Example 5. Analysis of steel tower

In this lesson you will learn how to:
– generate design model of steel tower;
– define wind pulsation load;
– analyse load on fragment.

Description:
Model of the tower is presented in Fig.5.1.
Steel tower with height 16 m.
Sections of elements of the tower (see Fig.5.1):
elements 1 – hot-rolled seamless pipe (Труба бесшовная горячекатаная), shape 45x3.5;
elements 2 – hot-rolled seamless pipe (Труба бесшовная горячекатаная), shape 25x3.5;

Loads:
– load case 1 – dead weight; uniformly distributed load p = 0.25 t/m^2 applied to upper bars;
– load case 2 – ice load;
– load case 3 – wind static load;
– load case 4 – pulsation for wind static load.

Example 5   Tutorial on LIRA-SAPR version 2014

1 - pipe 45x3,5
2 - pipe 25x3,5
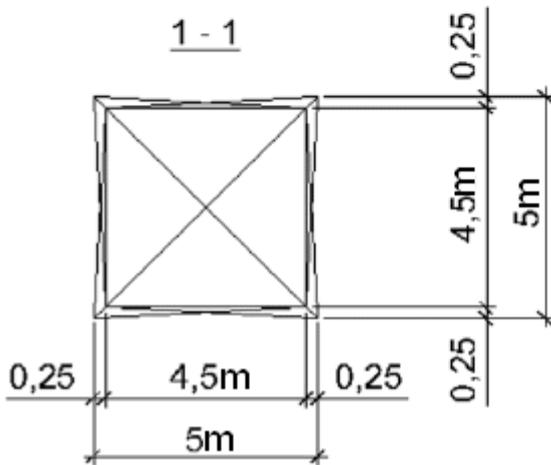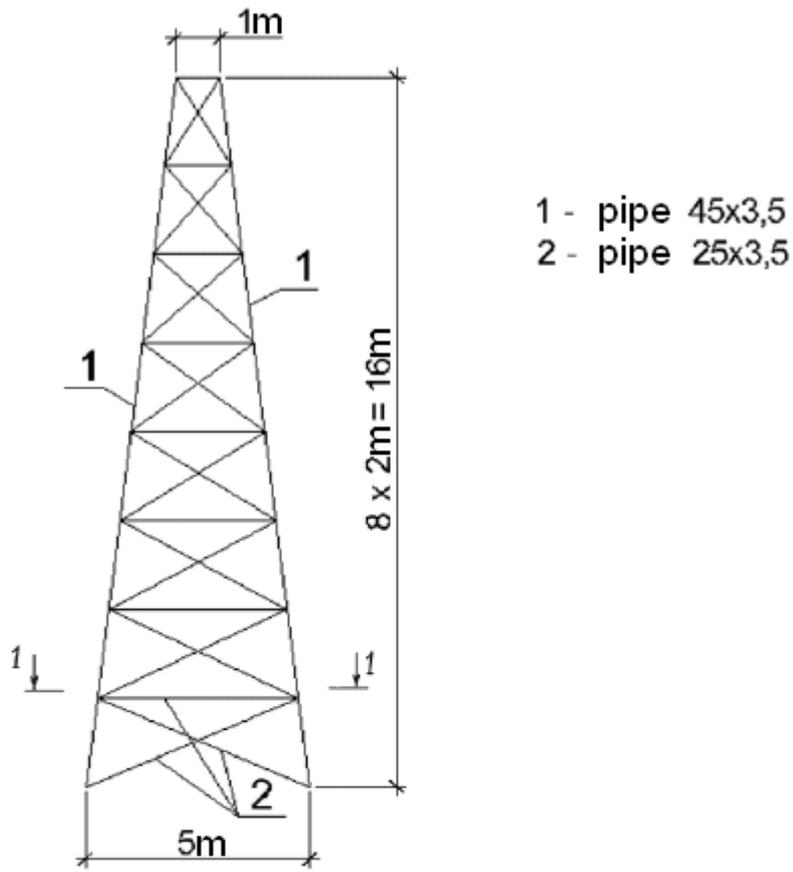
Figure 5.1   Model of tower

⇨ On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2014** and then click **LIRA-SAPR 2014**.

Step 1. Creating new problem

⇨ On the FILE menu, click **New** (button [] on the toolbar).

⇨ In the **Model type** dialog box (see Fig.5.2) specify the following data:

- problem name – **Example5** (problem code by default coincides with the problem name)

- model type – **4 – Three degrees of freedom per node** (translations X, Y, Z).

