

## **Example 11**

**Analysis of a chimney  
with a hyperbolic shell wall**

<b>Contents</b>	<b>Page</b>
1 Description of the problem .....	3
2 Chimney geometry and properties .....	3
3 Numerical Analysis .....	5
4 Creating the project .....	7
4.1 Calculation method .....	7
4.2 Project identification .....	10
4.3 FE-Net data .....	10
4.4 Shell properties .....	15
4.5 Supports/ Boundary Conditions .....	19
4.6 Loads .....	23
5 Carrying out the calculations .....	26
6 Viewing data and results .....	28

## Example 11

---

### 1 Description of the problem

An example of an axi-symmetrically chimney with hyperbolic shell wall under a uniform external pressure is selected to illustrate some features of *ELPLA* for analyzing shell elements.

### 2 Chimney geometry and properties

Consider a hyperbolic shell of revolution with the following geometry:

Throat radius	$R_o$	= 18	[m]
Throat height	$H_l$	= 45	[m]
Lower radius	$R_u$	= 36	[m]
Total height	$H$	= 72	[m]
Thickness of the wall	$t$	=0.24	[m]

Meridian equation of the hyperbolic shell of revolution is given by:

$$r^2(\xi) = \frac{R_u^2 - R_o^2}{H_l^2} (\xi - H_l)^2 + R_o^2$$
$$r^2(\xi) = \frac{36^2 - 18^2}{45^2} (\xi - 45)^2 + 18^2$$
$$r^2(\xi) = 0.48(\xi - 45)^2 + 324$$

where  $r$  [m] is the radius at height  $\xi$  [m].

Figure 11.1 shows the geometry the chimney with its hyperbolic shell wall with dimensions and supports, while the shell material are listed in Table 11.1.

Table 11.1 hyperboloid shell material

Modulus of Elasticity of the chimney material	$E_c$	= $3 \times 10^7$	[kN/m <sup>2</sup> ]
Poisson's ratio of the chimney material	$\nu_c$	= 0.3	[-]

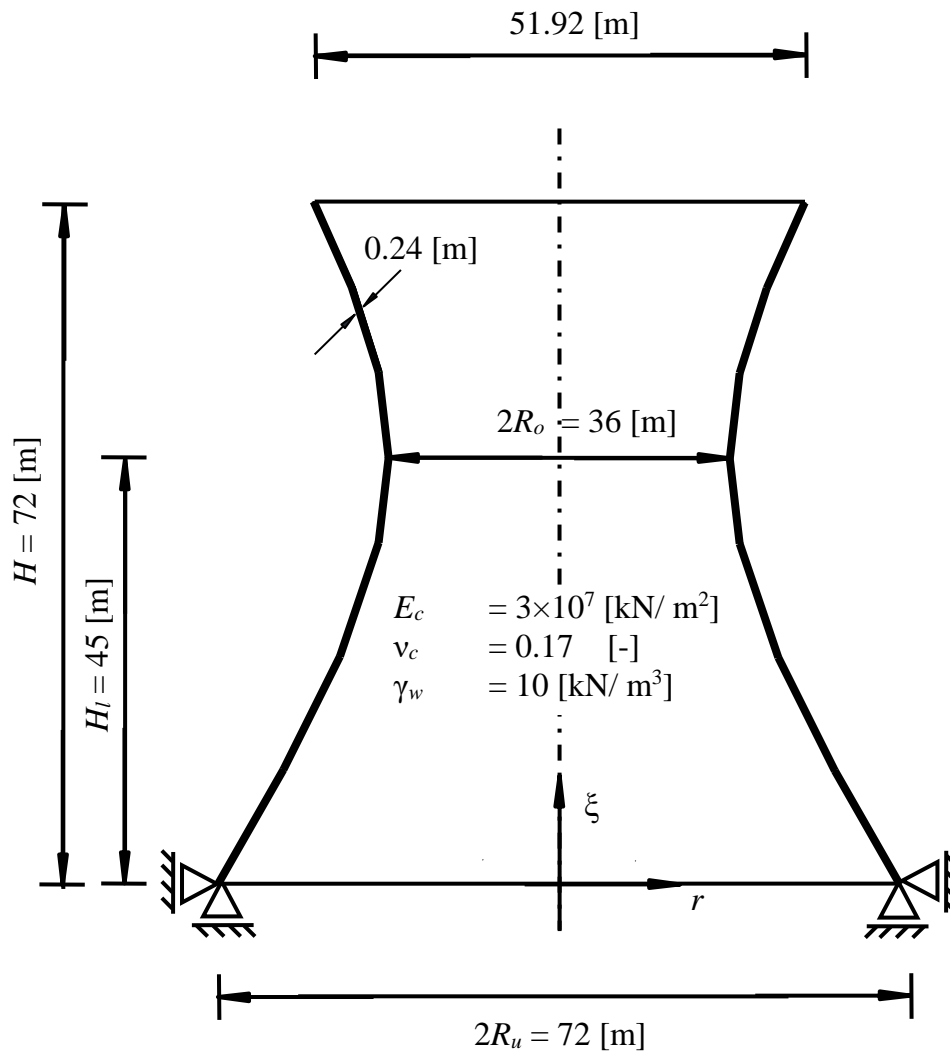


Figure 11.1 Geometry of the chimney with dimensions and supports

### 3 Numerical Analysis

In the analysis, the height of the chimney is divided into 7 main segments; each segment is divided into a number of elements. Segment dimensions and number of elements of each segment are shown in Figure 11.2.

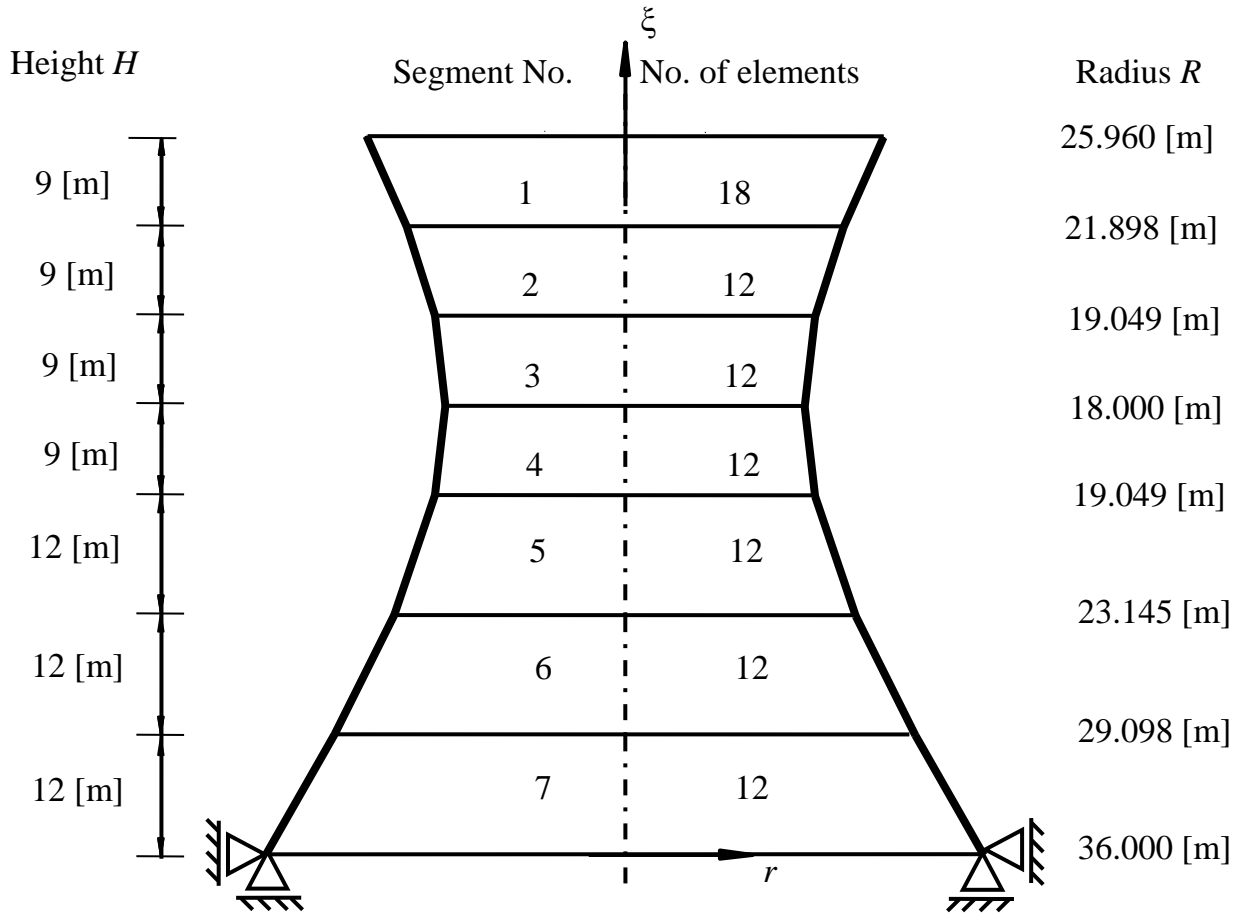


Figure 11.2 Segment dimensions and no. of elements in each segment

The chimney is exposed to a uniform external pressure of  $p_s = -10$  [kN/m<sup>2</sup>] as shown in Figure 11.3

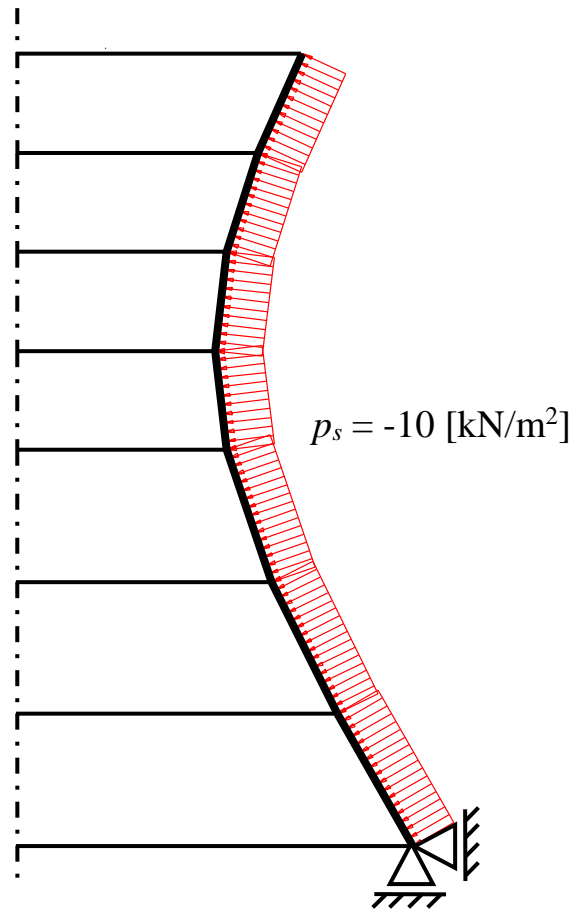


Figure 11.3 chimney with uniform external pressure of  $p_s = -10$  [kN/m<sup>2</sup>]

## 4 Creating the project

In this section, the user will learn how to create a project for analyzing a chimney with hyperbolic shell wall under a uniform external pressure. 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 11.4. 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 11.4).

