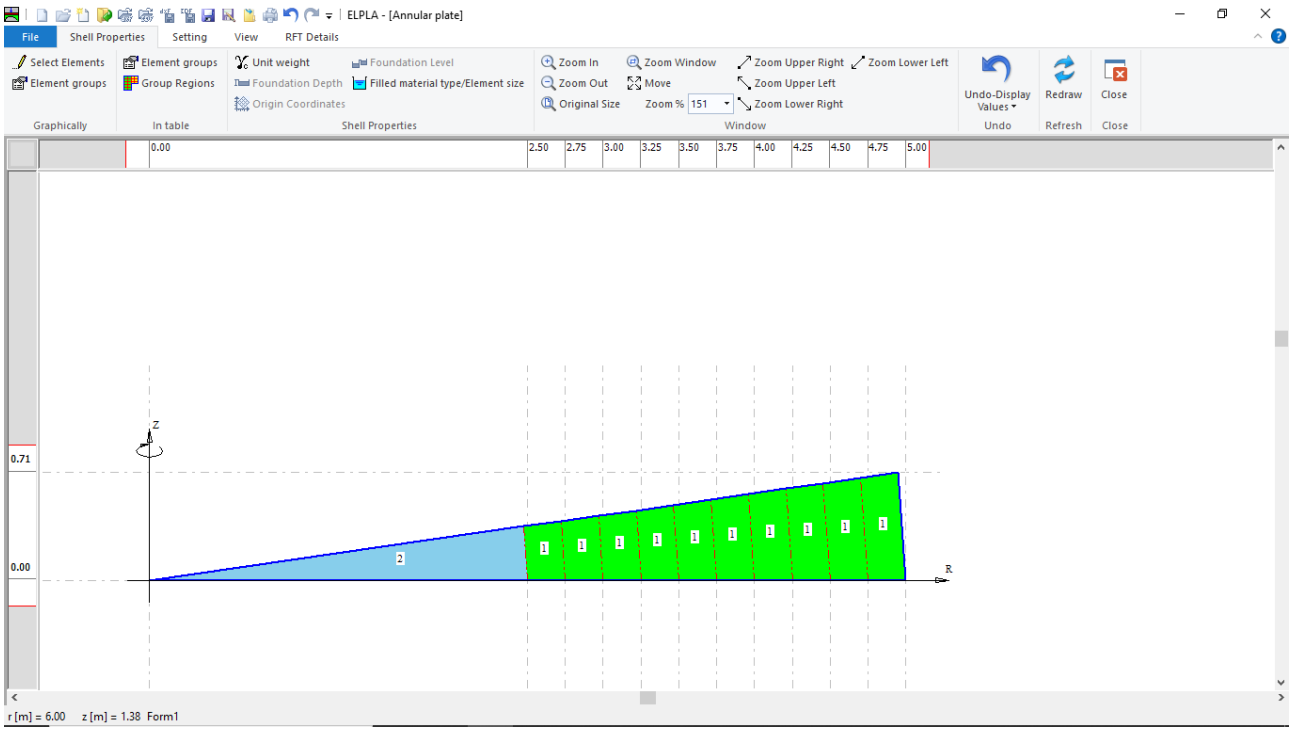

Figure 2.17 "Shell Properties" Window

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

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

4.6 Supports/ boundary conditions

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

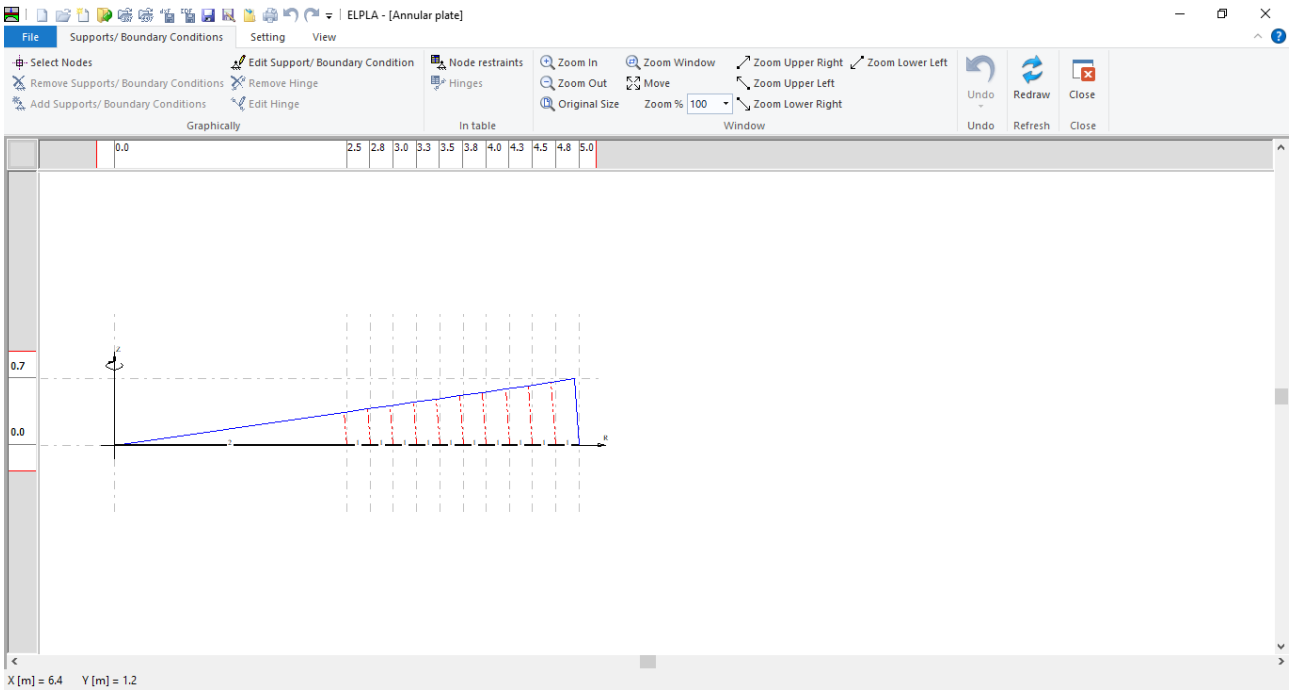

Figure 2.18 "Supports/ Boundary Conditions" Window

To define the supports

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

