

## Example 8. Geometrically nonlinear analysis of mast

In this lesson you will learn how to:

- generate design model of mast;
- simulate geometric nonlinearity.

Description:

Model of the mast and its boundary conditions are presented in Figure 8.1.

Steel mast of 40m height.

Sections of elements of the mast:

- mast – three pipes 133 x 5;
- guy ropes – cable, shape – 20.

The state of design model is evaluated after 365 and 730 days.

Loads:

- load case 1 – dead weight; concentrated load  $P = 0.15$  t/m applied to two upper nodes;
- load case 2 – wind load, II wind region, site type A.

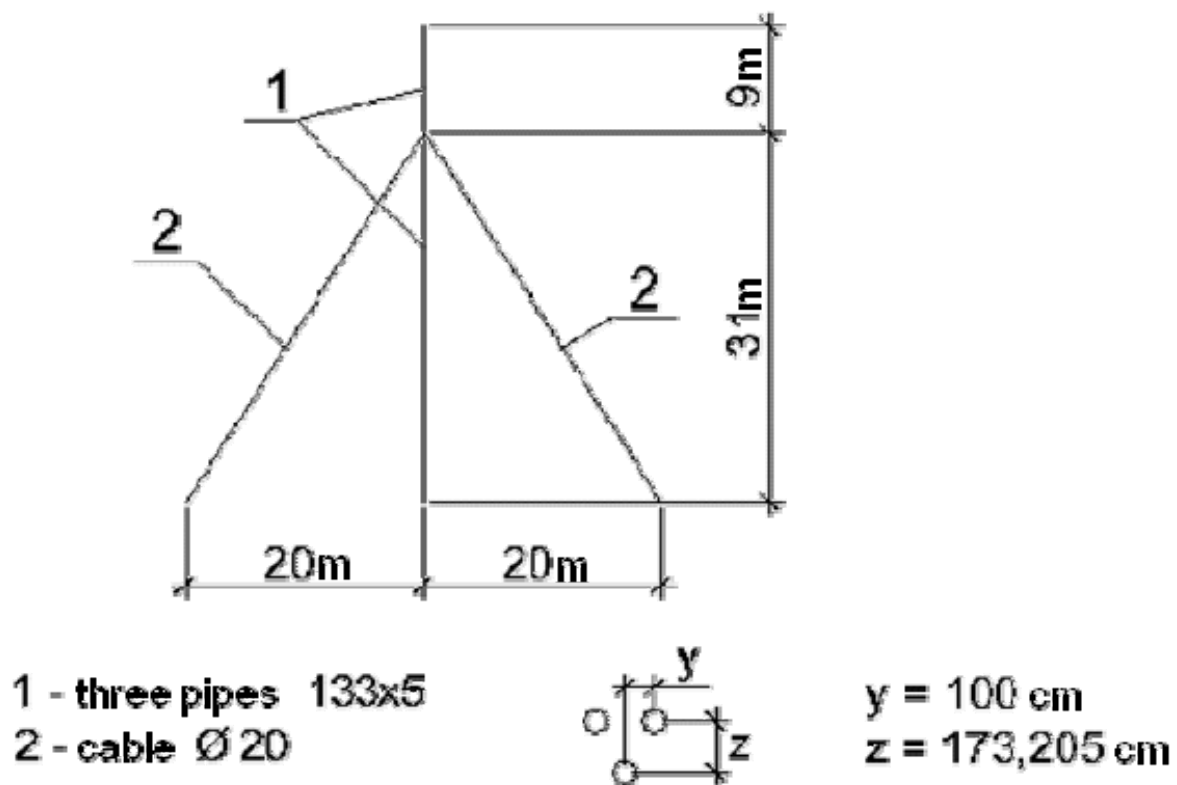




Figure 8.1 Model of the mast

- ⇒ On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2015** and then click **LIRA-SAPR 2015**.

### Step 1. Creating new problem

- ⇒ On the FILE menu, click **New** (button  on the toolbar).
- ⇒ In the **Model type** dialog box (see Figure 8.2) specify the following data:
- problem name – **Example8**;
  - problem code (by default it coincides with the problem name);
  - model type – **2 – Three degrees of freedom per node** (two translations and rotation) X0Z.
- ⇒ Click **OK** .

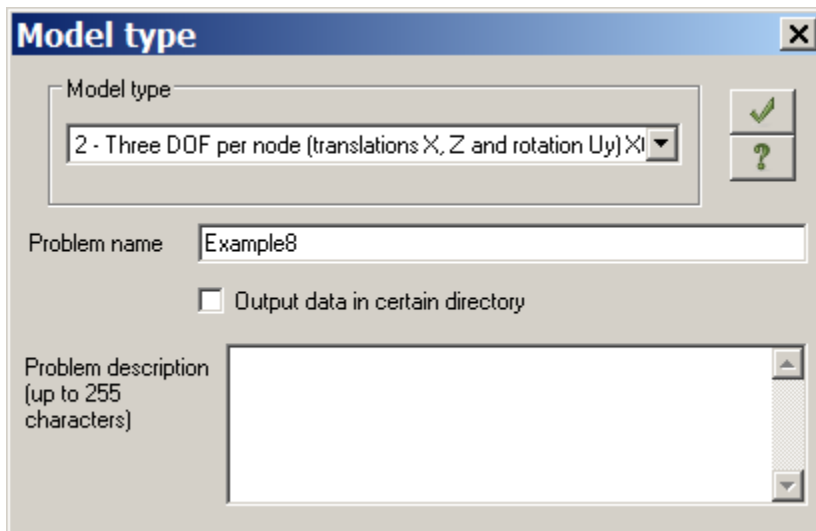




Figure 8.2 **Model type** dialog box



*It is also possible to open the **Model type** dialog box with a pre-defined type of model. To do this, on the **LIRA-SAPR menu** (Application menu), point to **New** and click **Model type 2 (Three DOF per node)***

*command . One more way to do the same: on the Quick Access Toolbar, click **New** and in the drop-*

*down menu select **Model type 2 (Three DOF per node)** command . Then you should define only problem name.*



*To save all output data files for the problem in certain directory, select appropriate check box. The directory name will coincide with the name of the problem. This directory will appear in the directory for files with analysis results. This is helpful if you have to find output data files for certain problem, then transfer these files or review and evaluate them with the help of Windows Explorer or other file managers.*

### Step 2. Generating model geometry

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** drop-down list and click the **Create frame**  command.
- ⇒ In the **Create plane fragments and grids** dialog box specify the following data:

- spacing along the second axis:

<b>L(i)</b>	<b>N</b>
1	40

- other parameters remain by default (see Figure 8.3).

⇒ Click **Apply** .

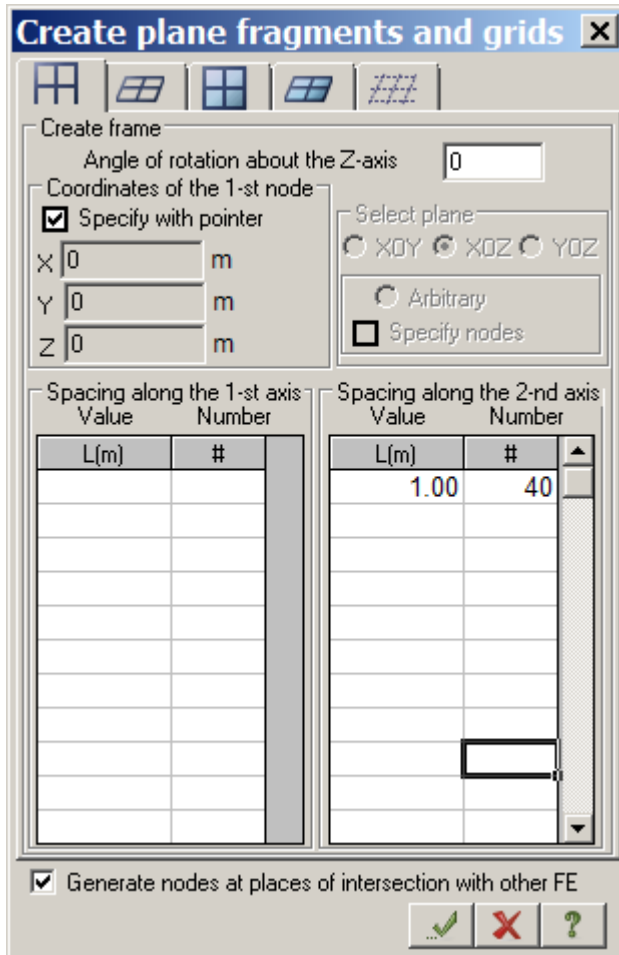


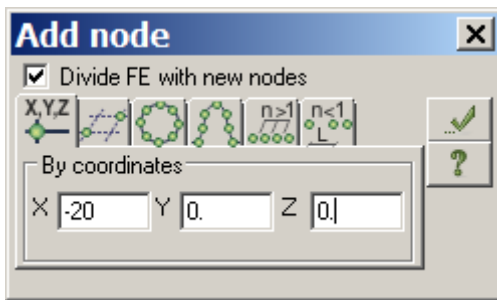


Figure 8.3 **Create plane fragments and grids** dialog box

To add nodes:

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add node** drop-down list and click the **Add node by coordinates**  command.
- ⇒ In the **Add node** dialog box (see Figure 8.4), define coordinates for the left bottom node:
- X(m) Y(m) Z(m)  
-20 0 0.
- ⇒ Click **Apply** .