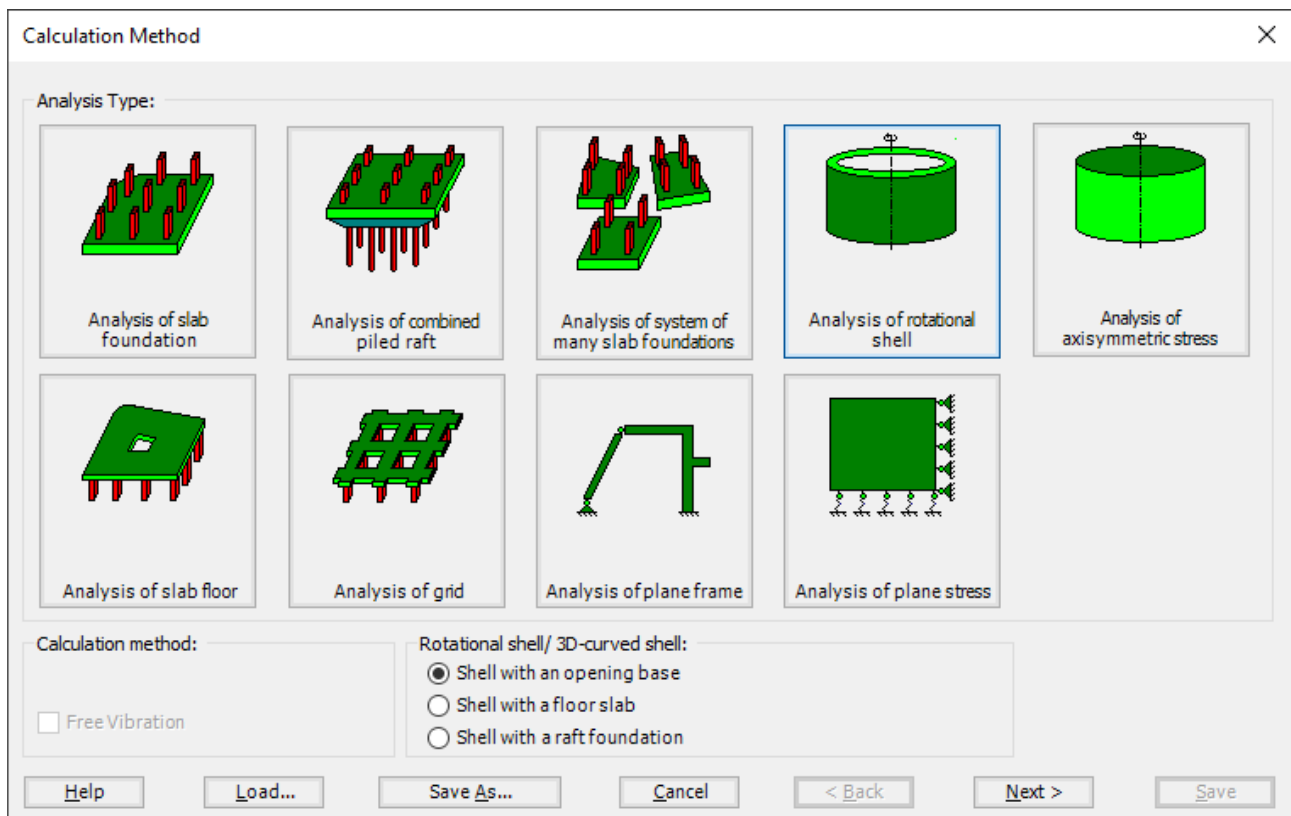


Figure 11.4 "Analysis Type" Form

In the "Analysis Type" Form in Figure 11.4, define the analysis type of the problem. As the analysis type is a hyperbolic shell 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 11.5.

The last Form in the wizard is the "Options" Form, Figure 11.5. 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.

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

Buttons: Help, Load..., Save As..., Cancel, < Back, Next >, Save

Figure 11.5 "Options" Form

After clicking "Save" button, the "Save as" dialog box appears, Figure 11.6. In this dialog box type a file name for the current project in "File name" edit box. For example, type "Hyperbolic Chimney". *ELPLA* will use automatically this file name in all reading and writing processes.



## Example 11

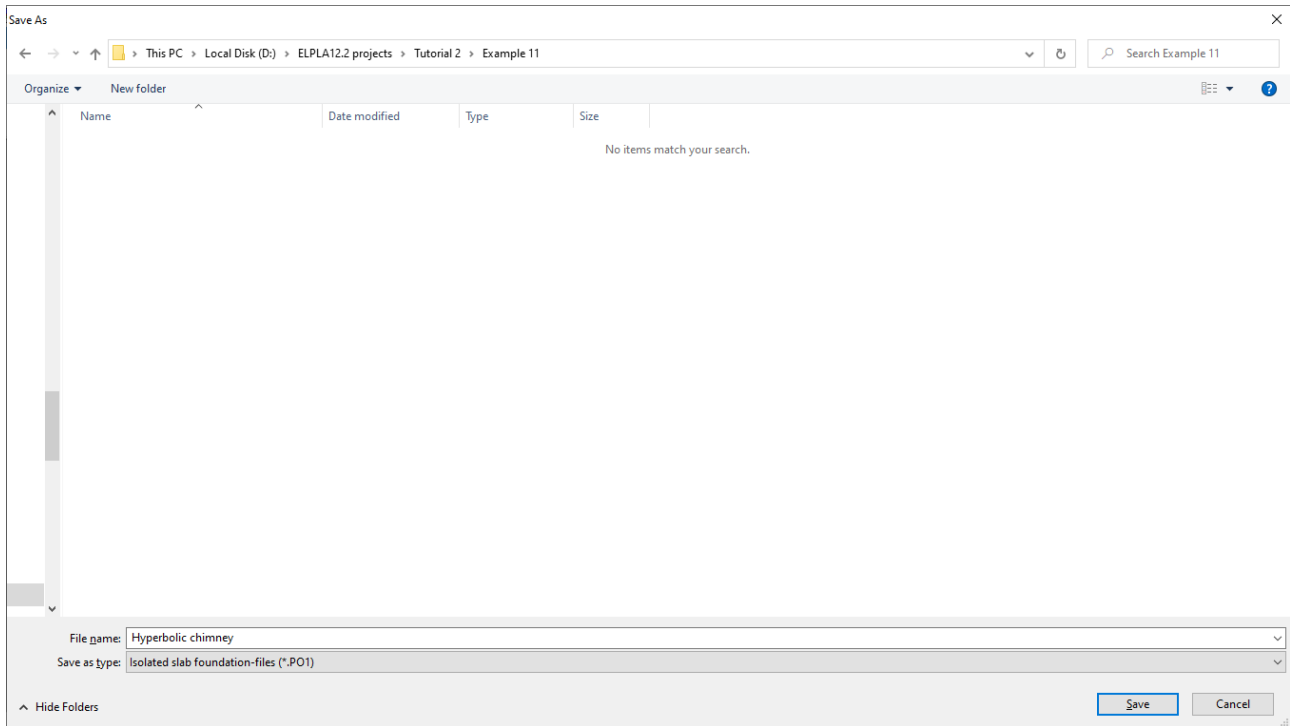


Figure 11.6 "Save as" dialog box

*ELPLA* will activate the "Data" Tab. In addition, the file name of the current project [Hyperbolic Chimney] 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 11.7 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box:  
"Analysis of a chimney with hyperbolic shell wall"
- 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

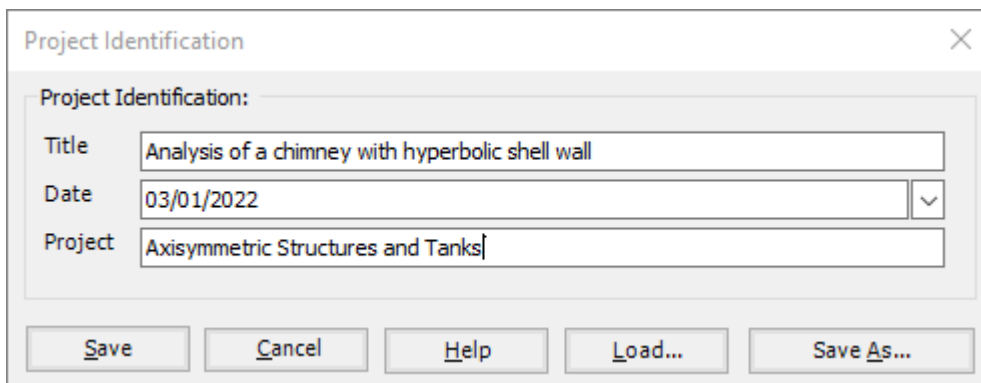


Figure 11.7 "Project Identification" dialog box

## 4.3 FE-Net data

For the given problem, the chimney has a lower radius of  $R_u = 36$  [m] and a total height of  $H = 72$  [m]. the wall height is divided into 7 main segments, each segment is divided into a number of 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 11.8. This wizard will guide you through the steps required to generate a FE-Net. As shown in Figure 11.8, 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 11

Analysis of rotational shell

Shell type:

Cylindrical shell

Conical shell

Spherical shell

Hyperbolic shell

Elliptical shell

Cycloidal shell

Parabolic shell

Irregular shell

Hyperbolic shell:

Throat radius	Ro	[m]	18.000
Total height	Hw	[m]	72.000
Lower radius	Ru	[m]	36.000
Throat height	HL	[m]	45.000

Number of segments:

Number of segments	Ns	[-]	7
--------------------	----	-----	---

Help Cancel < Back Next > Finish

Figure 11.8 "Analysis of rotational shell" wizard with "Shell type" Form

To generate the FE-Net

- In the "Shell type" options choose "Hyperbolic shell" button
- Type 18 in the "Throat radius" edit box
- Type 72 in the "Total height" edit box
- Type 36 in the "Lower radius" edit box
- Type 45 in the "Throat height" edit box
- Type 7 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 "Hyperbolic shell" Form appears Figure 11.9, *ELPLA* divides the height of the chimney wall into 7 equal segments, The user can edit the data of the segments individually by using "Modify" button, or all of them using "In Table" button.