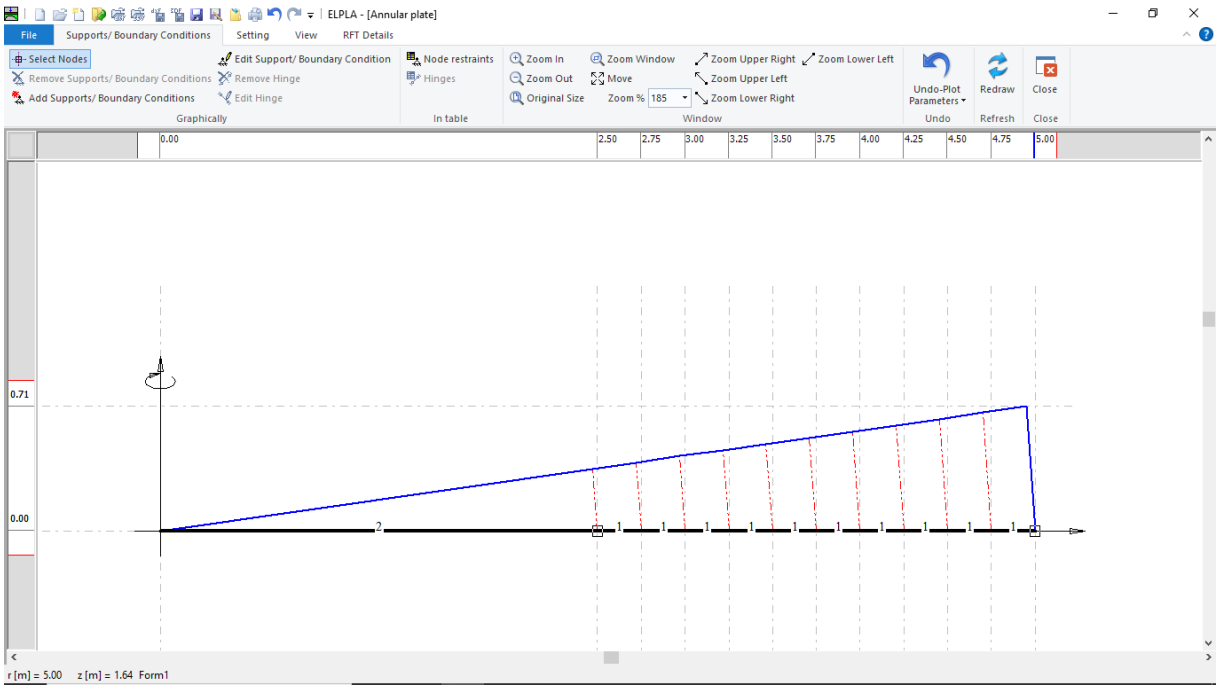

Figure 2.19 Selection of nodes that have supports

In this dialog box

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

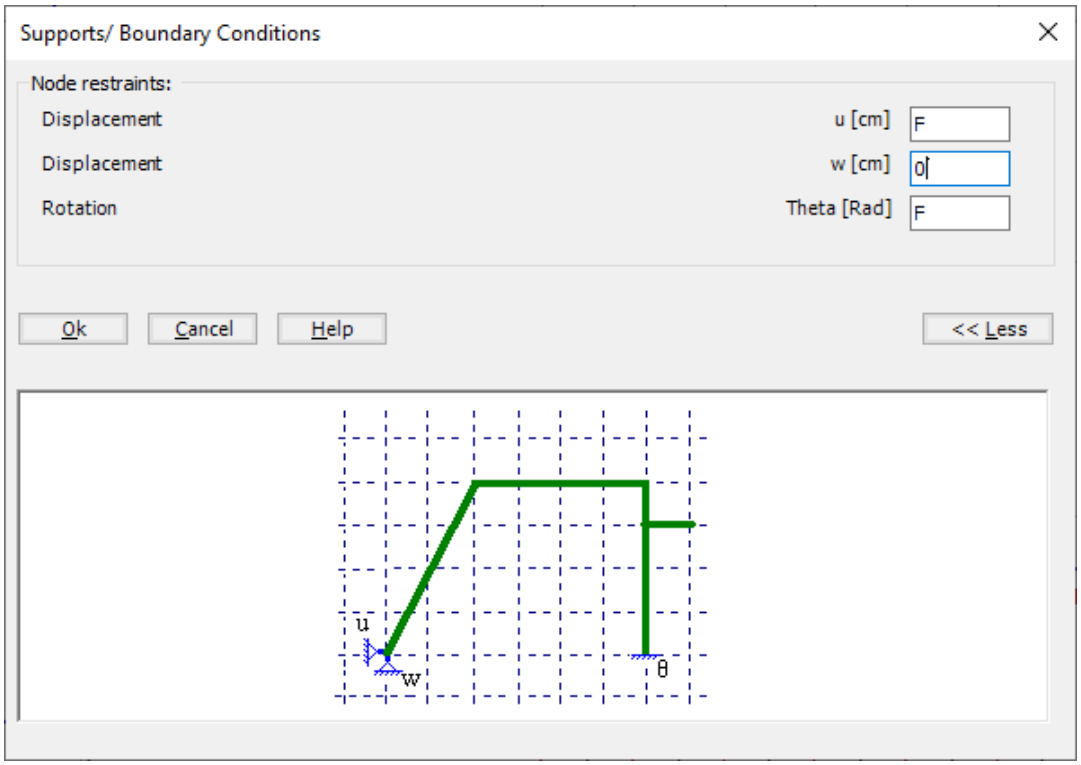

Figure 2.20 "Supports/ Boundary Conditions" dialog box

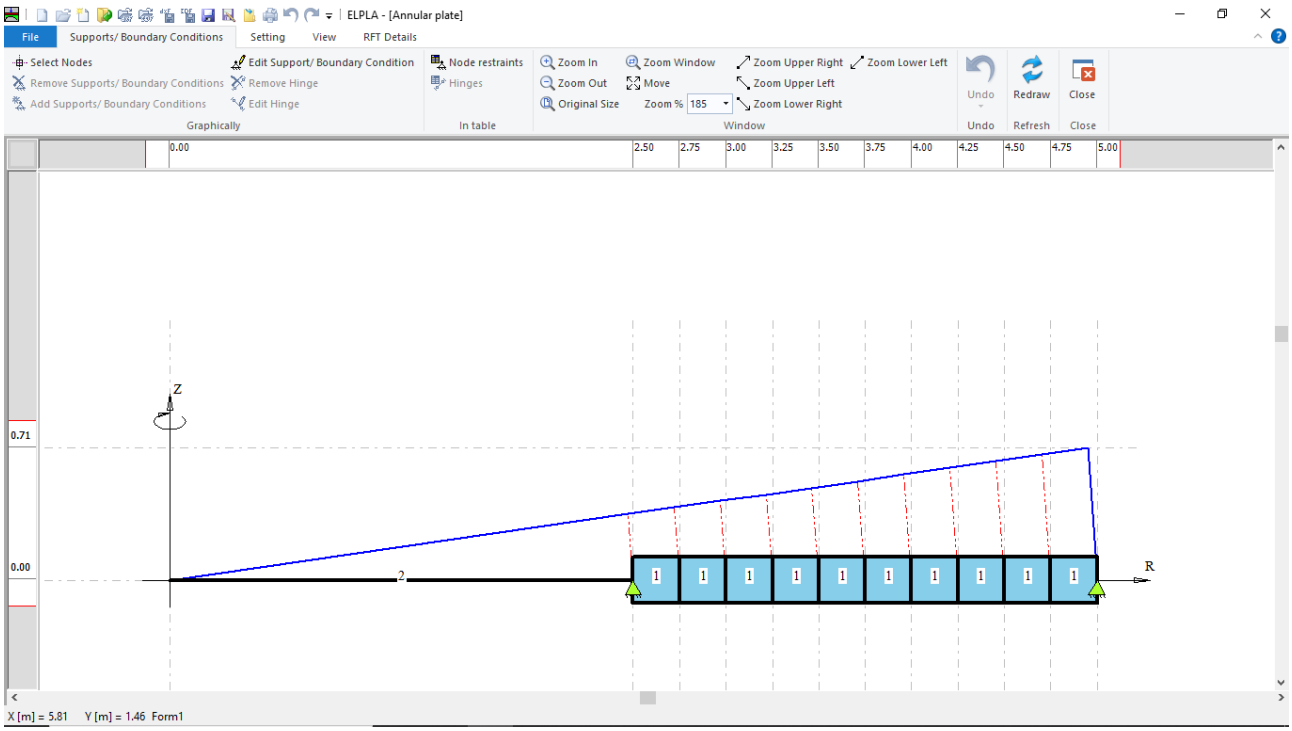

Figure 2.21 Supports on the screen

After entering supports, do the following two steps