Figure 8.4 **Add node** dialog box

⇒ Then define coordinates for the right bottom node:

▪ X(m)    Y(m)    Z(m)  
      20        0        0.

⇒ Click **Apply** .

To present numbers of nodes on the screen:

⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **Flags of drawing** button .

⇒ In the **Display** dialog box, select the **Node numbers** check box on the **Nodes** tab.

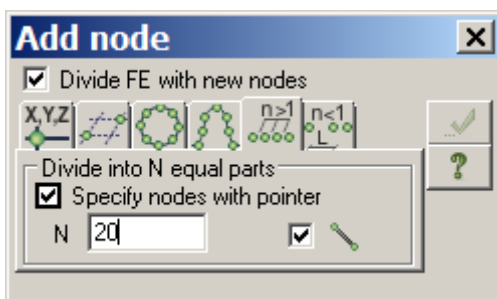
⇒ Click **Redraw** .

To add bar elements:

⇒ In the **Add node** dialog box (see Figure 8.5), click **Divide into N equal parts** tab and define N = 20.

⇒ In the same dialog box, select the **Specify nodes with pointer** and **Join nodes with bars** (  ) check boxes.

⇒ Select with the pointer nodes No.42 and 32 and then No.43 and 32 in sequence (the rubber-band line is automatically stretched between the nodes that you select).

Figure 8.5 **Add node** dialog box

The model that you will obtain is presented in Fig.8.6.

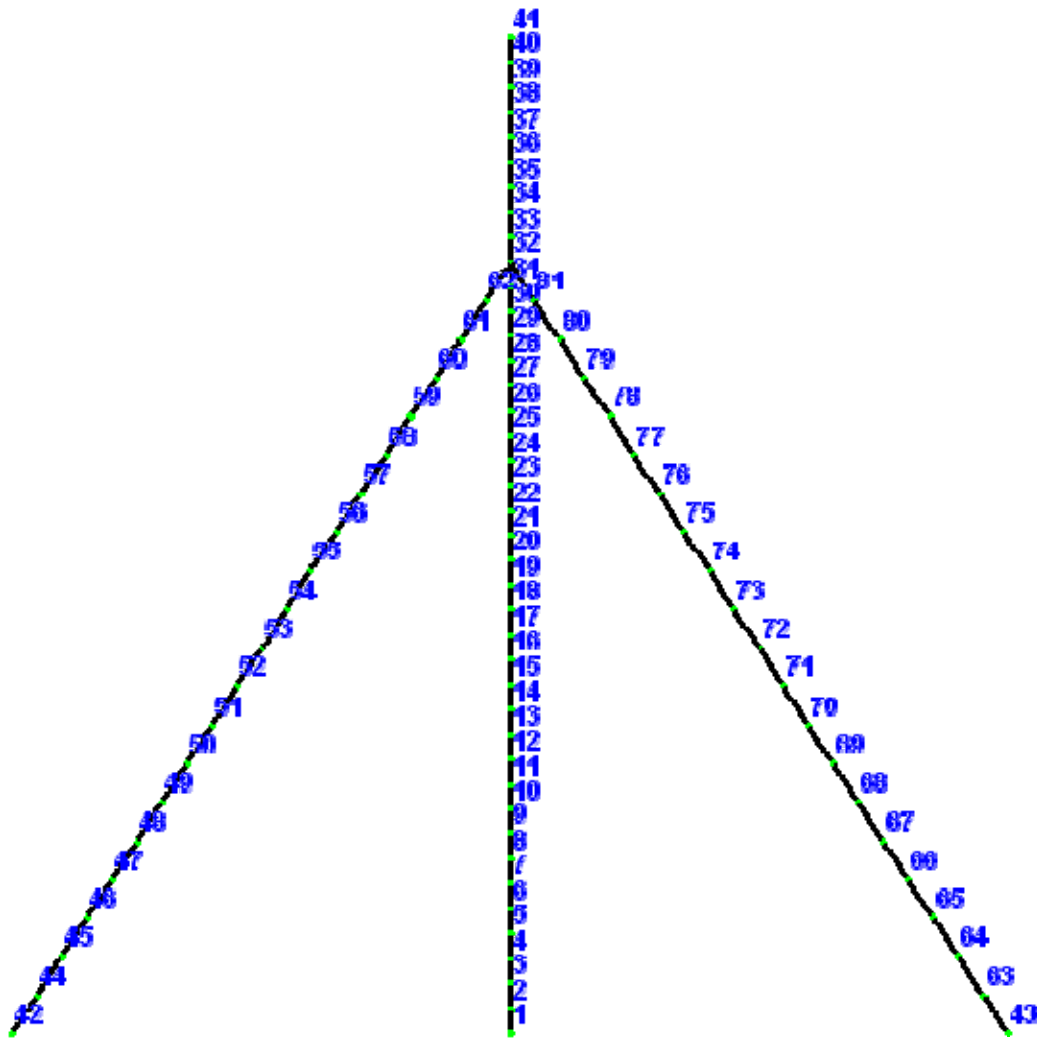




Figure 8.6 Design model with node numbers

To save data about design model:

- ⇒ On the **LIRA-SAPR menu** (Application menu), click **Save** command .
- ⇒ In the **Save as** dialog box specify the following data:
  - file name – **Example8**;
  - location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

### Step 3. Defining boundary conditions

To select nodes No.42 and 43:

- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Select nodes No.42 (left bottom node) and No.43 (right bottom node) with the pointer (the nodes will be coloured red).

To define boundary conditions for nodes No.42 and 43:





- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Restraints** button .
- ⇒ In the **Restraints on nodes** dialog box (see Fig.8.7) specify directions along which displacements of nodes are not allowed (X, Z). To do this, select appropriate check boxes.
- ⇒ Click **Apply**  (the nodes will be coloured blue).






Figure 8.7 **Restraints on nodes** dialog box

To define boundary conditions for node No.1:

- ⇒ Select node No.1 with the pointer.
- ⇒ In the **Restraints on nodes** dialog box specify directions along which displacements of nodes are not allowed (X, Z, UY). To do this, select appropriate check boxes.
- ⇒ Click **Apply** .
- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button  in order to make this command not active.

#### Step 4. Changing FE types

- ⇒ On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .
- ⇒ Select all elements of the model with the pointer.
- ⇒ On the **Advanced edit options** ribbon tab, on the **Model** panel, click **Change FE type** .
- ⇒ In the **Change FE type** dialog box (see Fig.8.8), in the list of FE types, select **FE type 310 - geometrically nonlinear arbitrary 3D bar (cable)**.
- ⇒ Click **Apply** .