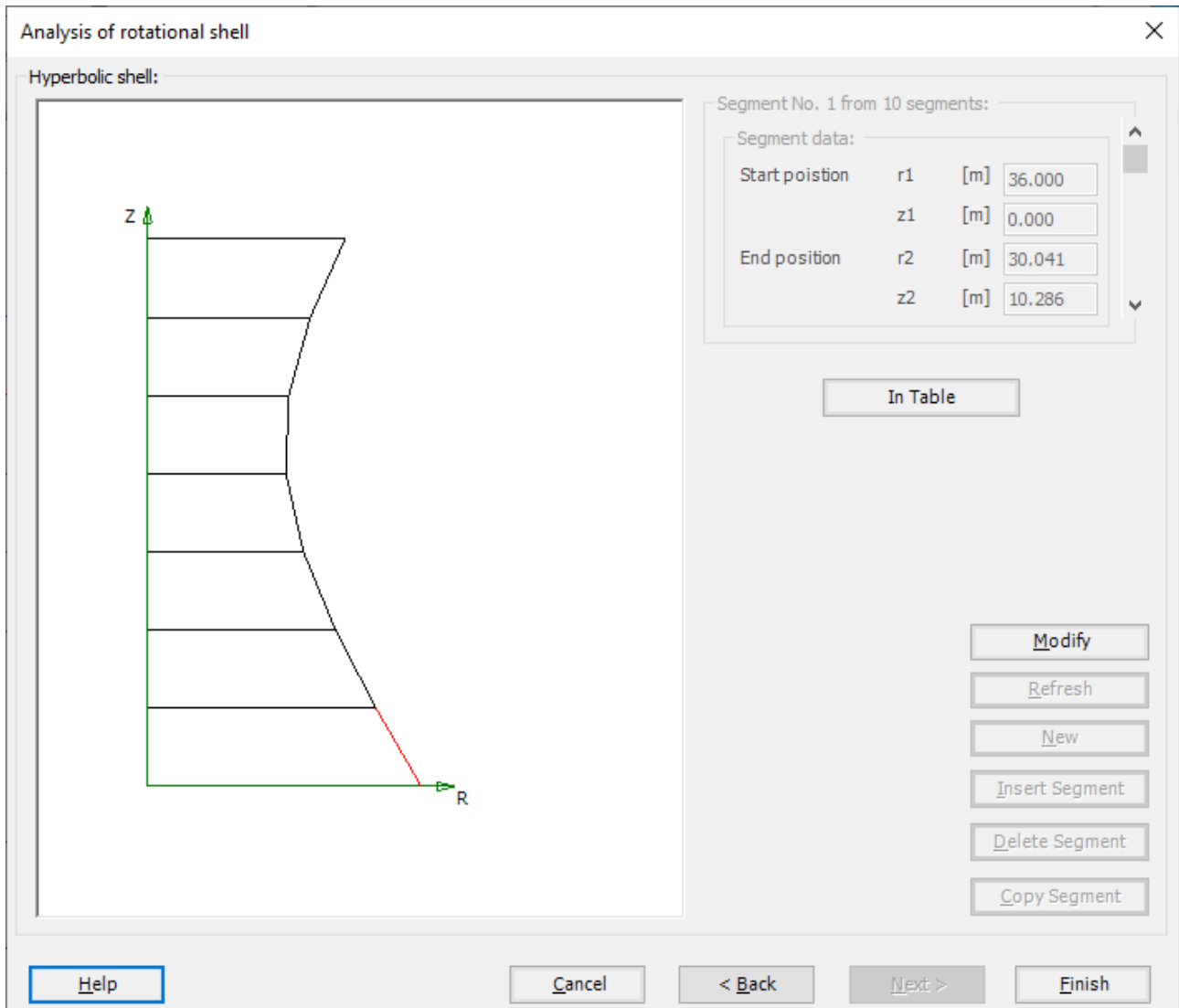
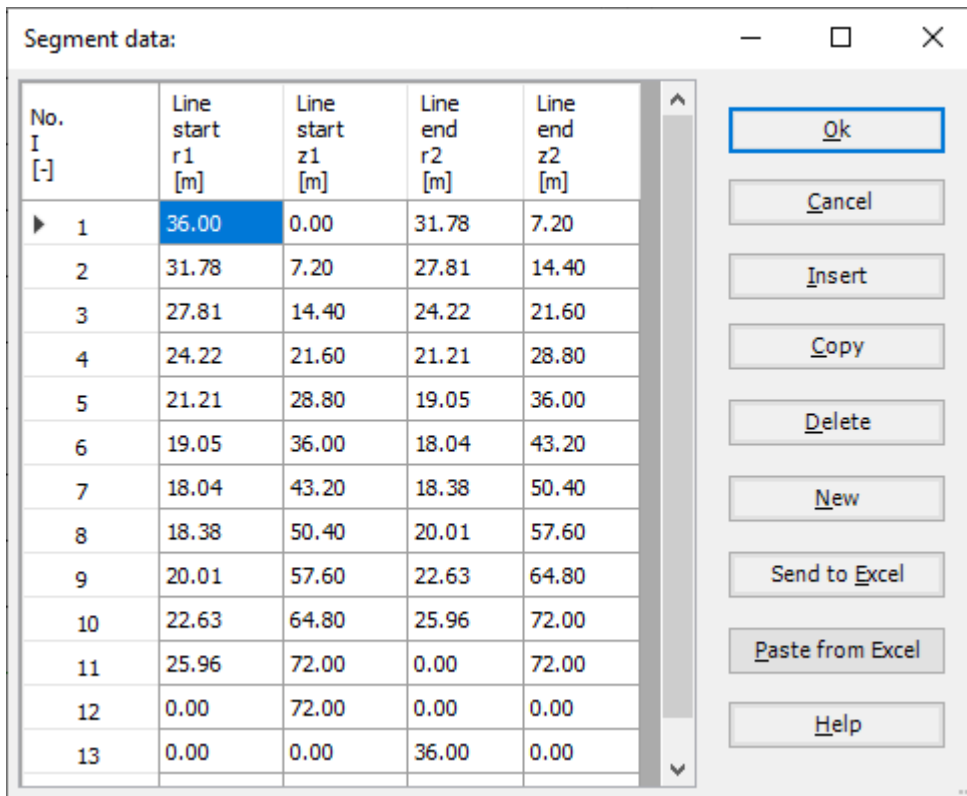


Figure 11.9 "Hyperbolic shell" Form

## Example 11



No. I [-]	Line start r1 [m]	Line start z1 [m]	Line end r2 [m]	Line end z2 [m]
▶ 1	36.00	0.00	31.78	7.20
2	31.78	7.20	27.81	14.40
3	27.81	14.40	24.22	21.60
4	24.22	21.60	21.21	28.80
5	21.21	28.80	19.05	36.00
6	19.05	36.00	18.04	43.20
7	18.04	43.20	18.38	50.40
8	18.38	50.40	20.01	57.60
9	20.01	57.60	22.63	64.80
10	22.63	64.80	25.96	72.00
11	25.96	72.00	0.00	72.00
12	0.00	72.00	0.00	0.00
13	0.00	0.00	36.00	0.00

Figure 11.10 "Segment data" Table

After modifying the coordinates of the segments, click "Finish" button, the FE-Net of the chimney appears in Figure 11.11

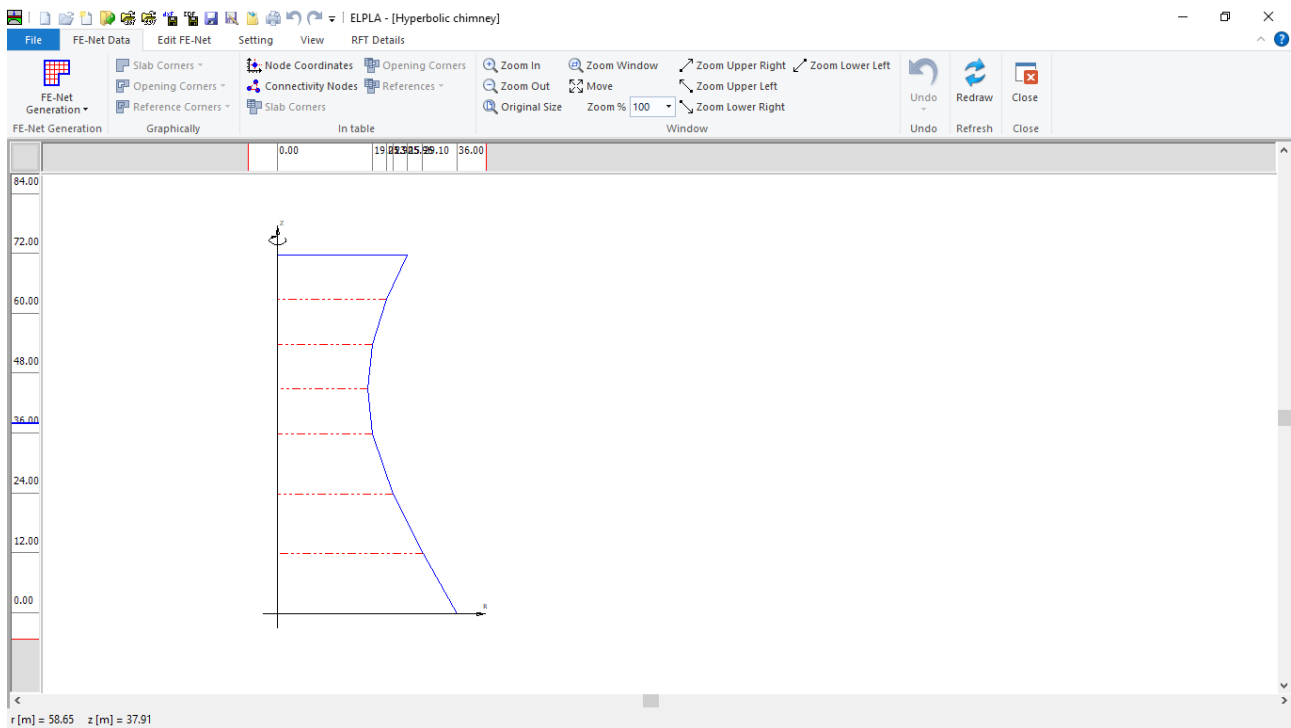


Figure 11.11 Generated FE-Net

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

- Choose "Save" command from "File" menu in Figure 11.11 to save the data of the FE-Net
- Choose "Close" command from "File" menu in Figure 11.11 to close the "FE-Net" window and return to *ELPLA* main window

## Example 11

### 4.4 Shell properties

To define the chimney properties, choose "Shell Properties" command from "Data" Tab. The following window in Figure 11.12 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 chimney, and element size.