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

4.7 Loads

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

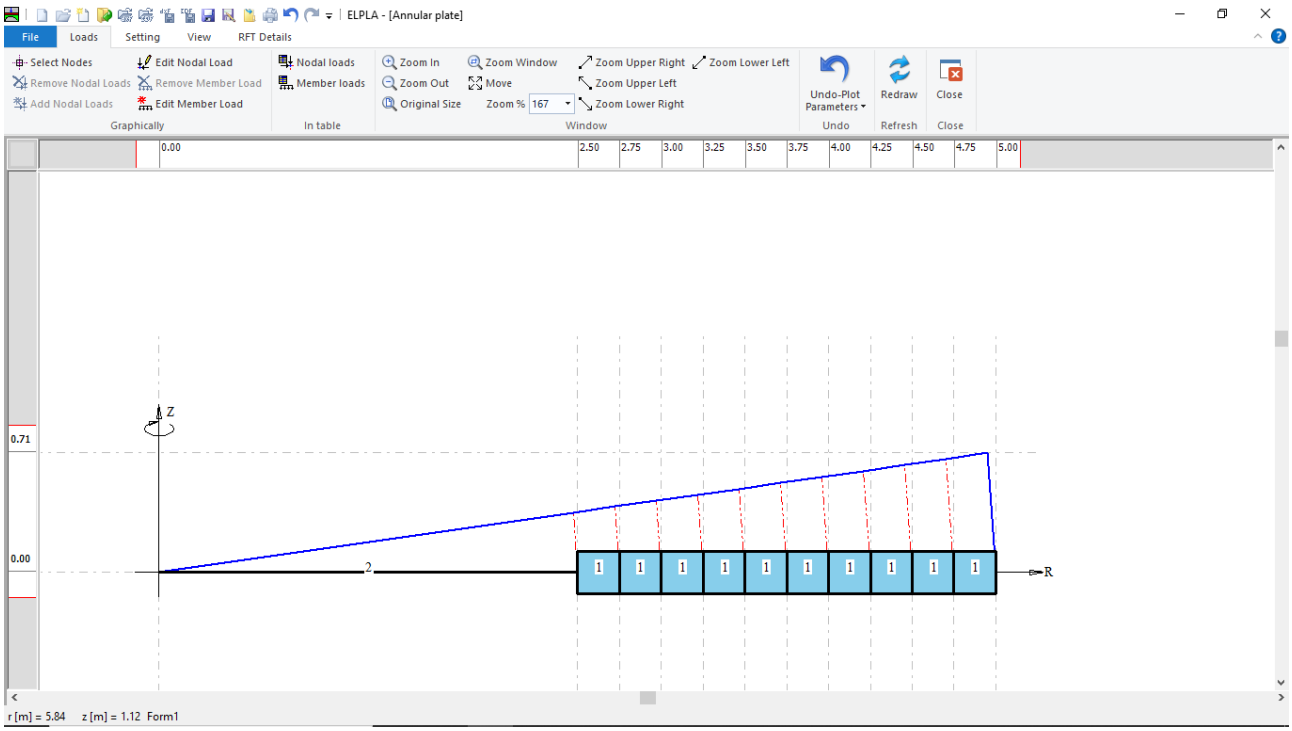

Figure 2.22 "Loads" Window

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

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

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

5 Carrying out the calculations

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