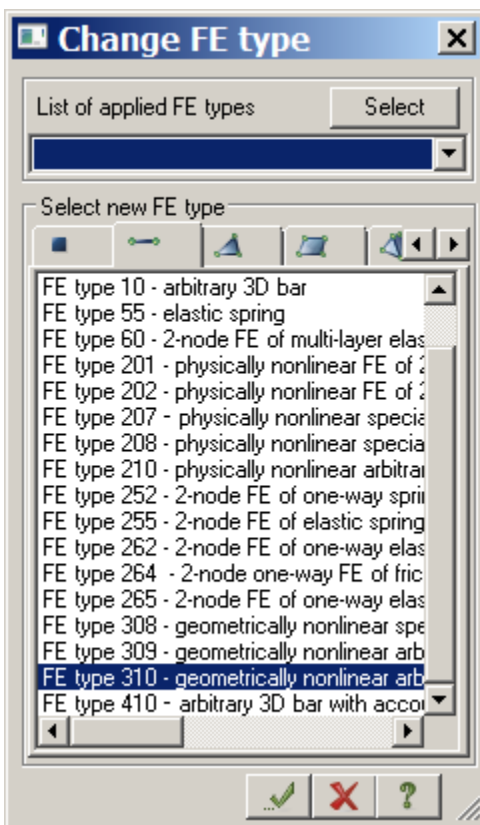



Figure 8.8 Change FE type dialog box

### Step 5. Defining material properties

To create material data sets:

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Material properties** button .
- ⇒ In the **Stiffness and materials** dialog box (see Fig.8.9a), click **Add**. The dialog box expands to display the library of stiffness parameters. In the **Add stiffness** dialog box (see Fig.8.9b), select the **Database of steel sections** tab (the second tab).

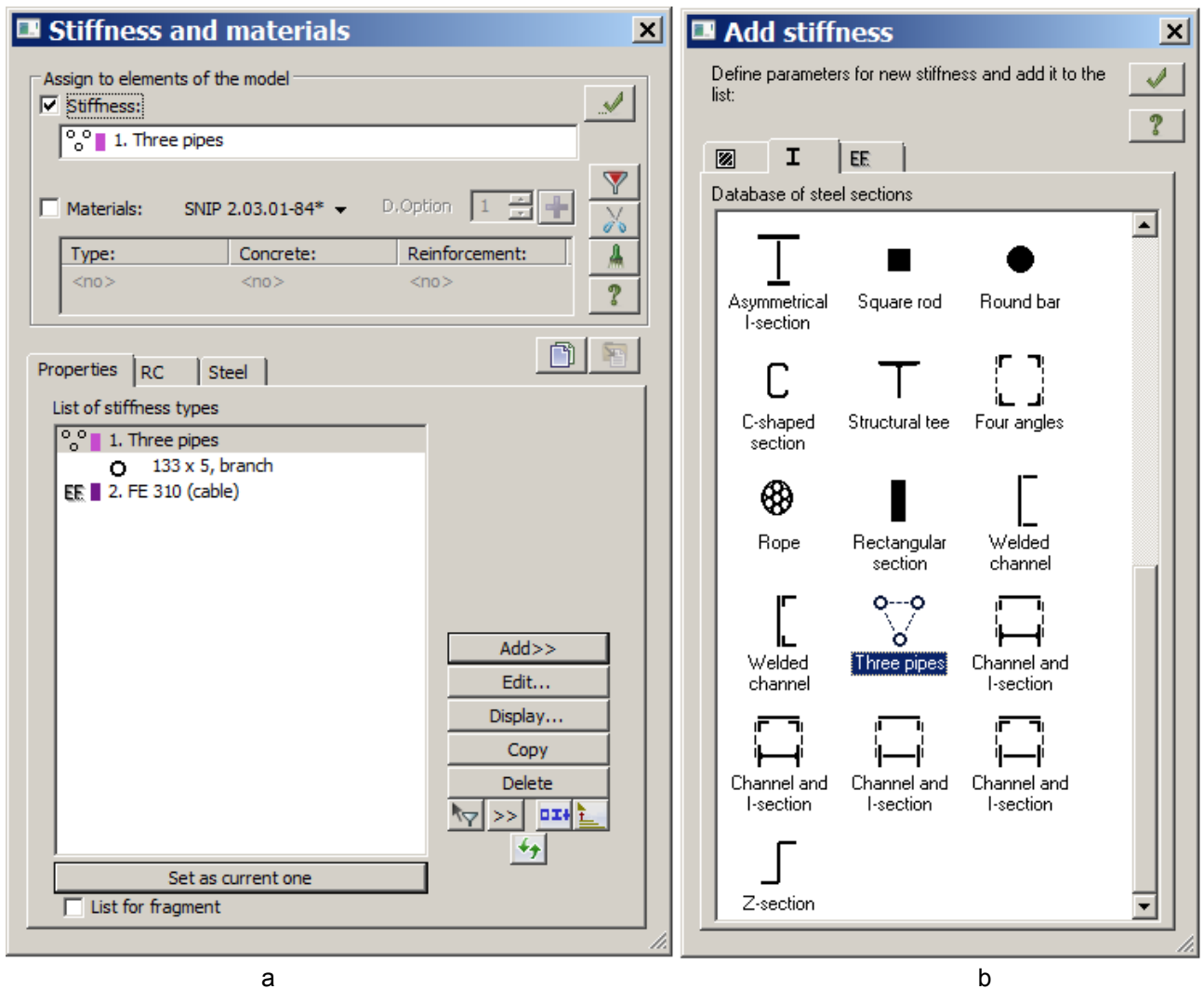


Figure 8.9 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ Double-click the **Three pipes** icon in the list. The **Steel cross-section** dialog box opens. In this dialog box you can define material properties for selected type of the section.
- ⇒ In the **Steel cross-section** dialog box, select **branch** and specify the following parameters for **Three pipes** section (see Fig.8.10):
  - in the **Profile** box, click **Труба бесшовная горячекатаная** ;
  - in the **Shape** box, click 133 x 5.
- ⇒ Click **Coupling**.



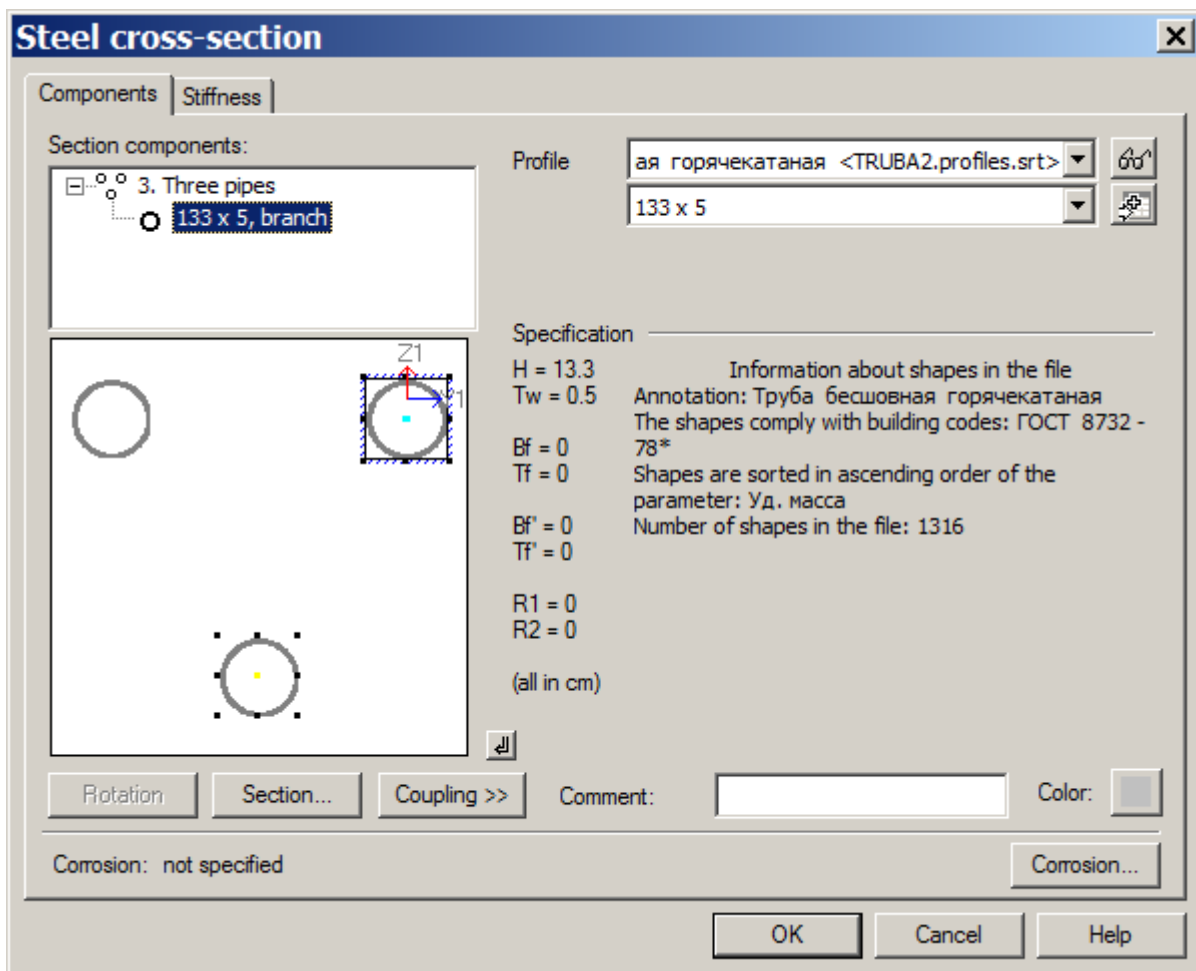


Figure 8.10 Steel cross-section dialog box

- ⇒ In the **Coupling** dialog box (see Fig.8.11) specify  $Y = 100$  cm (if the **Equilateral triangle** check box is selected, then  $Z$  value will be calculated automatically).
- ⇒ Click **OK**.