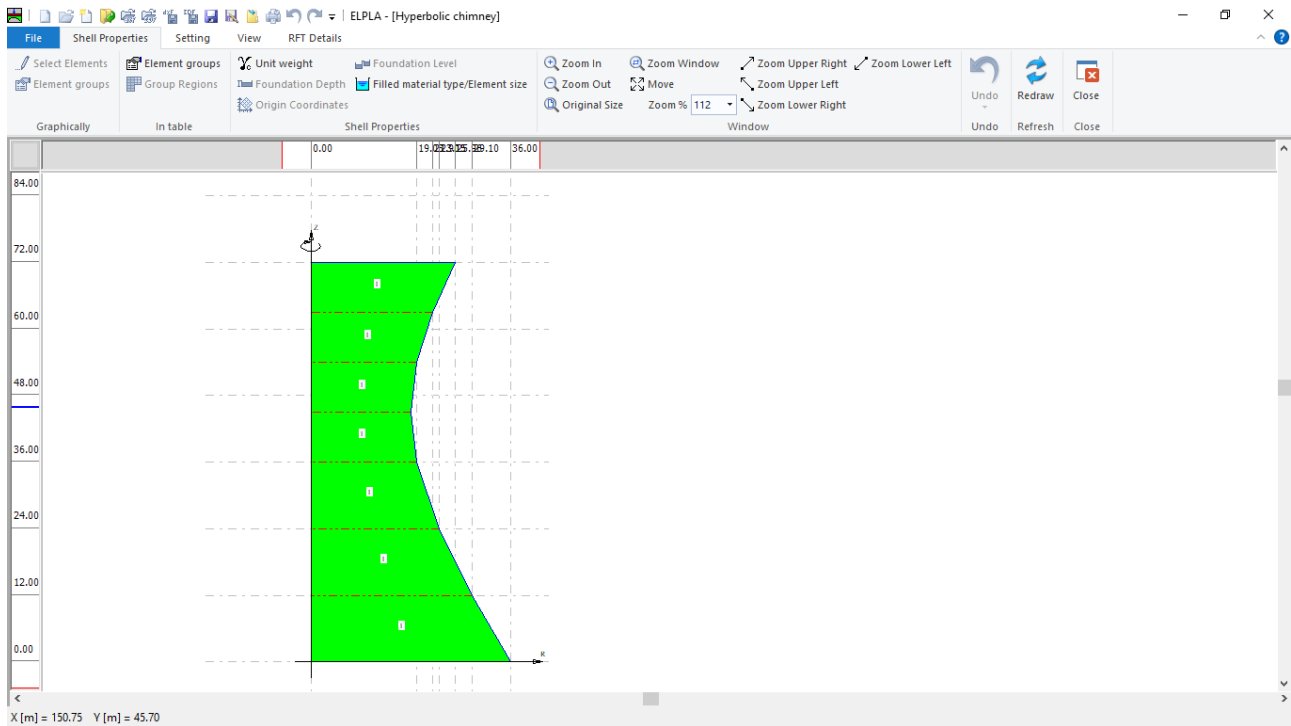


Figure 11.12 "Shell Properties" Window

Choose "Element groups" command from "In table" menu. The following list box in Figure 11.13 appears. In this list box, enter E-Modulus, *Poisson's* ratio and slab thickness. Then click "OK" button.

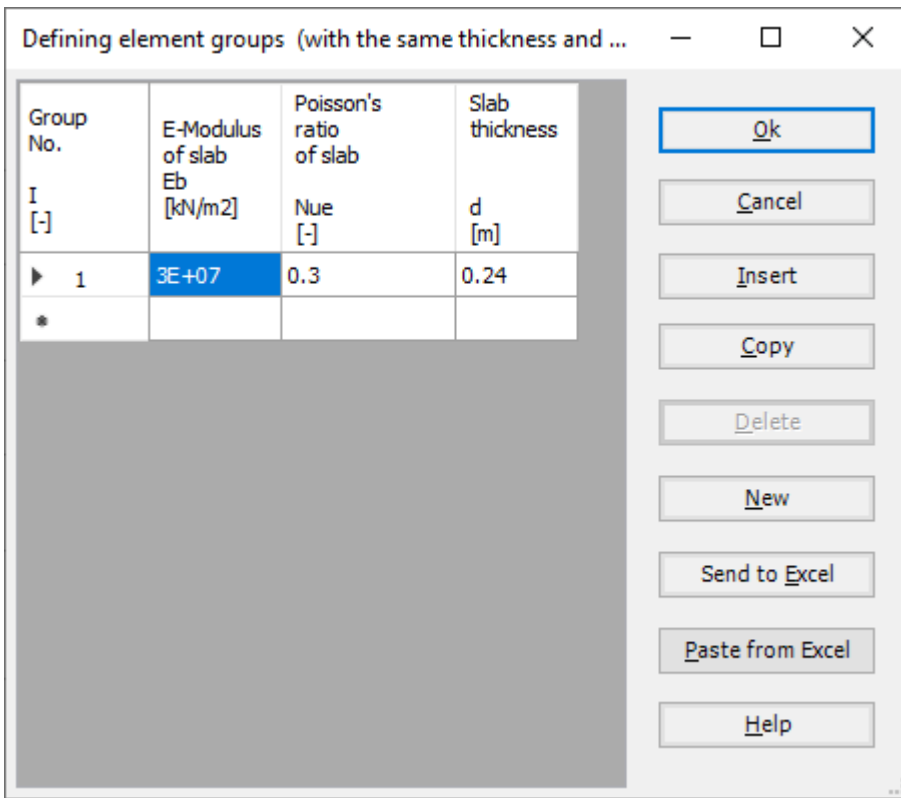


Figure 11.13 "Defining element groups" list box

To enter the unit weight of the chimney, choose "Unit weight" command from "Shell Properties" menu in Figure 11.12. The following dialog box in Figure 11.14 with a default unit weight of 25 [kN/m<sup>3</sup>] appears, type 0 in the "Unit weight" edit box to neglect the self-weight of the chimney, then click "OK" button.

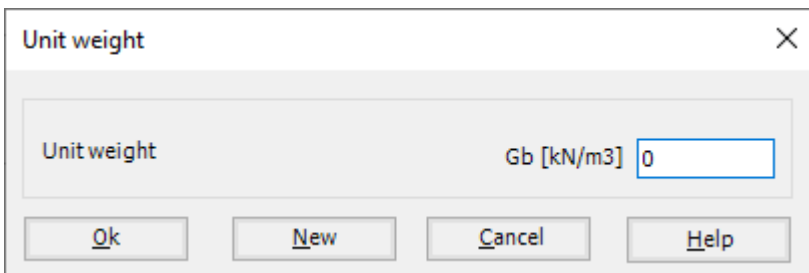


Figure 11.14 "Unit weight" dialog box



## Example 11

To define the element size of the chimney, choose "Filled material type/Element size" command from "Shell Properties" menu in Figure 11.12. The following form in Figure 11.15 appears, then:

- Select "Empty container" option
- uncheck the "Constant element sizes in z-direction" check box
- Click "Element size in each segment" button, the following list box in Figure 11.16 appears
- Define the size of the element in each segment as in the following list box in Figure 11.16, to keep the same number of elements in each segment
- Click "OK" button

Filled material type/Element size

Filled material type:

Empty container  
 Liquid container  
 Granular material container

Liquid Properties:

Height of the liquid Hl [m] 0.00  
Unit weight of the liquid Yw [kN/m3] 9.81

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  $\phi$  [°] 25  
Angle of the wall friction  $\delta$  [°] 20

Element size:

Constant element sizes in z-direction

Element size in each shell segment... Dl [m] 0.2000

Ok Cancel Help

Figure 11.15 "Filled material type/Element size" Form

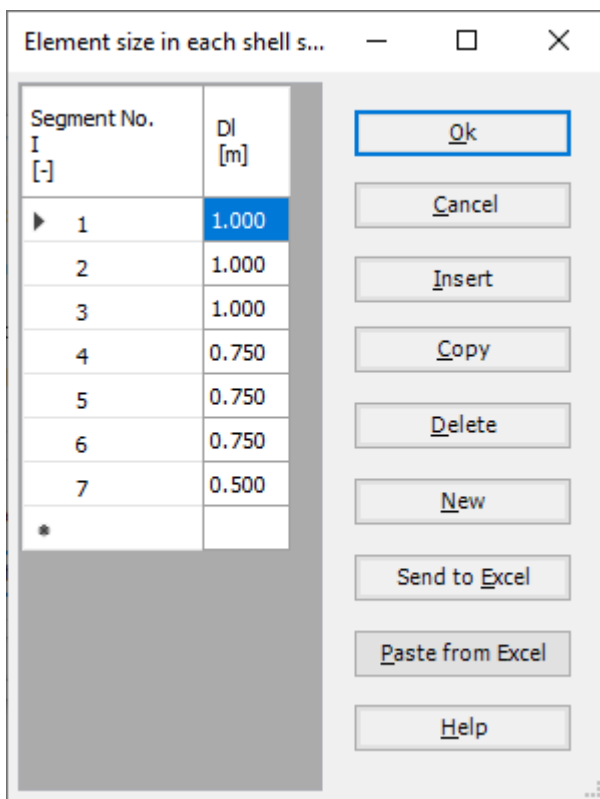


Figure 11.16 "Element size in each segment" list box

After defining the chimney properties, do the following two steps:

- Choose "Save" command from "File" menu in Figure 11.12 to save the shell properties
- Choose "Close" command from "File" menu in Figure 11.12 to close the "Shell properties" window and return to *ELPLA* main window

## Example 11

### 4.5 Supports/ Boundary Conditions

To define the hinged support, choose "Supports/ Boundary Conditions" command from "Data" Tab. The following window in Figure 11.17 appears.