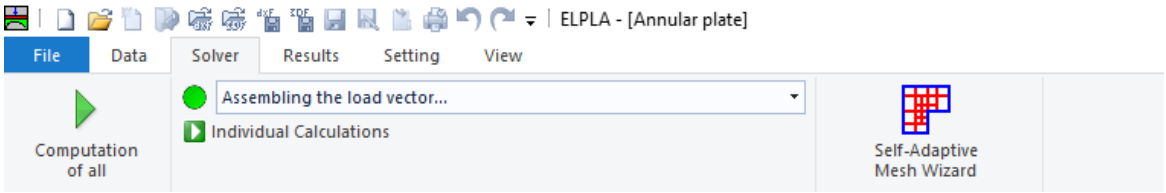

Figure 2.23 "Solver" Tab

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

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

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

Carrying out all computations

To carry out all computations in one time

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

Analysis progress

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

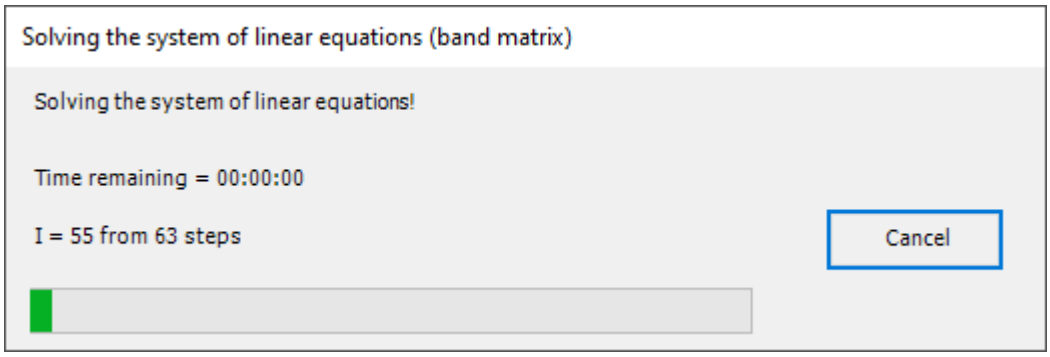

Figure 2.24 Analysis progress menu

Check of the solution

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

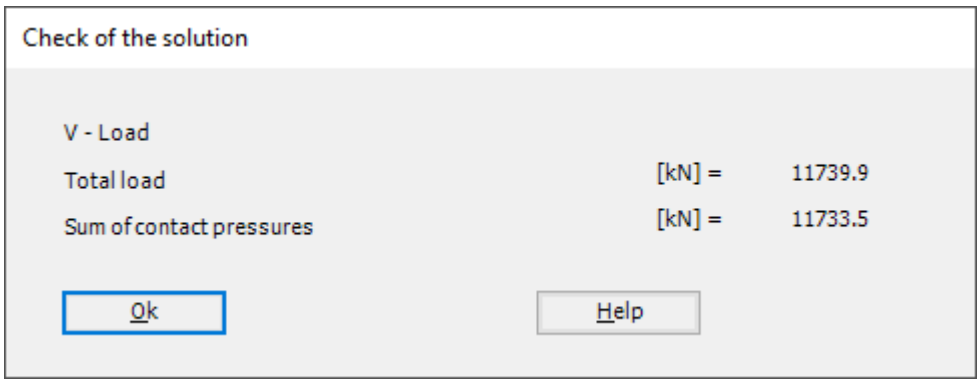

Figure 2.25 Menu "Check of the solution"

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

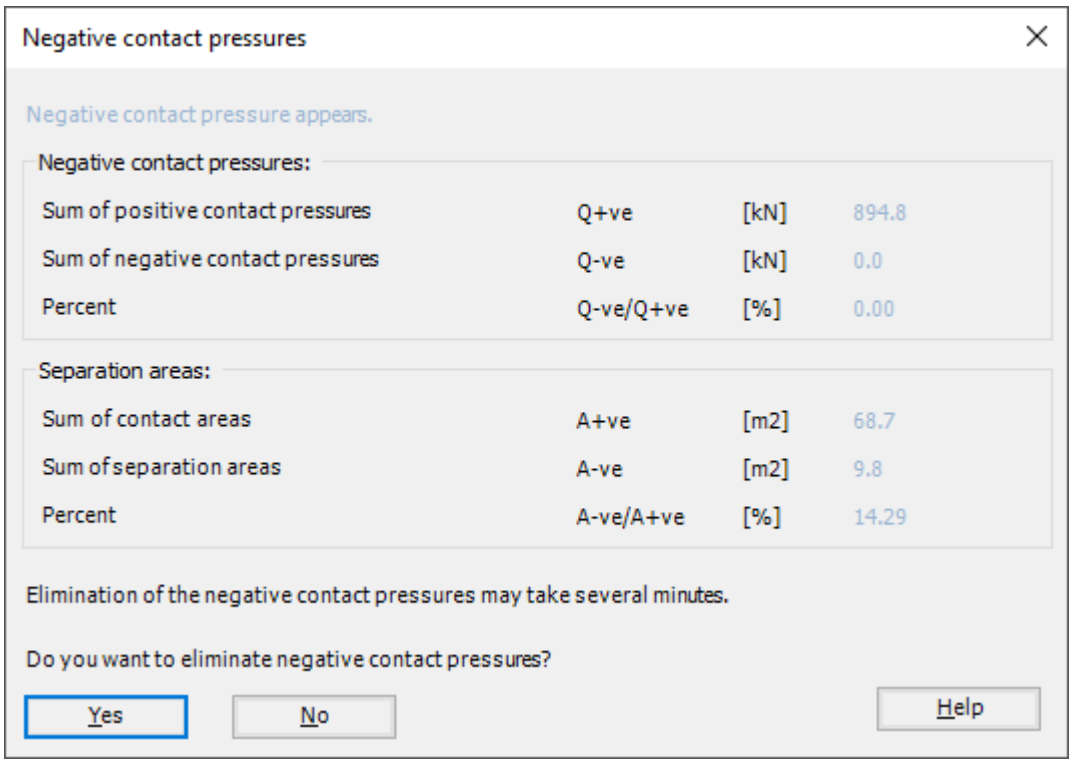

Figure 2.26 Menu "Check of the solution"

Click "OK" button to finish analyzing the problem.