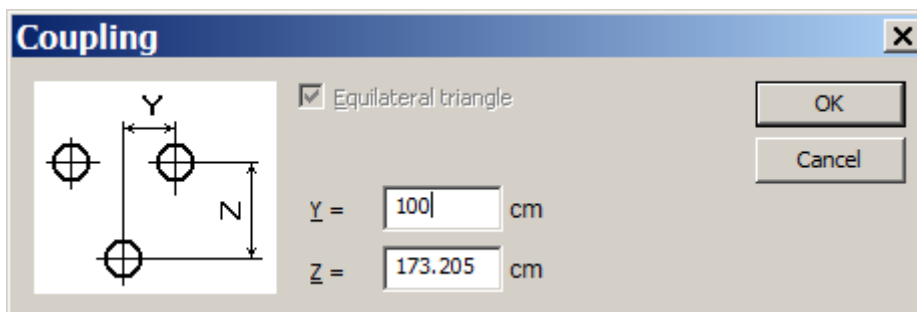


Figure 8.11 Coupling dialog box

- ⇒ In the **Steel cross-section** dialog box, click **OK**.
- ⇒ In the **Stiffness of elements** dialog box, click the tab with numerical description of stiffness (the third tab).
- ⇒ Double-click **FE 310 (cable)** icon.
- ⇒ In the **Numerical description for FE 310 (cable)** dialog box (see Fig.8.12), select the **Wire rope** option.

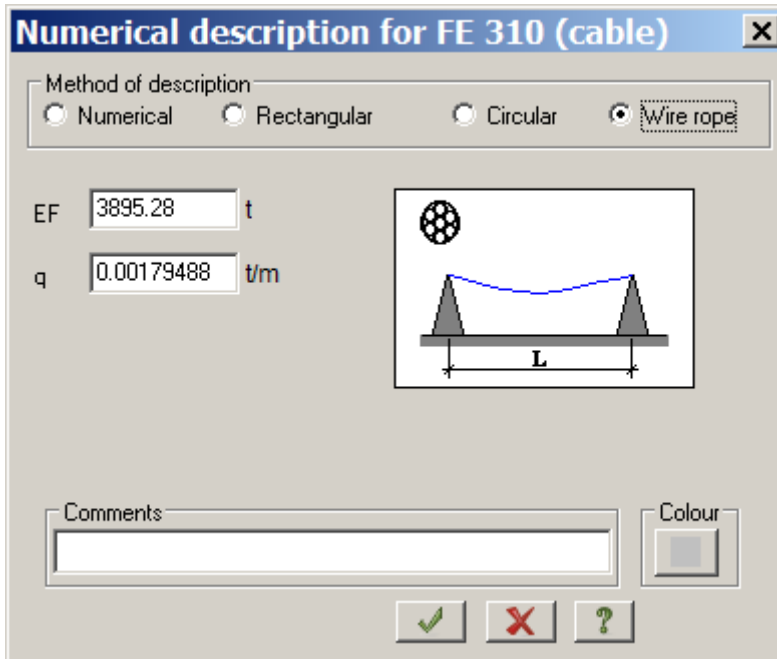


Figure 8.12 Numerical description for FE 310 (cable) dialog box

⇒ In the **Steel cross-section** dialog box (see Fig.8.13), specify parameters for **Cable** section:

- in the **Profile** box, click **Кабель однопровитой канатной тросовой конструкции 1х37(1+6+12+18)** ;
- in the **Shape** box, click 20.

⇒ Click **OK** .



*Axial stiffness ( $EF$ ) and weight per unit length ( $q$ ) will be calculated automatically and displayed in the appropriate fields of the **Numerical description for FE 310 (cable)** dialog box.*

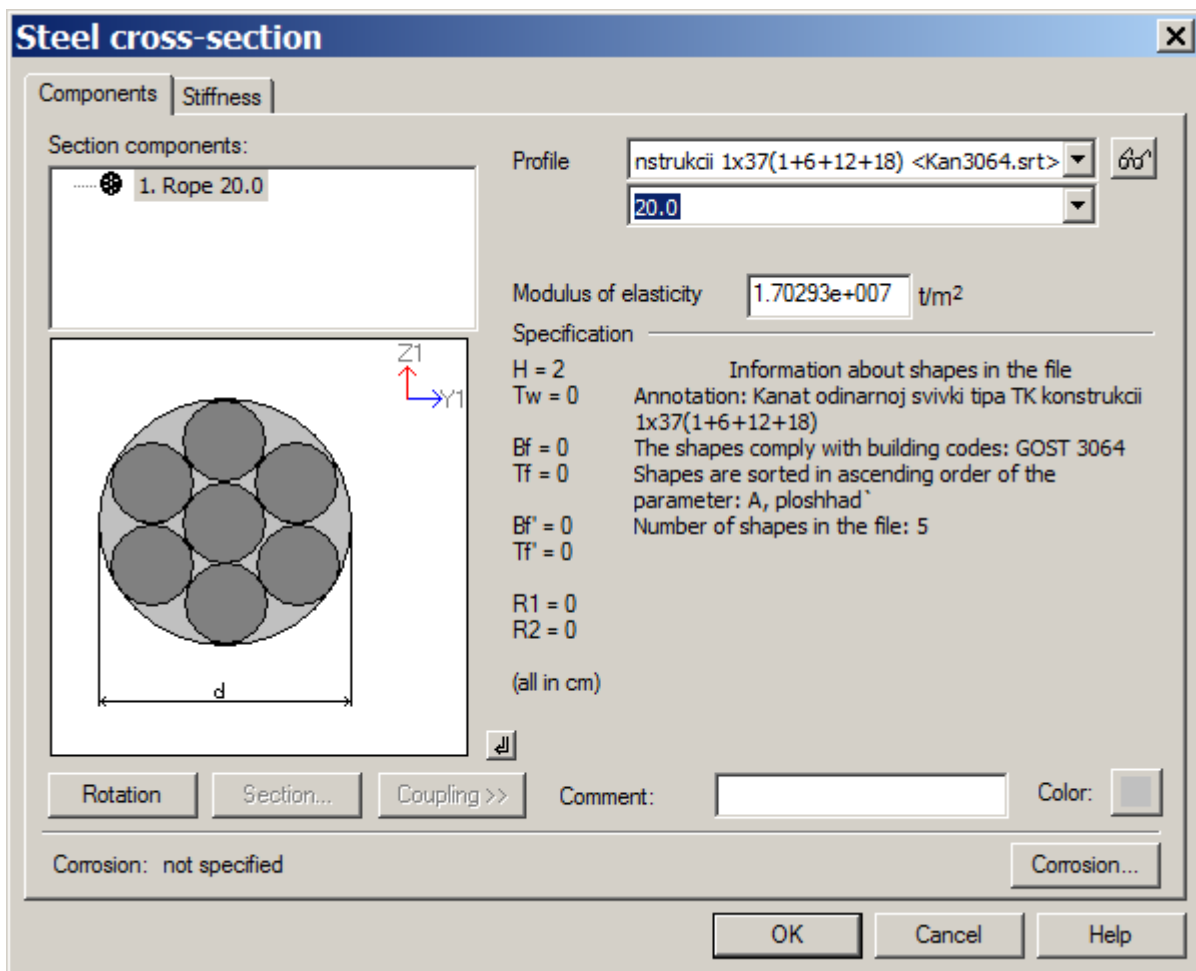







Figure 8.13 Steel cross-section dialog box

- ⇒ Click **OK**  in the **Numerical description for FE 310 (cable)** dialog box.
- ⇒ To hide library of stiffness properties, in the **Stiffness of elements** dialog box click **Add** unfold button.