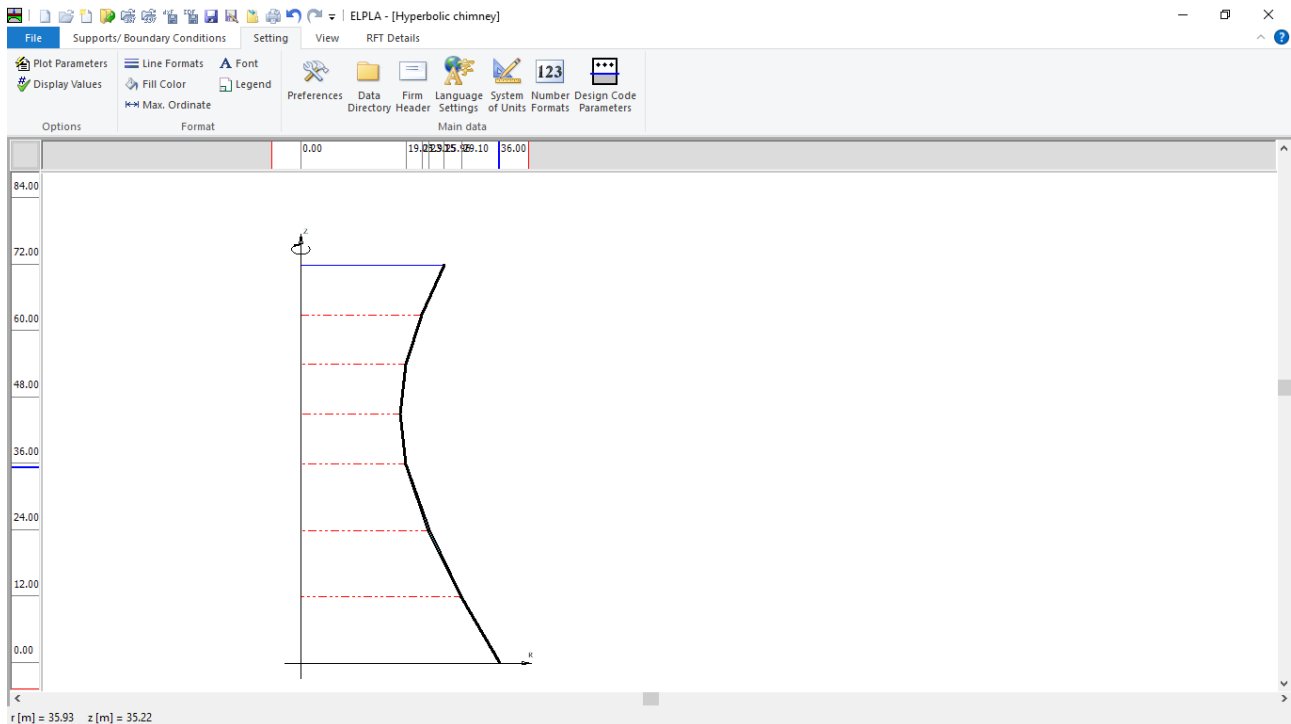


Figure 11.17 "Supports/ Boundary Conditions" Window

To define the support on the net:

- Choose "Select Nodes" command from "Graphically" menu in Figure 11.17. 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 11.18
- After selecting the node, choose "Add Supports/ Boundary Conditions" command from "Graphically" menu (Figure 11.17). The "Supports/ Boundary Conditions" dialog box in Figure 11.19 appears.