6 Viewing data and results

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

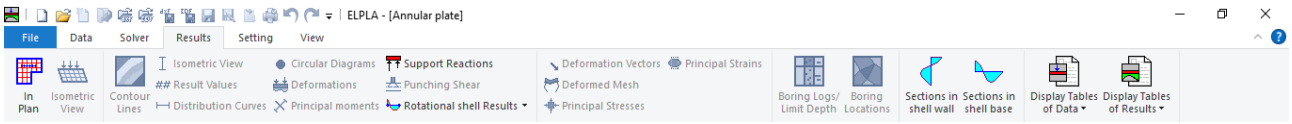

Figure 2.27 "Results" Tab

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

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

To view the meridional moments in the shell base

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

The Results are now displayed as shown in Figure 2.29.

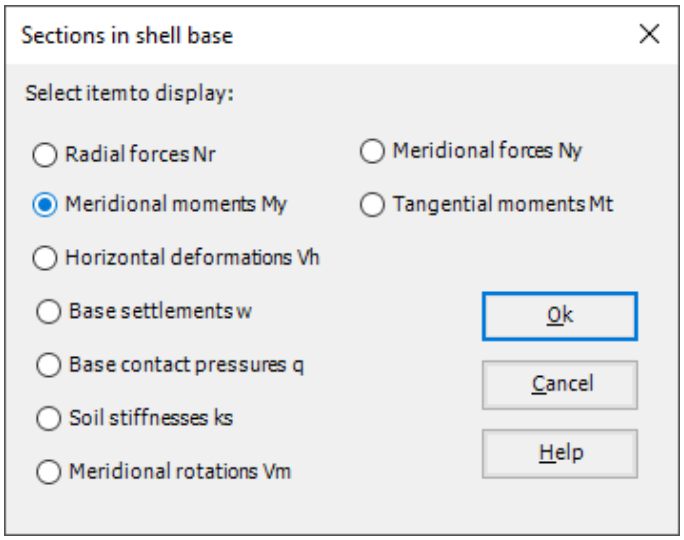

Figure 2.28 "Sections in shell base" option box

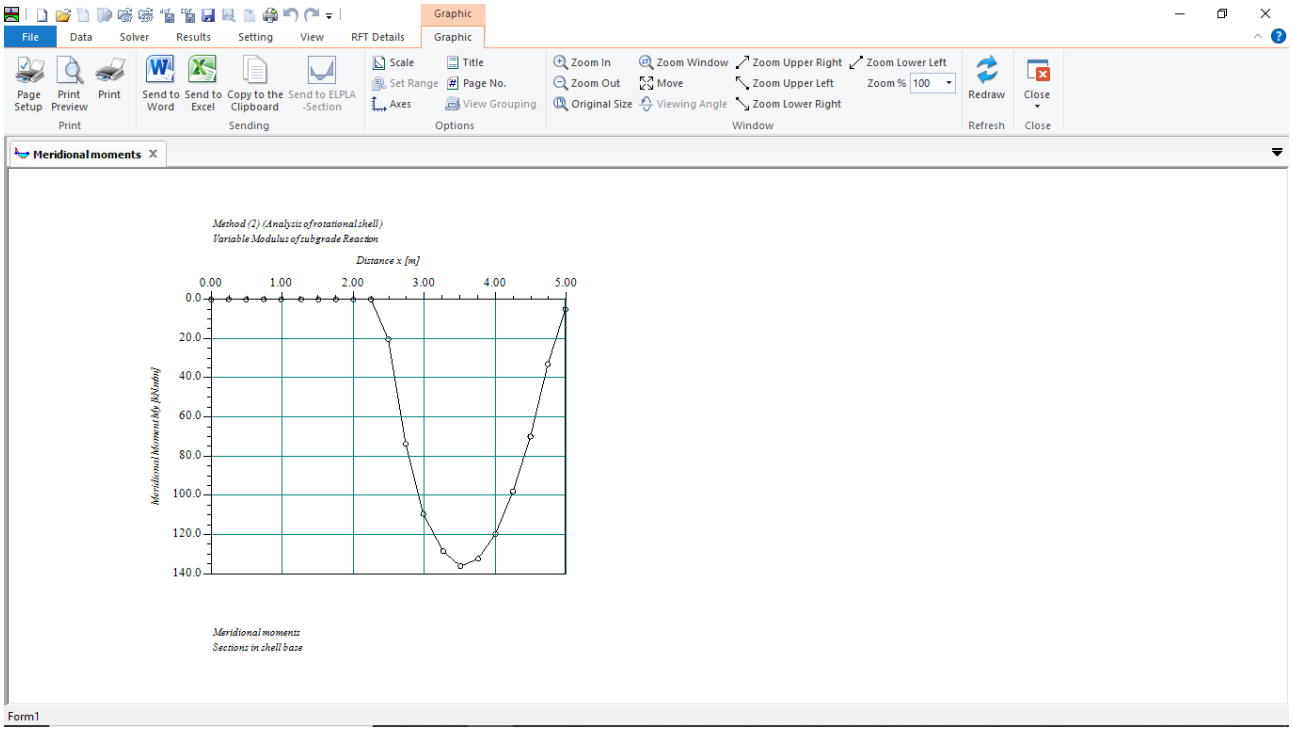

Figure 2.29 Meridional moments in shell base

To view element groups of the plate

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

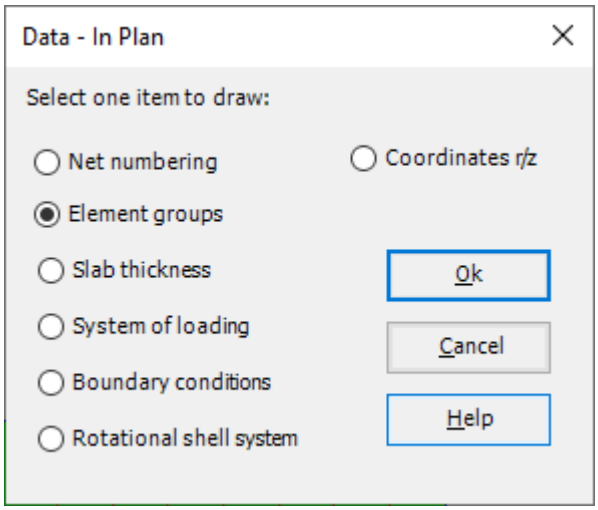

Figure 2.30 "Data – In Plan" option box

To draw the thickness of the annular plate