To assign material properties to elements of the mast:

- ⇒ On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .
- ⇒ Select all elements of the model with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ Then, in the same dialog box, in the **List of stiffness types**, select the stiffness type '**1.Three pipes**'.
- ⇒ Click **Set as current type**. In this case selected type will be displayed in the **Stiffness** box in the **Assign to elements of the model** area. To assign current type of stiffness, you could also double-click appropriate row in the list.
- ⇒ On the **Select** menu, click **Select vertical elements** (button  on the toolbar).
- ⇒ Select all vertical elements of the model with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ The **Warning** box is displayed (see Fig.8.14). Click **OK**.

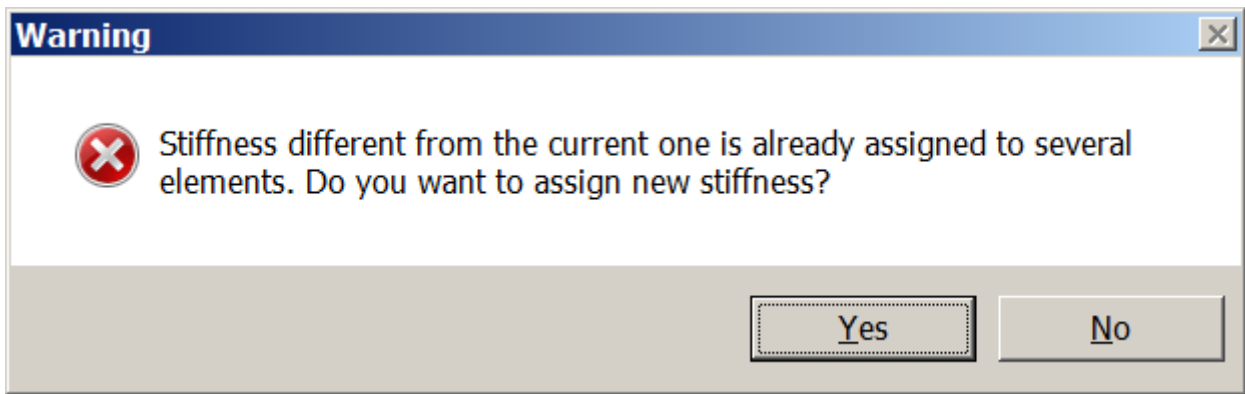




Figure 8.14 Warning box

### Step 6. Applying loads

To create load case No.1:

- ⇒ To define load from dead weight of the slab, on the **Create and edit** ribbon tab, select the **Loads** panel and click **Add dead weight** .
- ⇒ In the **Add dead weight** dialog box (see Fig.8.15), click **All elements** and specify **Load factor** as equal to **1.05** (as in [SRS-SAPR \(Steel Tables\)](#) module the unit weight is specified as normative value, it should be converted to design value).
- ⇒ Click **Apply**  (uniformly distributed load equal to unit weight of elements is automatically applied to all elements of the structure).

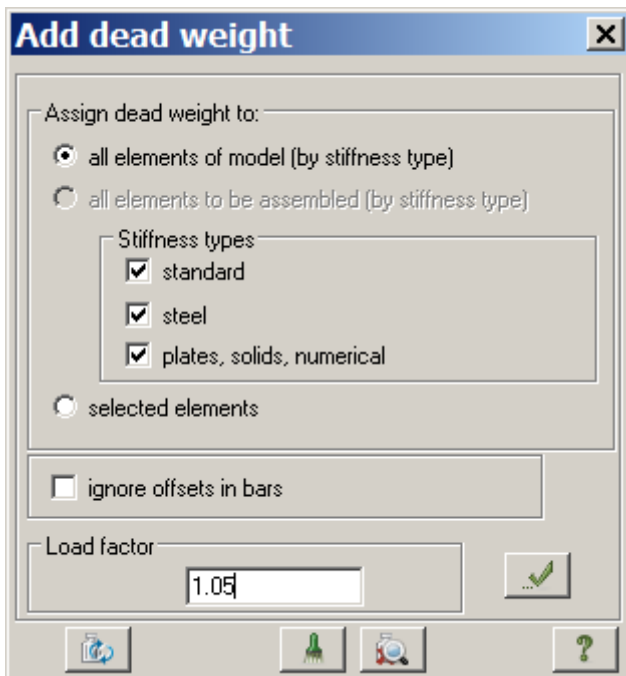




Figure 8.15 Add dead weight dialog box

- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Select nodes No.40 and 41 with the pointer.

- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel, then select **Load on nodes** command  from the **Loads on nodes and elements** drop-down list.
- ⇒ In the **Define loads** dialog box (see Fig.8.16), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

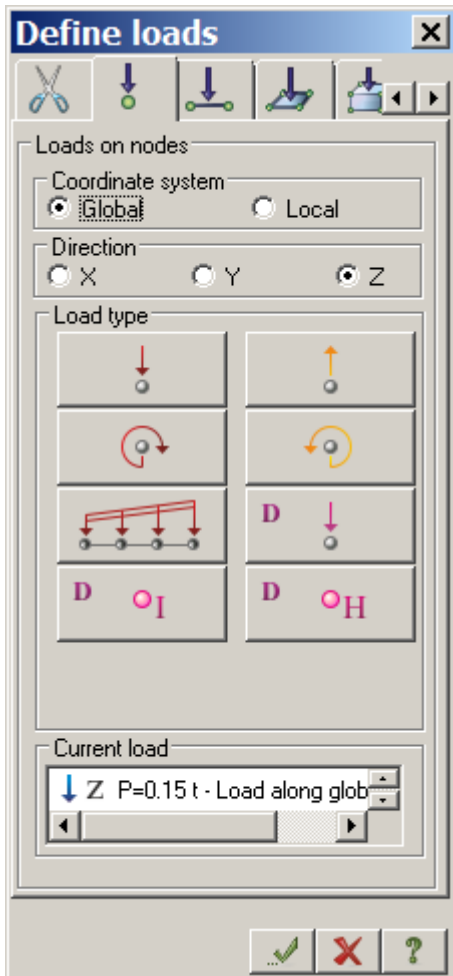



Figure 8.16 Define loads dialog box

- ⇒ In the **Load type** area, click the **Concentrated load** button .
- ⇒ In the **Load parameters** dialog box specify  $P = 0.15t$  (see Fig.8.17).
- ⇒ Click **OK**.

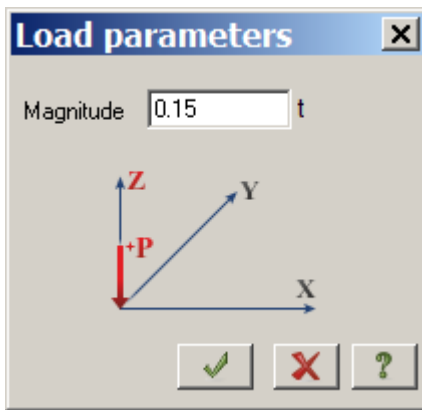








Figure 8.17 Load parameters dialog box

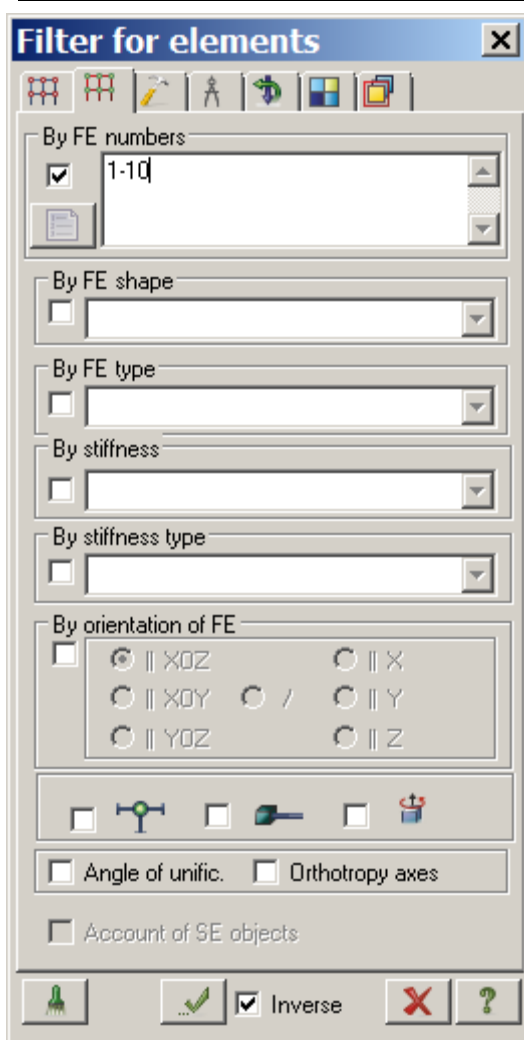
⇒ In the **Define loads** dialog box, click **Apply** .