## Analyzing Axisymmetric Structures and Tanks by *ELPLA*

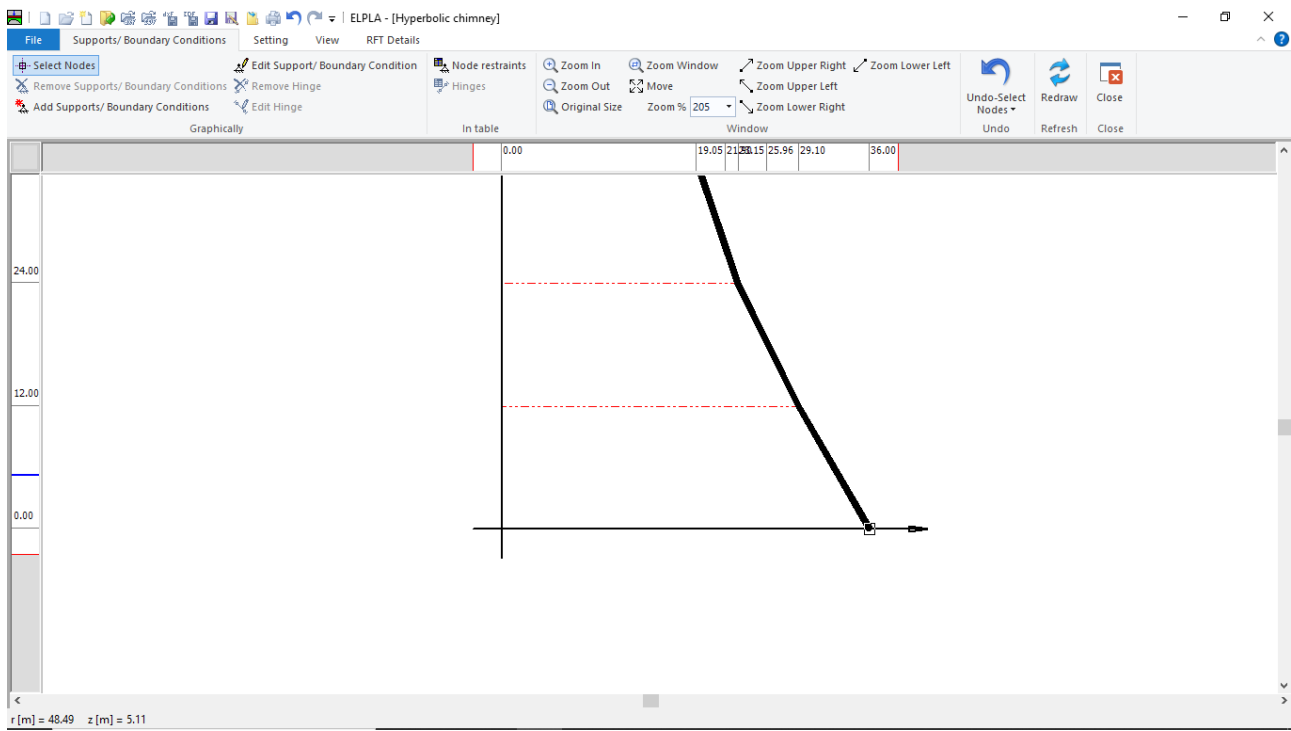


Figure 11.18 Selection of the node that has a hinged support

## Example 11

---

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
- Click "OK" button

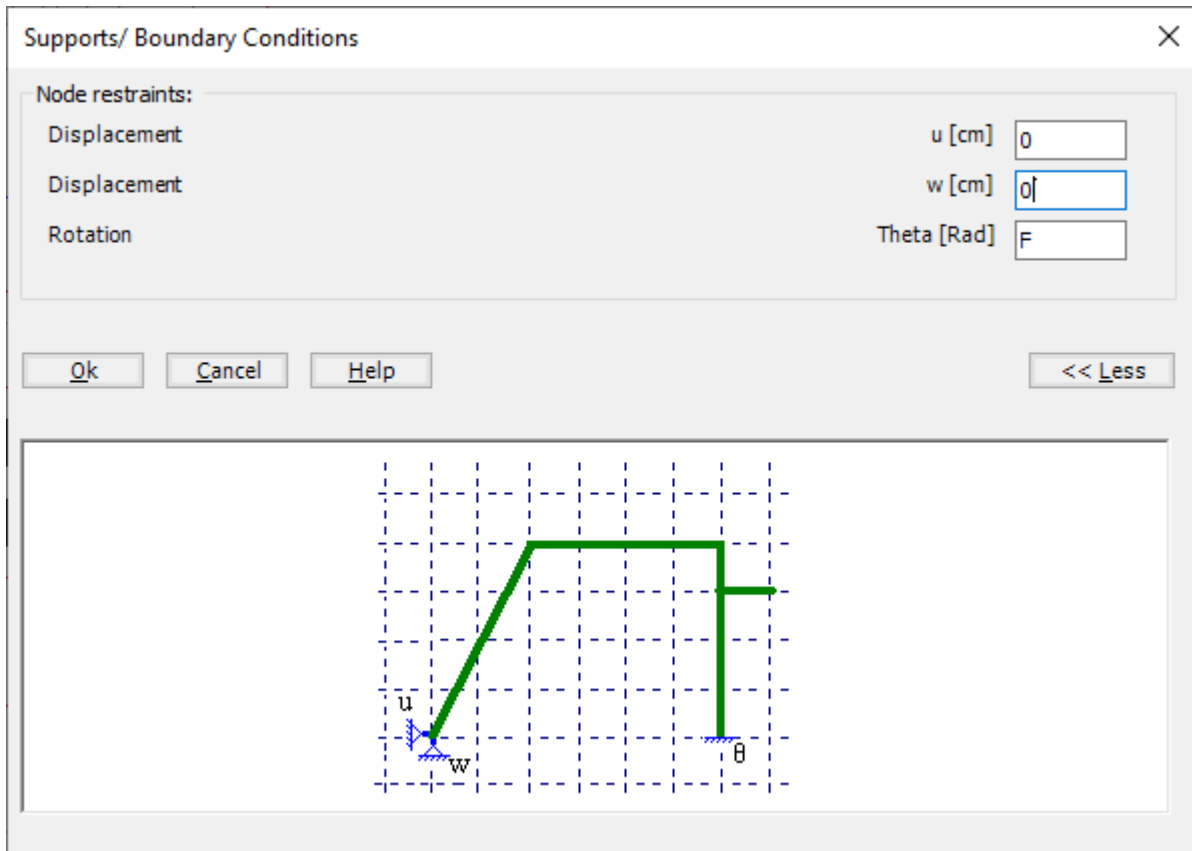


Figure 11.19 "Supports/ Boundary Conditions" dialog box

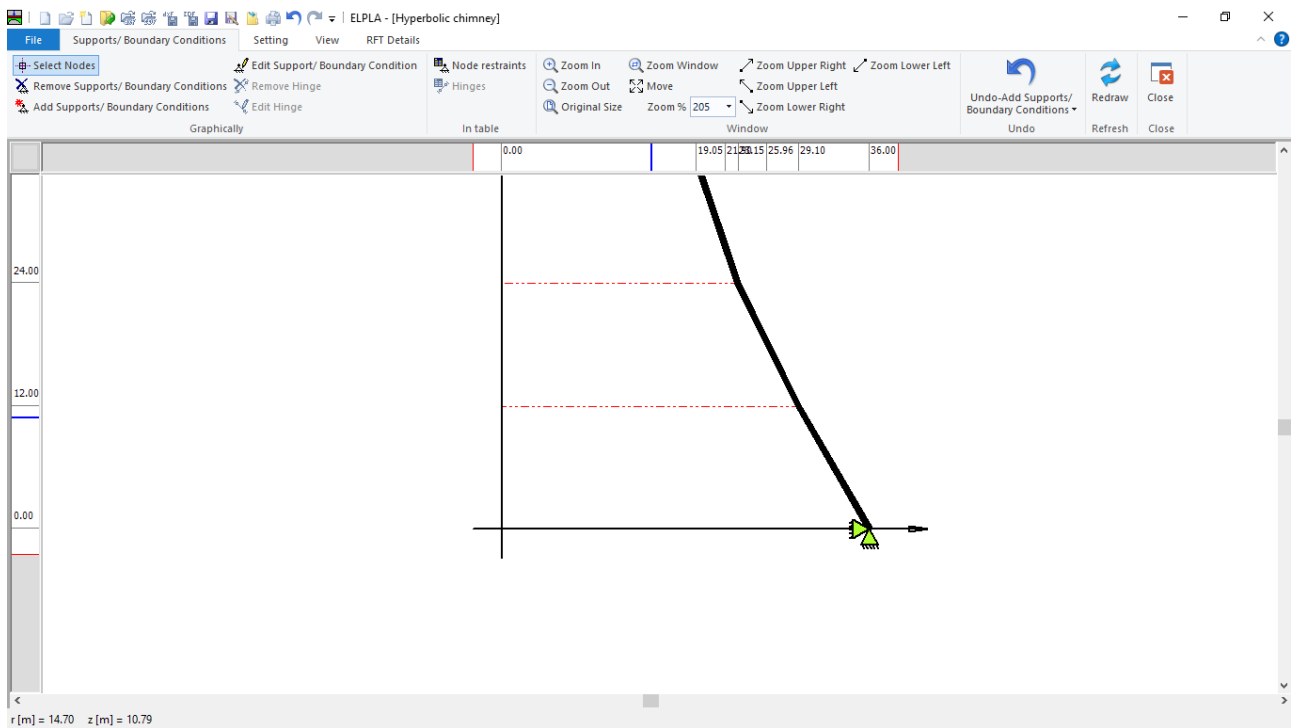


Figure 11.20 Supports on the screen

After defining the supports, do the following two steps

- Choose "Save " command from "File" menu in Figure 11.17 to save the data of supports
- Choose "Close" command from "File" menu in Figure 11.17 to close the "Supports/ Boundary conditions" window and return to the main window.

## Example 11

### 4.6 Loads

To define the loads, choose "Loads" command from "Data" Tab. The following window in Figure 11.21 appears.

In *ELPLA*, entering loads may be carried out either numerically (in a table) or graphically using the commands of "Loads" Tab in Figure 11.21. In this example, the hyperbolic chimney is exposed to a uniform external pressure of  $p_s = -10$  [kN/m<sup>2</sup>].

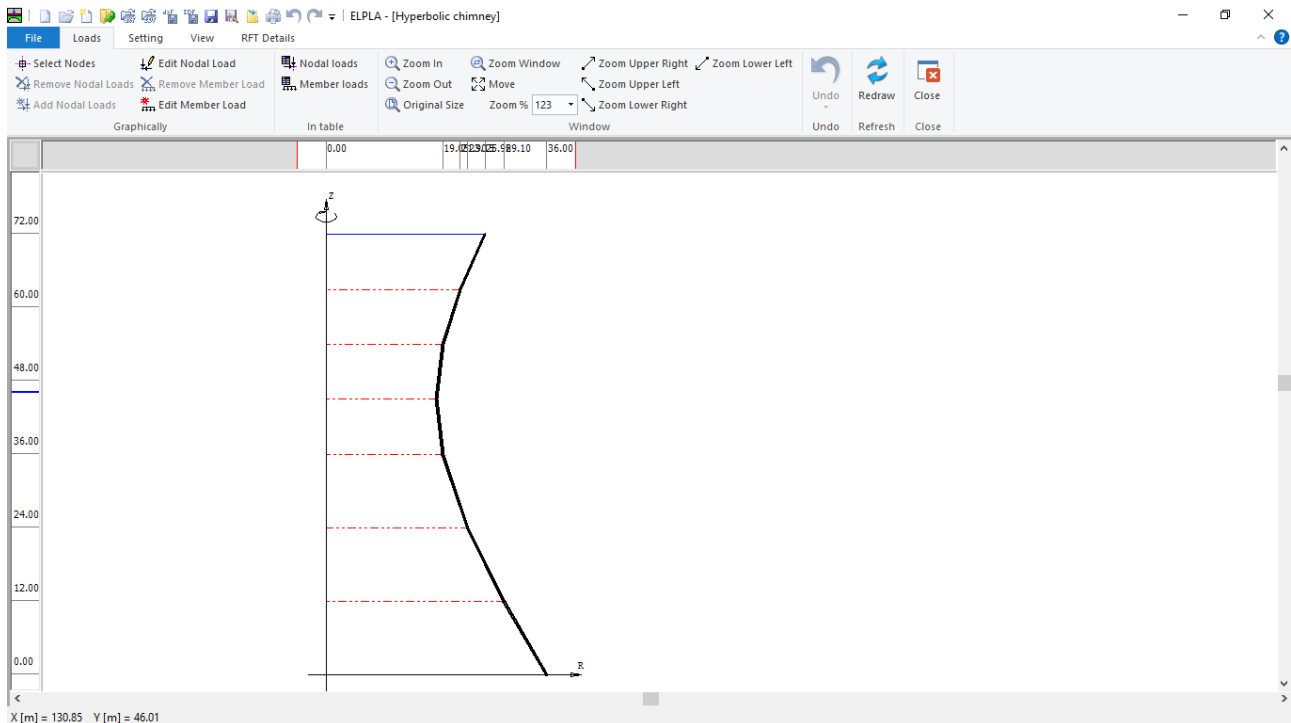


Figure 11.21 "Loads" Window

To define the uniform distributed load:

- choose "Edit Member Load" command from "Graphically" menu in Figure 11.21. When "Edit Member Load" command is chosen, the cursor will change from an arrow to a cross hair
- Double click on the element which is exposed to the uniform distributed load
- "Edit Member Load" dialog box in Figure 11.22 appears, type "-10" in "Member loads" edit box
- Click "OK" button