⇨ Click **OK** [].
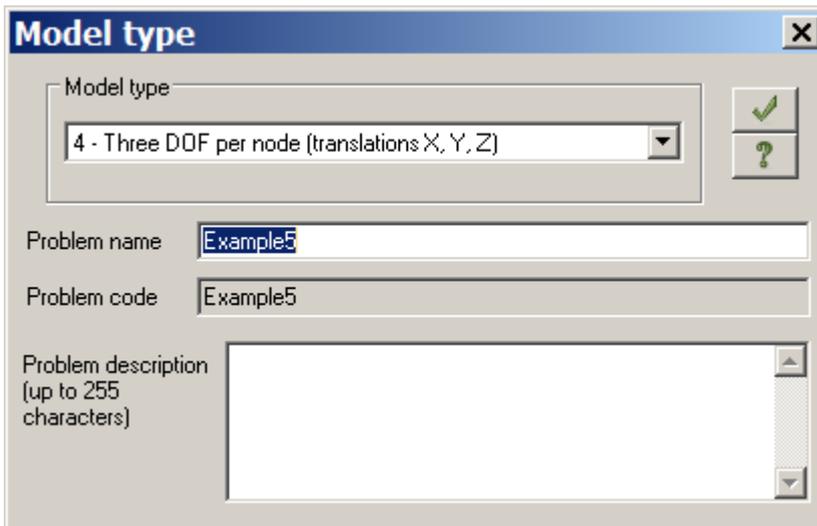
Figure 5.2   **Model type** dialog box

ⓘ   *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 4 (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 4 (Three DOF per node)** command . Then you should define only problem name.*

Step 2. Generating model geometry

To define nodes:

⇨ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add node** drop-down list and click **Add node by coordinates** [].

⇨ In the **Add node** dialog box (see Fig.5.3), define coordinates of the base node:

-     X   Y   Z
    0   0   0

⇨ Click **Apply** [].

⇨ Then define coordinates of the lower left node of the tower:

-     X   Y   Z
    -2.5   -2.5   0

⇨ Click **Apply**.

⇨ Then define coordinates of the upper left node of the tower:

Example 5   Tutorial on LIRA-SAPR version 2014

▪   X   Y   Z
   -0.5   -0.5   16

⇨   Click **Apply**.
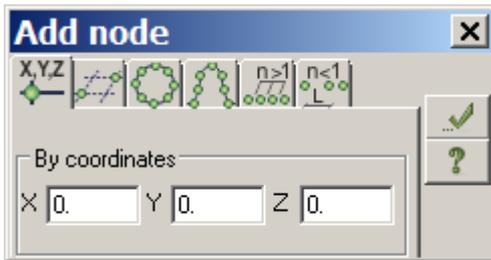
Figure 5.3   **Add node** dialog box

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 (see Fig.5.4), select the **Node numbers** check box on the **Nodes** tab.

⇨   Click **Redraw** .
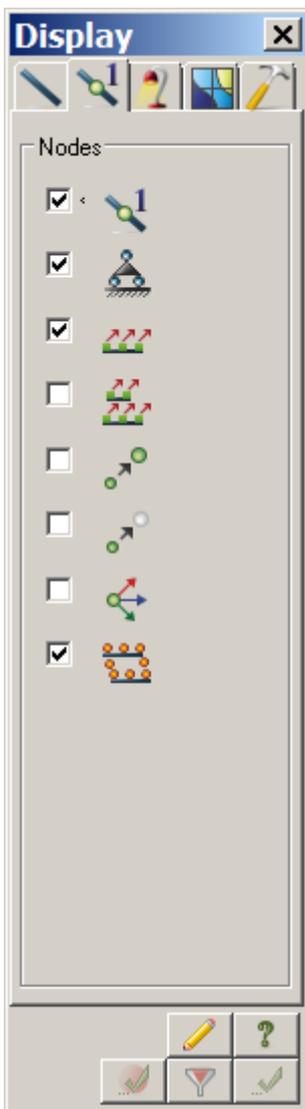

Figure 5.4   **Display** dialog box

                www.liraland.com

To add bar elements:

⇨   In the **Add node** dialog box, click **Divide into N equal parts** tab and define N = 8.

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

⇨   Select with the pointer nodes No.2 and 3 in sequence (the rubber-band line is automatically stretched between the nodes that you select).

To copy elements of the model:

⇨   On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .

⇨   Select with the pointer all elements of the model.

⇨   On the **Create and edit** ribbon tab, on the **Edit** panel, point to **Copy** drop-down list and click **Mirror copy** .