To present numbers of elements on the screen:

- ⇒ On the **Select** toolbar, click **Flags of drawing** button .
- ⇒ In the **Display** dialog box, select the **Element numbers** check box on the **Elements** tab.
- ⇒ Click **Redraw** .

To create load case No.2:

- ⇒ To change the number of the current load case, click the **Next load case** button  located on the Status bar or on the toolbar.
- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **PolyFilter** .
- ⇒ In the **PolyFilter** dialog box, select the **Filter for elements** tab (the second tab) (see Fig.8.18).
- ⇒ Select **By FE numbers** check box and specify the element numbers '1-10'.
- ⇒ Click **Apply** .

Figure 8.18 **Filter for elements** tab

- ⇒ In the **Define loads** dialog box, select the **Load on bars** tab  (the third tab).
- ⇒ Specify direction of load along the X-axis (see Fig.8.19).

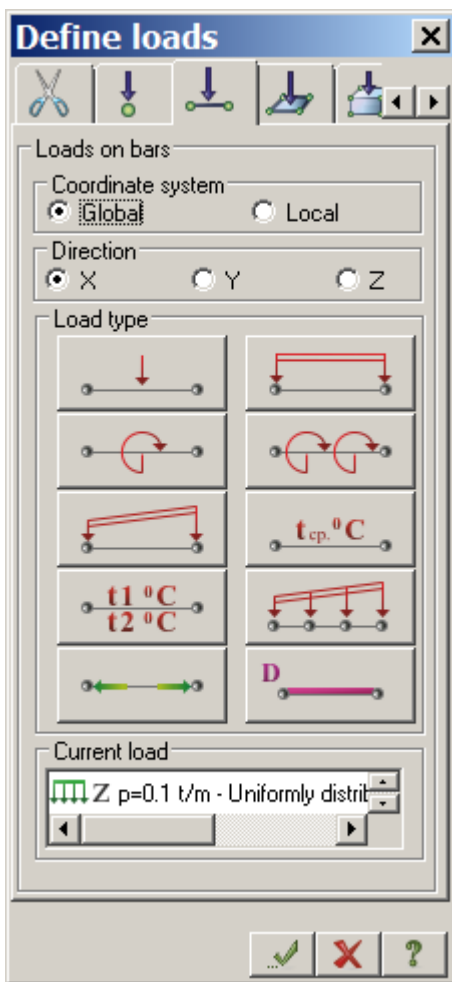


Figure 8.19 Define loads dialog box

⇒ In the **Load type** area, click **Uniformly distributed load** button



⇒ In the **Load parameters** dialog box specify  $p = 0.1 \text{ t/m}$ .

⇒ Click **OK** .

⇒ In the **PolyFilter** dialog box, select the **Filter for elements** tab (the second tab).

⇒ Select **By FE numbers** check box and specify the element numbers '11-20'.

⇒ Click **Apply** .

⇒ In the **Load type** area, click **Trapezoidal load on group of bars** button



⇒ In the **Non-uniformly distributed load** dialog box (see Fig.8.20), specify  $P1 = 0.1 \text{ t/m}$ ,  $P2 = 0.12 \text{ t/m}$  and direction along which the load is changed (Z-axis).

⇒ Click **OK**.



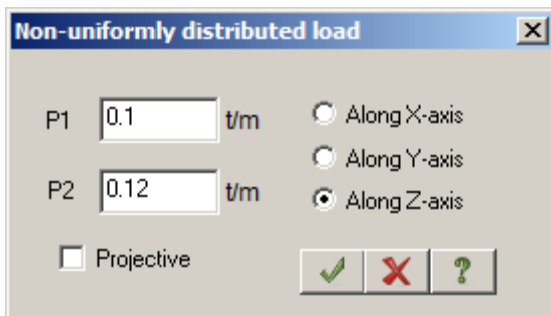


Figure 8.20 Non-uniformly distributed load dialog box

⇒ In the **PolyFilter** dialog box, select the **Filter for elements** tab (the second tab).

⇒ Select **By FE numbers** check box and specify the element numbers '21-40'.

⇒ Click **Apply** .

⇒ In the **Load type** area, click **Trapezoidal load on group of bars** button .


⇒ In the **Non-uniformly distributed load** dialog box, specify P1 = 0.12 t/m, P2 = 0.15 t/m and direction along which the load is changed (Z-axis).

⇒ Click **OK**.

⇒ In the **Define loads** dialog box, click **Apply** .


### Step 7. Modelling nonlinear load cases with account of creep in concrete

⇒ On the **Analysis** ribbon tab, on the **Nonlinearity** panel, click **Step-type method** (button  on the toolbar).

⇒ In the **Model nonlinear load cases of structure** dialog box (see Figure 8.21), click the **Add** button  (in the left part of the dialog box, under **History**, the first load history is added and the row with load case indicated with question mark will become selected automatically).

⇒ For the first load case define the following parameters:

- load case No. – 1;
- select **Step (4) Assign step automatically for geometrically and physically nonlinear problems** in the **Analysis method** list box;
- select **Displacement and forces after every step** in the **Print results** list box;
- in the **Display intermediate results** list, select **Display all** option.

⇒ To add rows for parameters of the second load case, select the row for the first load case and then click the **Add** button .

⇒ Select the row for the second load case and define the following parameters:

- load case No. – 2;
- select **Step (4) Assign step automatically for geometrically and physically nonlinear problems** in the **Analysis method** list box;
- select **Displacement and forces after every step** in the **Print results** list box;

- in the **Display intermediate results** list, select **Display all** option.