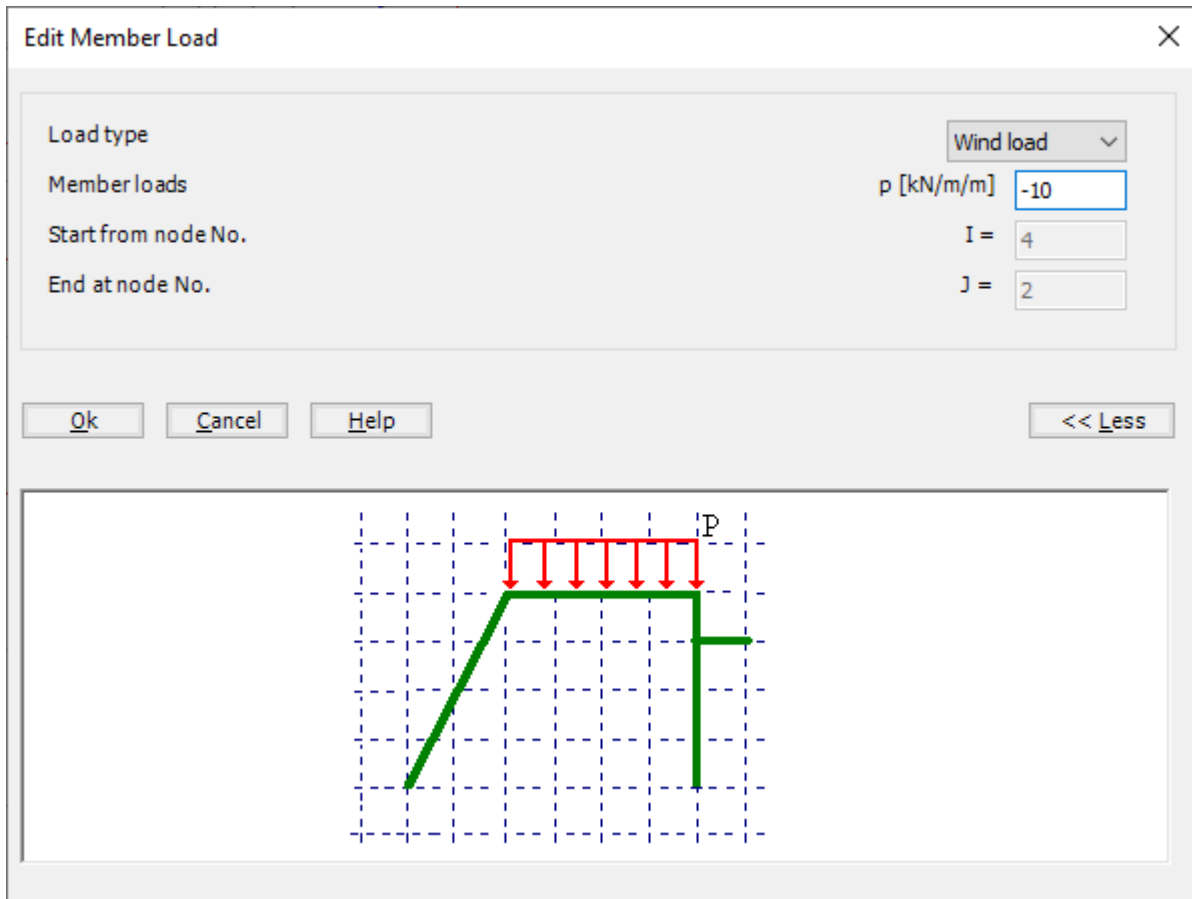


Figure 11.22 "Edit Member Load" dialog box



## Example 11

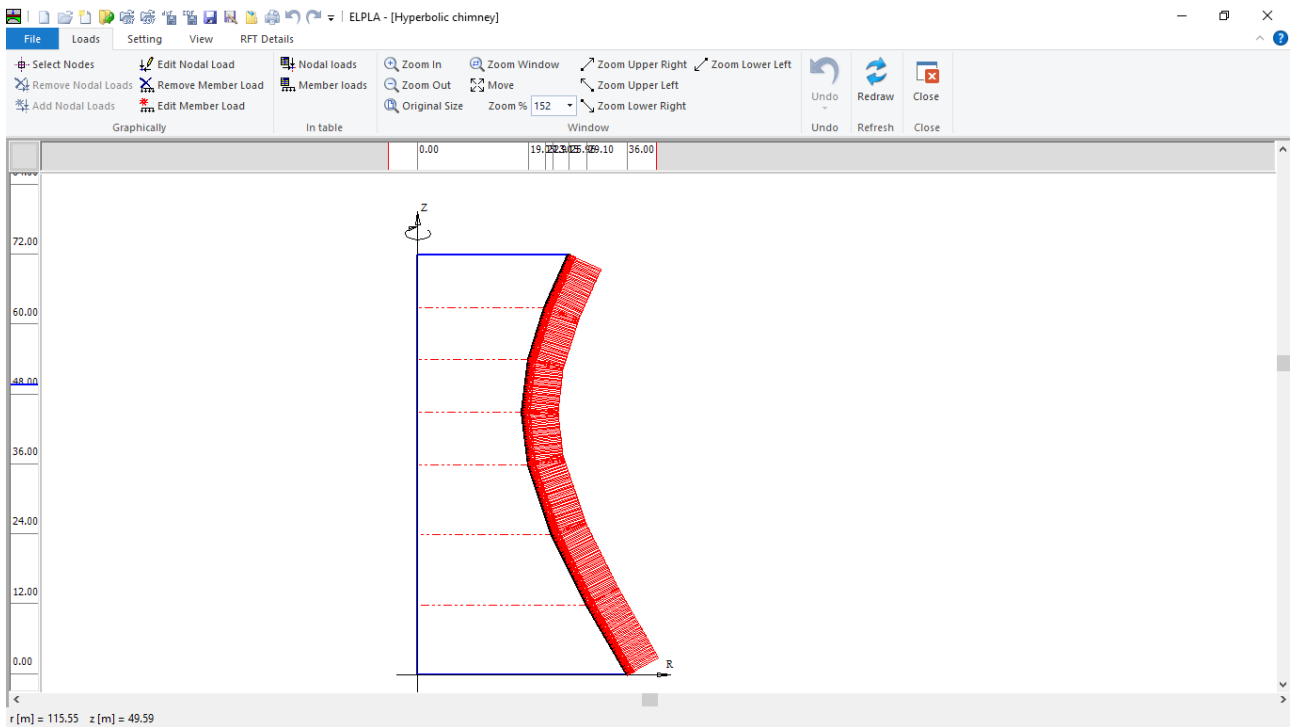


Figure 11.23 "Loads" window after defining the uniform distributed load

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

- Choose "Save" command from "File" menu in Figure 11.21 to save the load data
- Choose "Close" command from "File" menu in Figure 11.21 to close the "Loads" window and return to *ELPLA* main window.

Creating the project of the chimney with a hyperbolic shell wall 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 11.24.

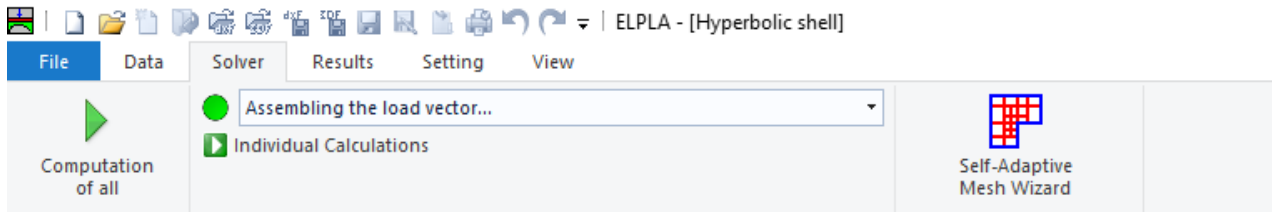


Figure 11.24 "Solver" Tab

*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 11.25 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.

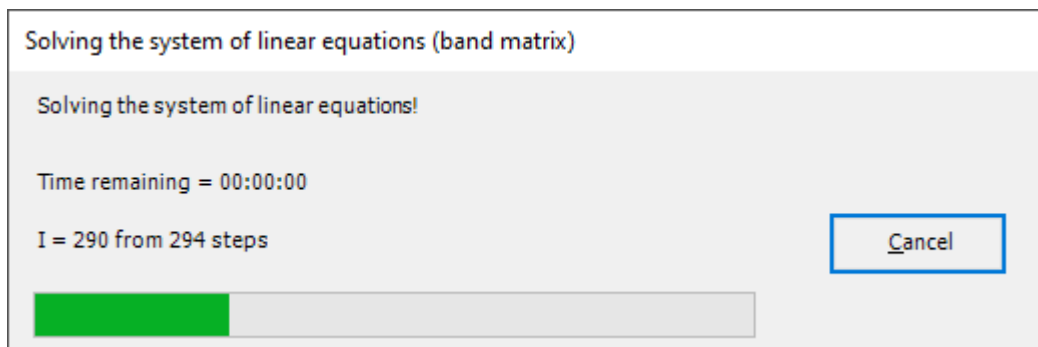


Figure 11.25 Analysis progress menu

## Example 11

---

### Check of the solution

Once the analysis is carried out, a check menu of the solution appears, Figure 11.26. This menu compares between the values of actions and reactions. Through this comparative examination, the user can assess the calculation accuracy.

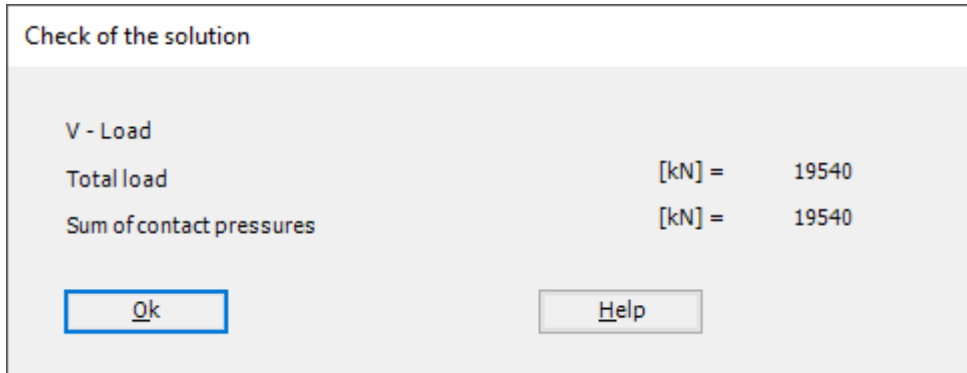


Figure 11.26 Menu "Check of the solution"

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 11.27).

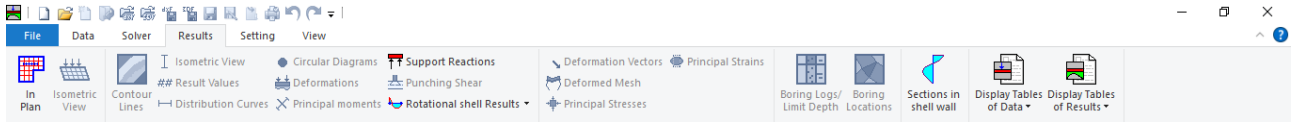


Figure 11.27 "Results" Tab

The "Results" 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
- Supports reactions
- Sections in shell wall
- Display tables of data
- Display tables of results

To view the radial forces in the shell wall

- From "Rotational shell results" command in the "Results" menu, choose "In Plan" command. The following option box in Figure 11.28 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 11.29.

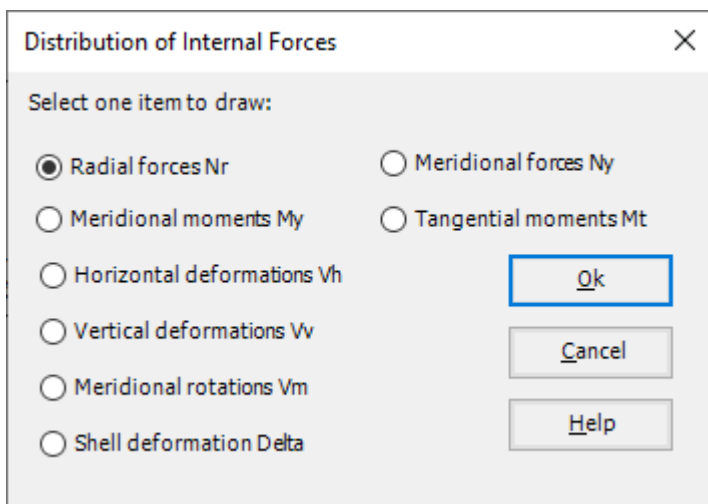


Figure 11.28 "Distribution of internal forces" option box

## Example 11

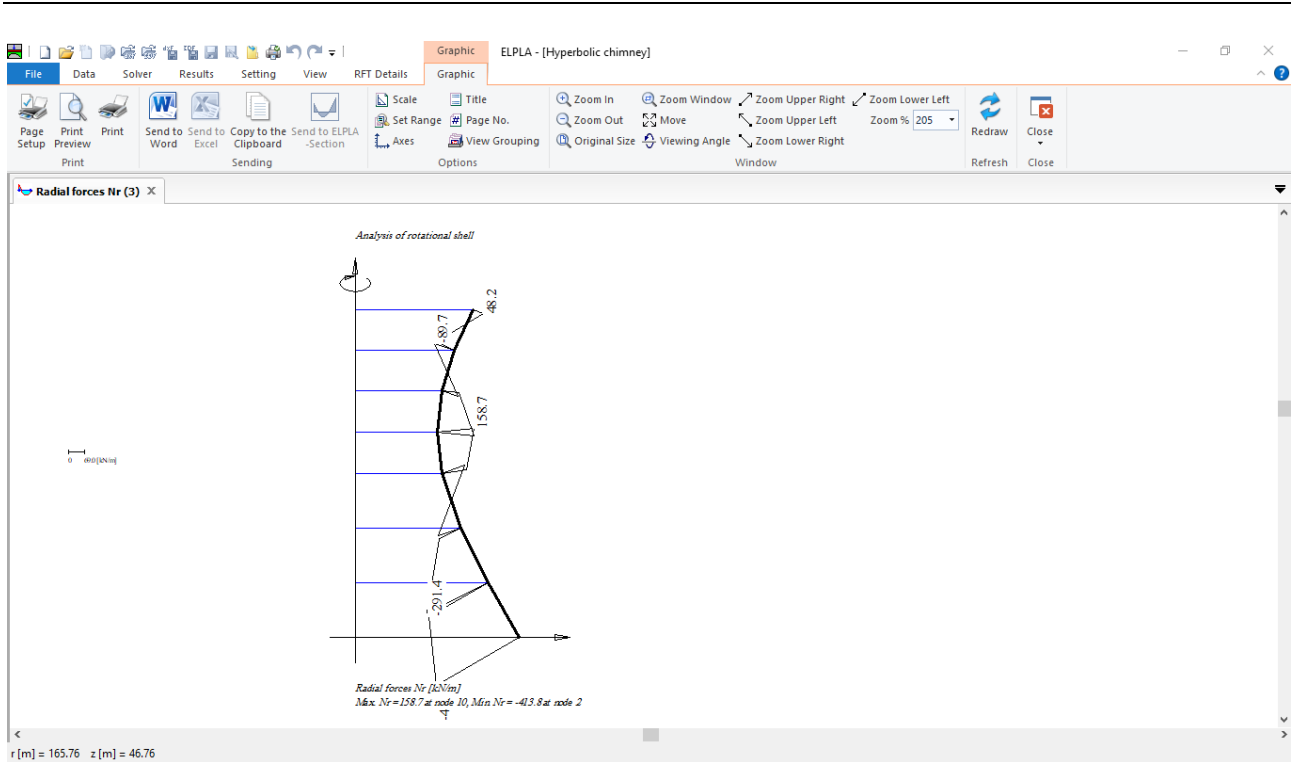


Figure 11.29 Radial forces in shell wall

To view the vertical deformations of the chimney wall

- From "Rotational shell results" command in the "Results" menu, choose "In Plan" command. The following option box in Figure 11.30 appears
- In the "Distribution of Internal Forces" option box, select "Vertical deformations  $V_V$ " as an example for the results to be displayed
- Click "OK" button

The Results are now displayed as shown in Figure 11.31.