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

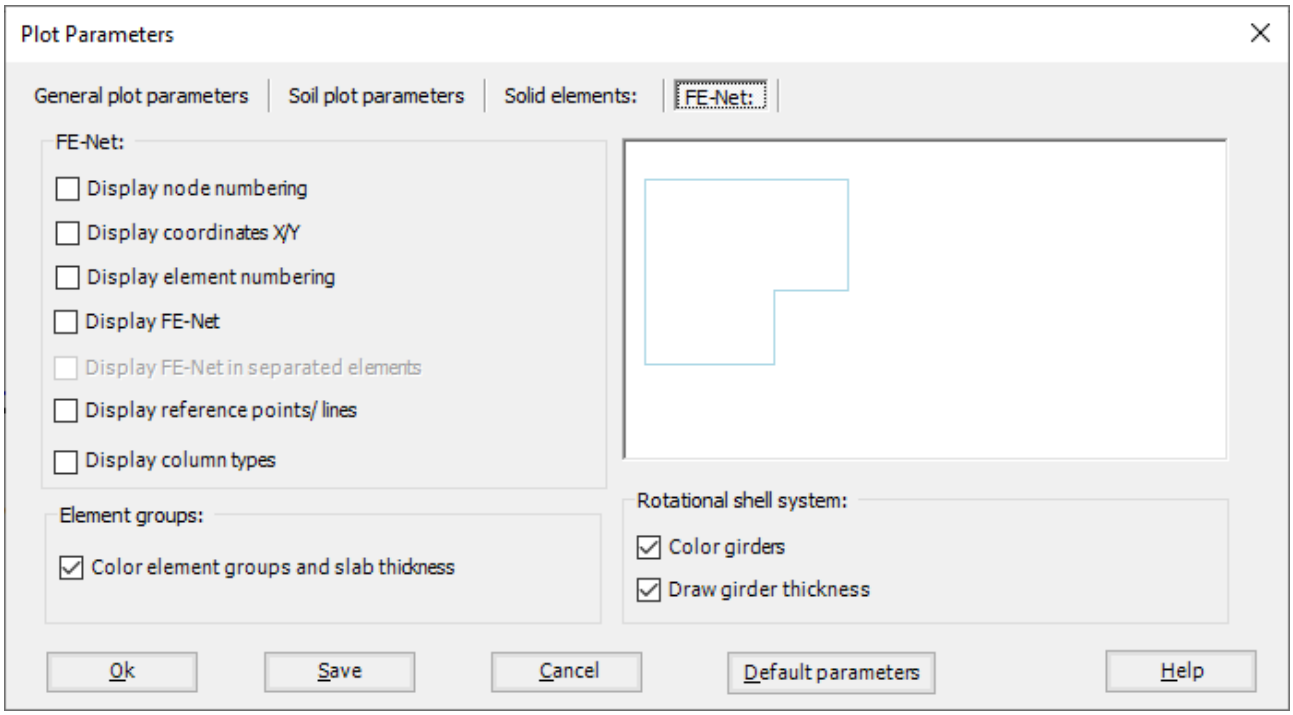

Figure 2.31 "Plot Parameters" dialog box

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

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

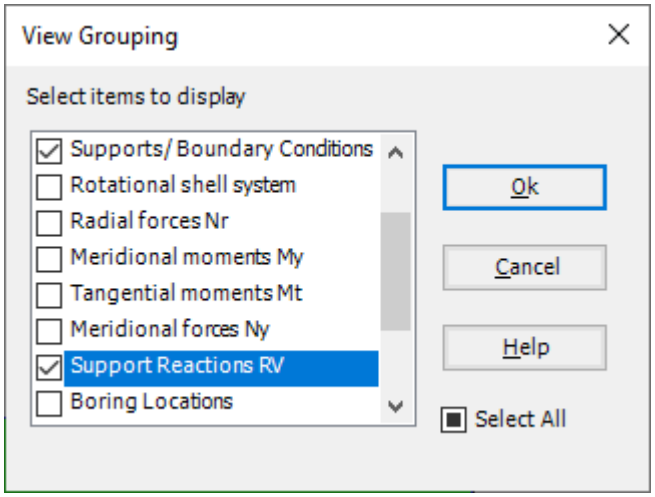

Figure 2.32 "View Grouping" check group box

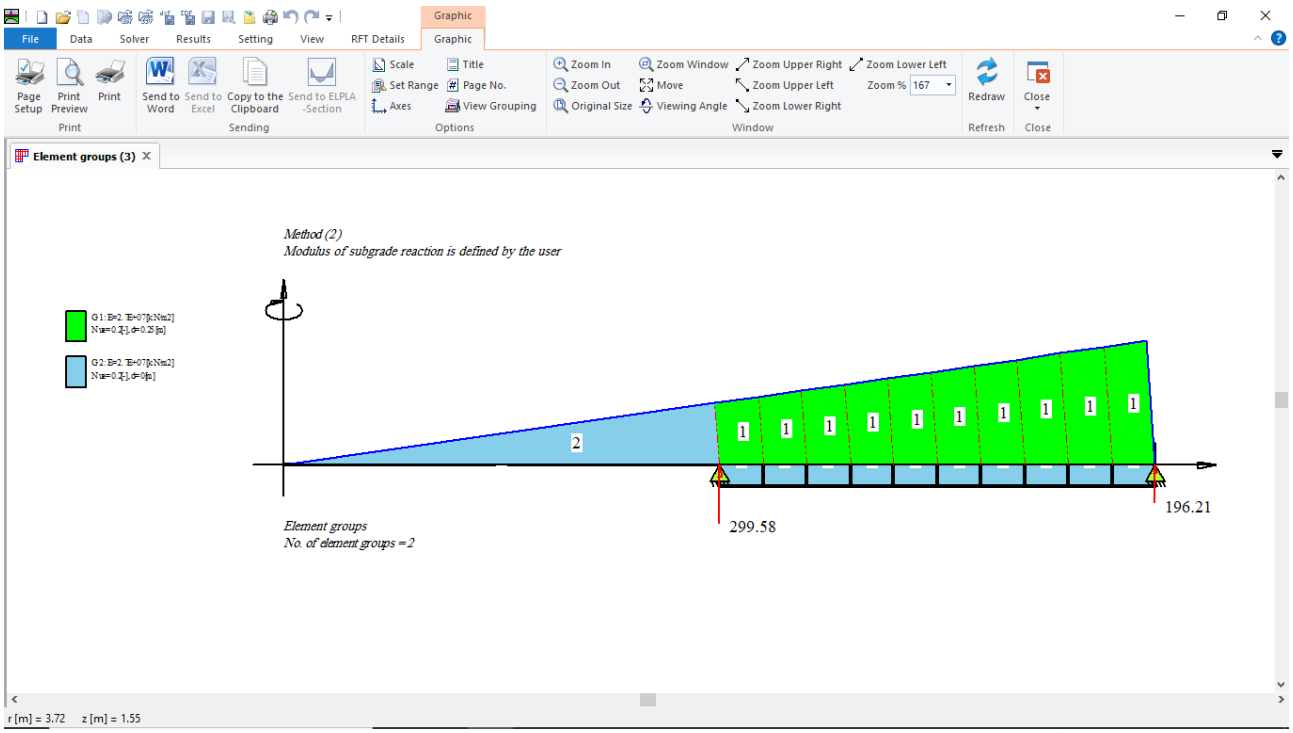

Figure 2.33 Element groups of the annular plate