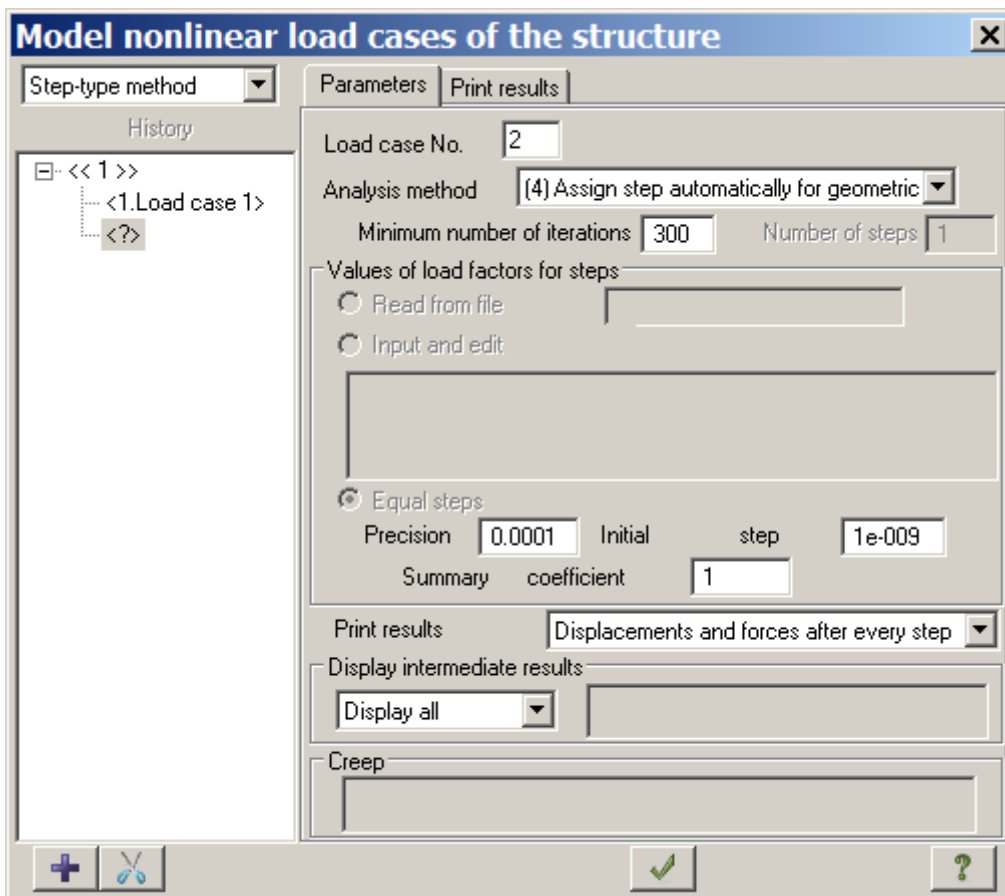




Figure 8.21 **Model nonlinear load cases of structure** dialog box

⇒ To confirm the data, click **Apply** .


### Step 8. Geometrically nonlinear analysis of mast

⇒ To carry out complete analysis, on the **Analysis** ribbon tab, select the **Analysis** panel and in the **Analyse** drop-down list, click **Complete analysis** .

### Step 9. Review and evaluation of static analysis results



*When geometrically nonlinear analysis procedure is complete, to review and evaluate analysis results, select the **Results** ribbon tab.*

⇒ In the mode of analysis results visualization, by default design model is presented with account of nodal displacements (see Fig.8.22). To display the model without nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button .

Nonlinear load case 1

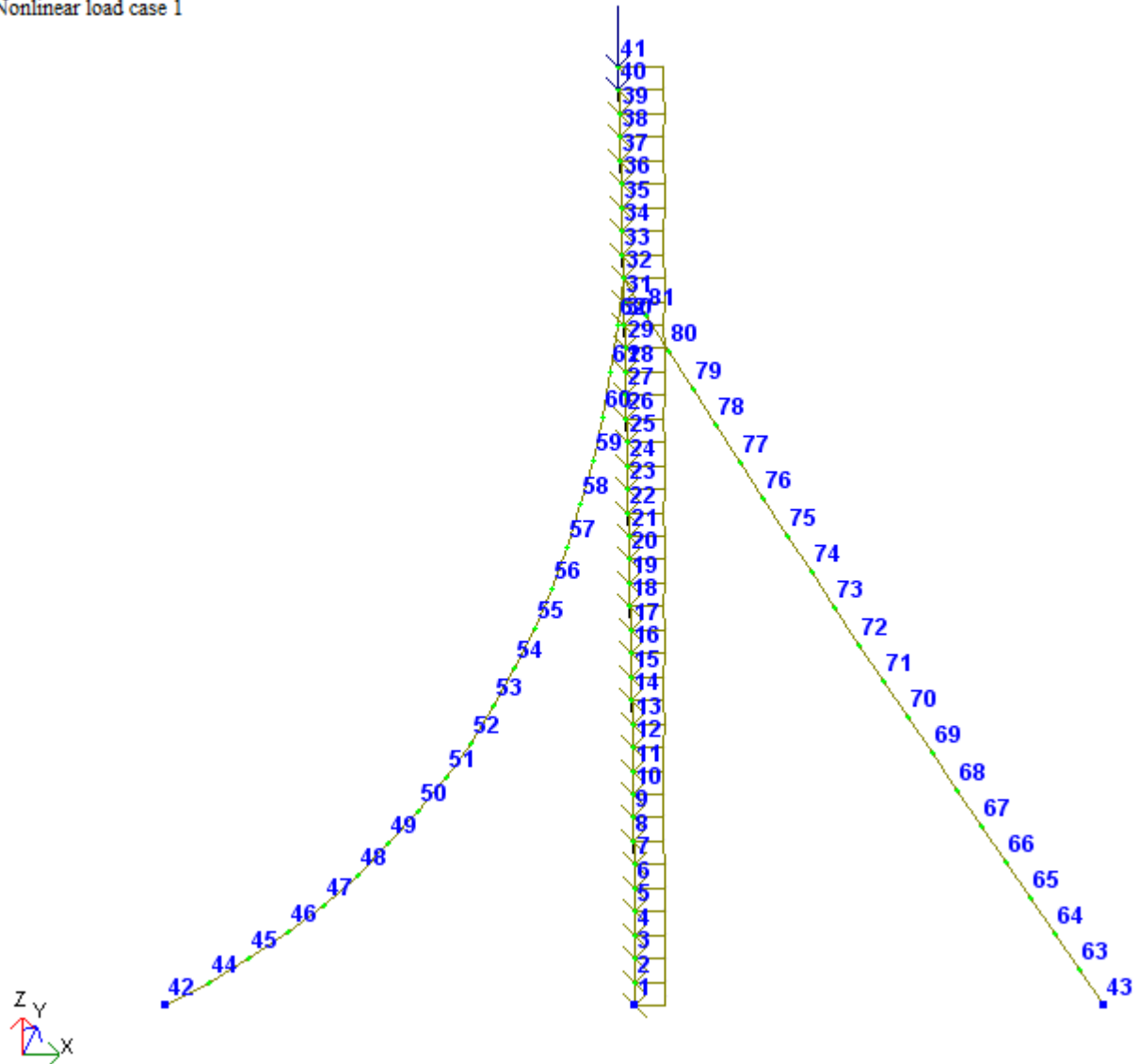






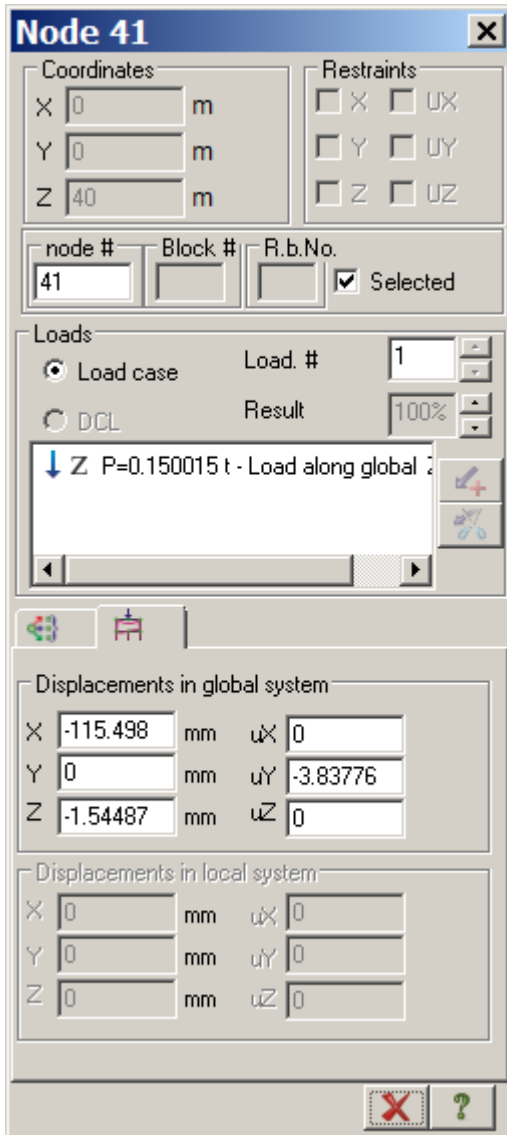
Figure 8.22 Design model with account of nodal displacements

To present mosaic plots for displacements and forces:

- ⇒ To present mosaic plot of displacements along the X-axis, on the **Results** ribbon tab, on the **Deformations** panel, select the **Displacement mosaic plot in global coordinate system** command  in the **Displacement mosaic/contour plot** drop-down list. Then click **Displacements along X** button  on the same panel.
- ⇒ To display mosaic plots **N**, on the **Results** tab, select **Forces in bars** panel and click **Mosaic plot of forces in bars** command  in the **Force diagrams/Mosaic plots** drop-down list.