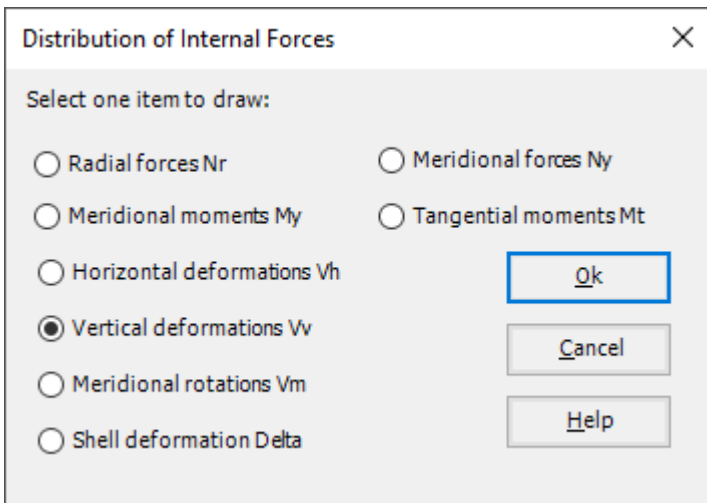


Figure 11.30 "Distribution of internal forces" option box

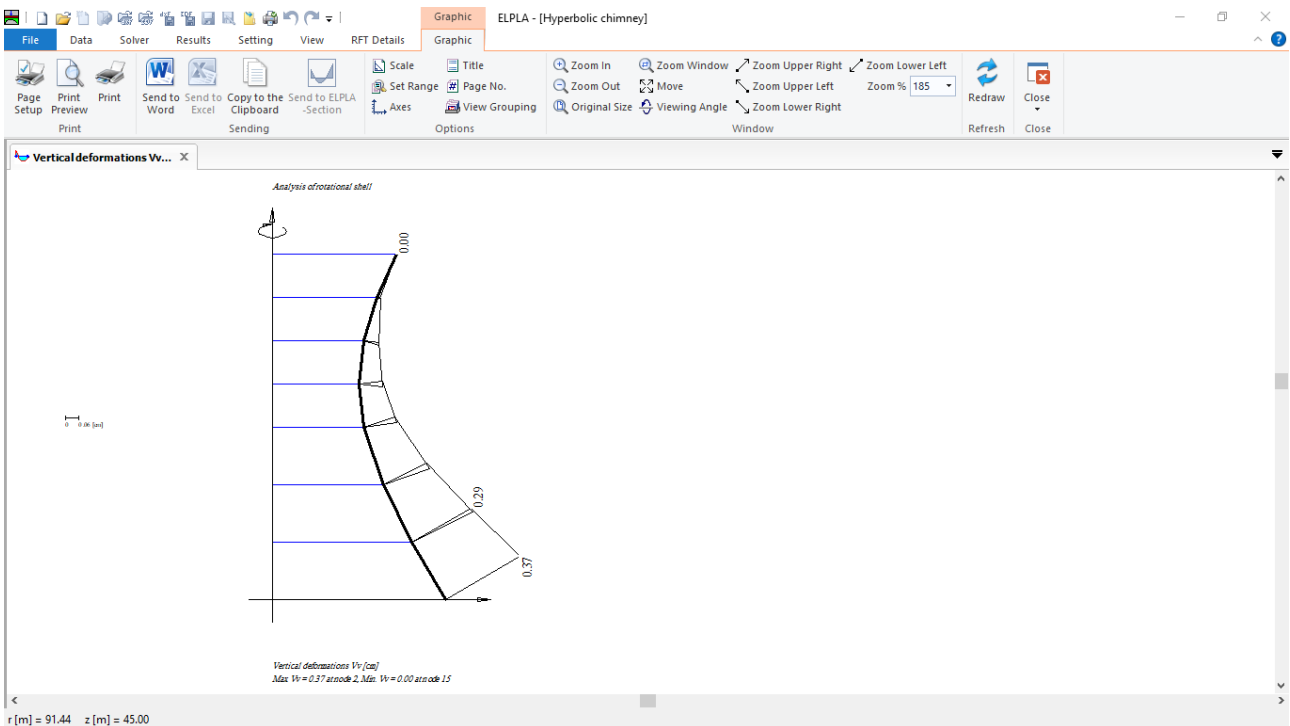


Figure 11.31 Vertical deformations of the chimney wall

## Example 11

---

To view the slab thickness of the chimney

- Choose "Slab thickness" from "In Plan" command in "Data" menu. The following option box in Figure 11.32 appears
- Click "OK" button

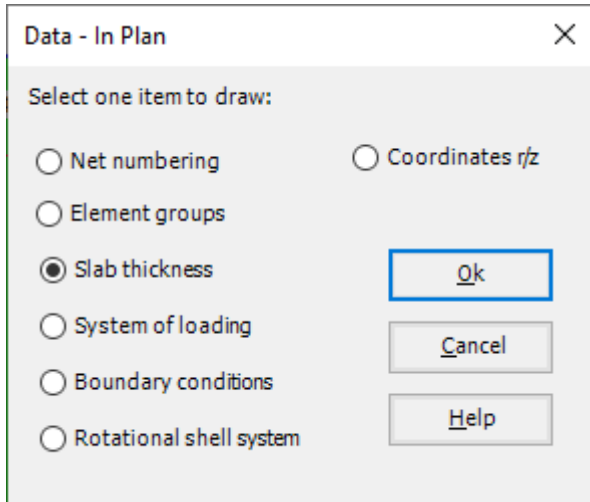


Figure 11.32 "Data – In Plan" option box

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 11.33 appears
- In this check group box, check "Supports /Boundary Conditions" check box
- The user can choose any other data to be viewed
- Click "OK" button

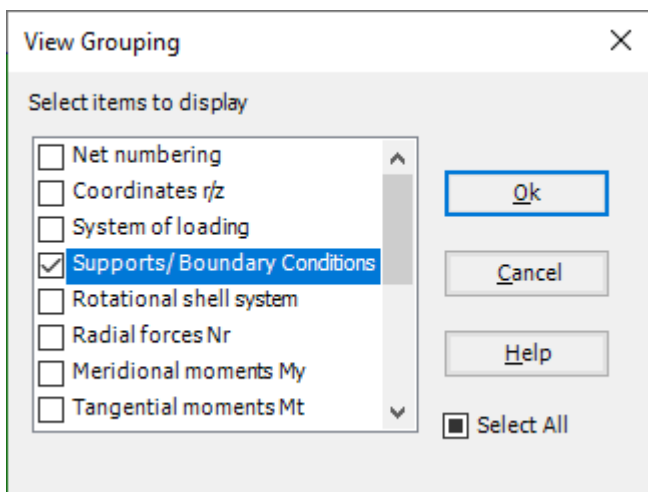


Figure 11.33 "View Grouping" check group box

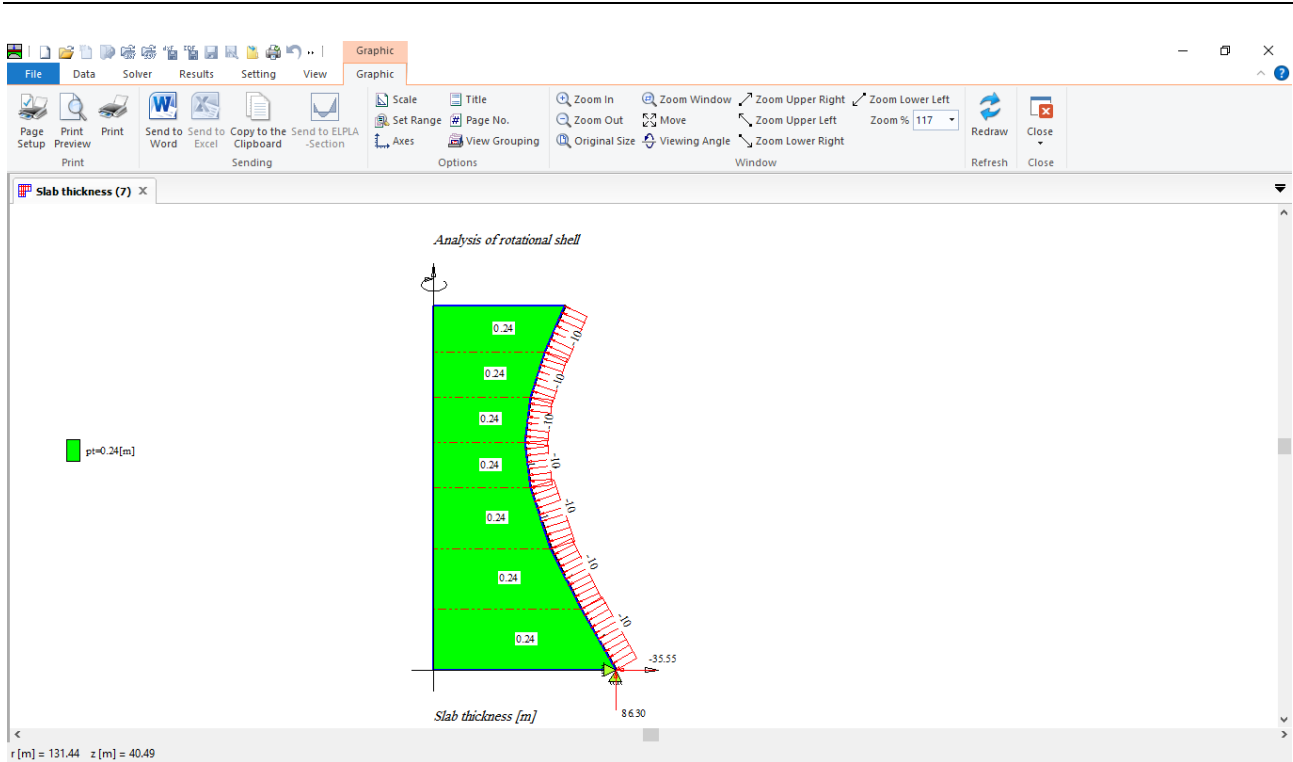


Figure 11.34 Slab thickness with the load system