To display information about certain node:

- ⇒ To preview information about displacements in a certain node, on the **Select** toolbar, click the **Information about nodes and elements** button  and then specify with the pointer certain node, e.g. node No.41.
- ⇒ In the **Information about node** dialog box (see Fig. 8.23) you will see the following data: node No., load case No., value of load on the node, nodal displacements in global coordinate system.
- ⇒ To preview intermediate results, use the **Result** box.



**Node 41**

Coordinates			Restraints		
X	0	m	<input type="checkbox"/> X	<input type="checkbox"/> UX	
Y	0	m	<input type="checkbox"/> Y	<input type="checkbox"/> UY	
Z	40	m	<input type="checkbox"/> Z	<input type="checkbox"/> UZ	

node # 41    Block #    R.b.No.    ☒ Selected

Loads

☒ Load case    Load. # 1

☐ DCL    Result 100%

↓ Z P=0.150015 t - Load along global Z

Displacements in global system

X	-115.498	mm	uX	0
Y	0	mm	uY	-3.83776
Z	-1.54487	mm	uZ	0

Displacements in local system

X	0	mm	uX	0
Y	0	mm	uY	0
Z	0	mm	uZ	0

Figure 8.23 **Information about node No.41** dialog box

To display information about certain element:

- ⇒ Specify with the pointer certain element, e.g. element No. 1. The **Information about element** dialog box (see Fig. 8.24) will appear on the screen.

**Element 1**

Nodes No.

Eler  Block #  ☐ Selected

Stiffness type

FE type  Sect.number  Orthotropy

Length, centre of gravity coordinates  
L=1m, Xc=0m, Yc=0m, Zc=0.5m

☒ Load case Load. #    
☐ DCL Result

☒ Show sect.

N	-6.20173	t
Mx	0	t*m
My	-31.7617	t*m
Qz	2.41702	t
Mz	0	t*m
Qy	0	t
Ry	0	t/m
Rz	0	t/m

☒ Diagrams

Figure 8.24 Information about element No.1 dialog box



The dialog box contains the following data:

- element No.;
- No. of its nodes on the model;
- check box to select the element on the model;
- No. of block where the element is included;
- stiffness type;
- FE type;
- number of design sections;
- length and coordinates of gravity centre of an element in global coordinate system;
- spin box with load case No., section No. and intermediate results.

The dialog box contains the following tabs:

- loads (this tab is active by default);
- subgrade moduli;
- offsets (for bars or plates);
- hinges (for bars);
- rotation angle for local axes;
- forces in the section of bar for the current load case.

To display diagrams of forces and deflections for the bar in the separate **Navigation window**, select the **Diagrams** check box.

To show or hide certain diagram in the **Navigation window**, use appropriate buttons in the window.

- ⇒ To present diagrams of forces and displacements in the **Navigation window** (see Fig.8.25), select the **Diagrams** check box.

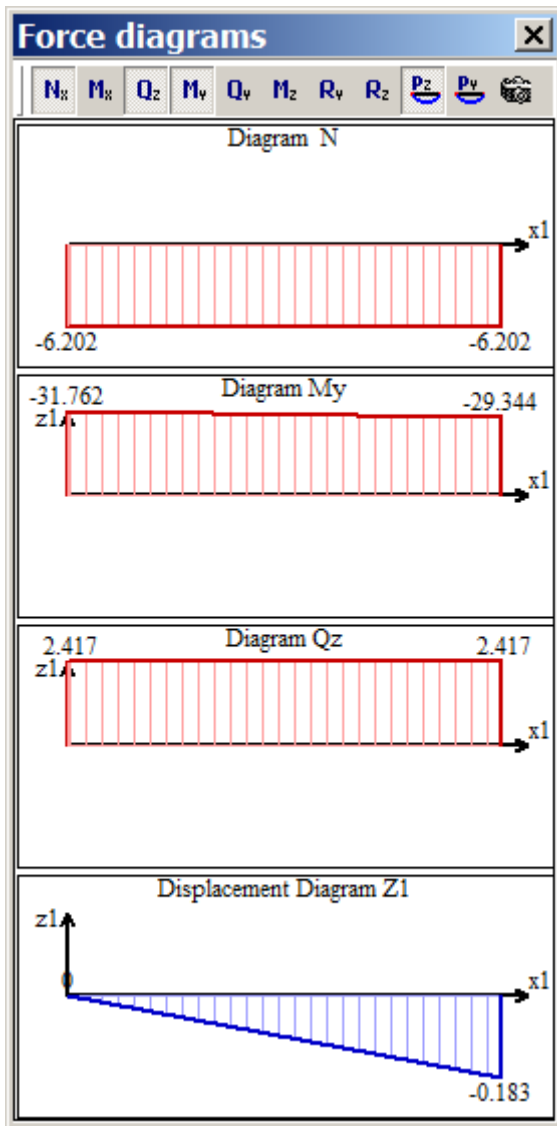



Figure 8.25 **Force diagrams** dialog box

To generate and review tables of analysis results:

- ⇒ To present table with analysis protocol, on the **Results** ribbon tab, select **Tables** panel and click **Standard tables**  in the **Documents** drop-down list.
- ⇒ In the **Standard tables** dialog box (see Fig.8.26), select **Analysis protocol** in the list.
- ⇒ Click **Apply**.

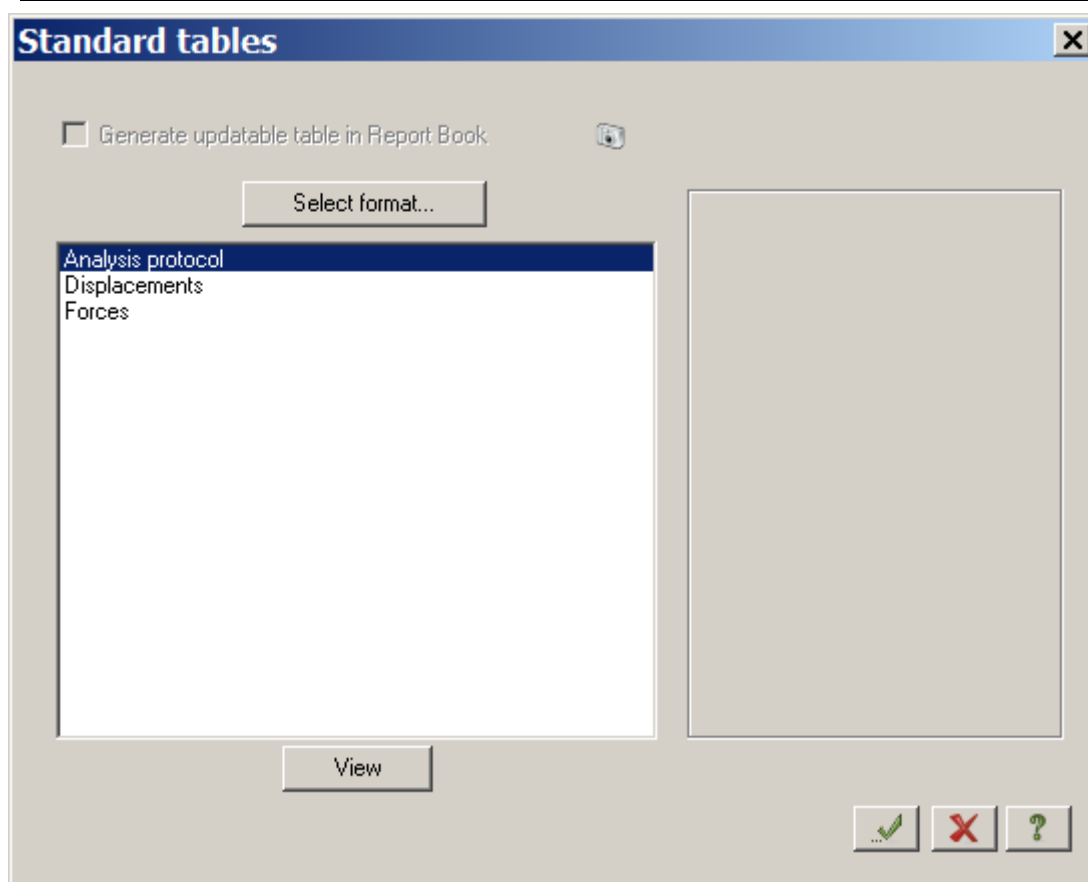


Figure 8.26 **Standard tables** dialog box