⇨   The **Copy objects** dialog box is presented with the **Mirror copy** tab open (see Fig.5.5).

⇨   To define the plane about which the copy will be made, under **Relative to the plane**, click YOZ-plane.

⇨   Select the **Specify base node** check box and specify with the pointer node No.1 on the model (the node will be coloured pink).

⇨   Click **Apply** .
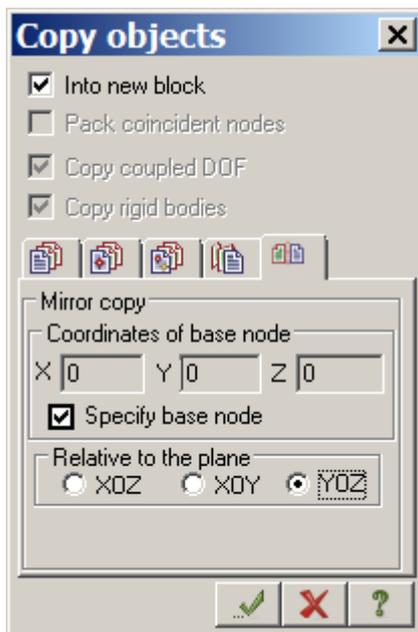
Figure 5.5   **Copy objects** dialog box

⇨   To unselect nodes and elements, on the **Select** toolbar, click **Unselect all** button .

To add bar elements of lacing:

⇨   On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click **Add bar** .

⇨   The **Add element** dialog box is presented with the **Add bar** tab open (see Fig.5.6).

Example 5   Tutorial on LIRA-SAPR version 2014
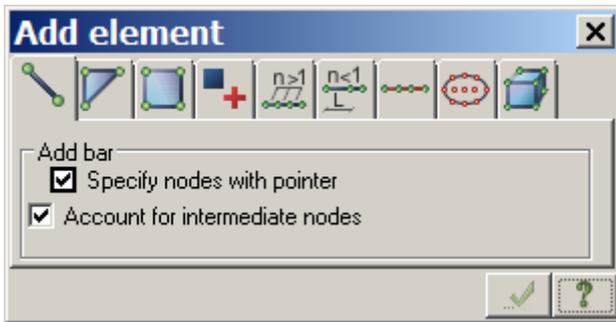
Figure 5.6   **Add element** dialog box

⇨   To add bar elements between nodes, specify with the pointer the following pairs of nodes in sequence: No.2 and No.13, No.4 and No.11, No.4 and No.13, No.4 and No.14, No.5 and No.13, No.5 and No.14, etc. up to the top of the tower (in this case the rubber-band line is automatically stretched between the nodes that you select).

The model that you will obtain is presented in Fig.5.7.
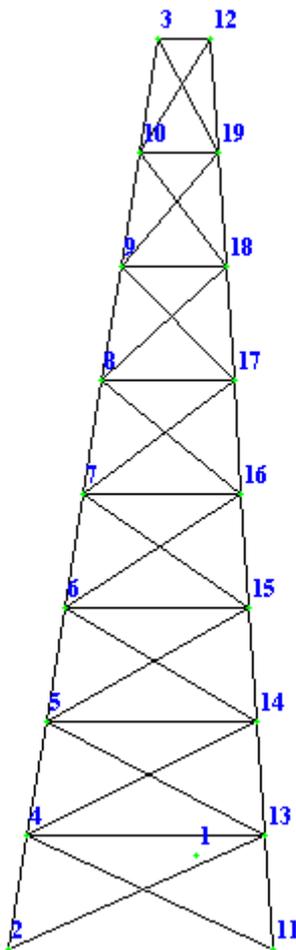

Figure 5.7   Model of fragment of a tower

Step 3. Defining boundary conditions

To select nodes:

⇨   On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button.

⇨   Select nodes No.2 and 11 with the pointer. The nodes will be coloured red.

To define boundary conditions:

➭   On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Restraints** [icon] .

➭   In the **Restraints on nodes** dialog box (see Fig.5.8), specify directions along which displacements of nodes are not allowed (X, Y, Z). To do this, select appropriate check boxes.

➭   Click **Apply** [icon] (the nodes will be coloured blue).

Figure 5.8   **Restraints on nodes** dialog box

Step 4. 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 [icon] .

➭   In the **Stiffness and materials** dialog box (see Fig.5.9a), click **Add**. The dialog box expands to display the library of stiffness parameters. In the **Add stiffness** dialog box (see Fig.5.9b), select the **Database of steel sections** tab (the second tab).

Example 5   Tutorial on LIRA-SAPR version 2014

a   b
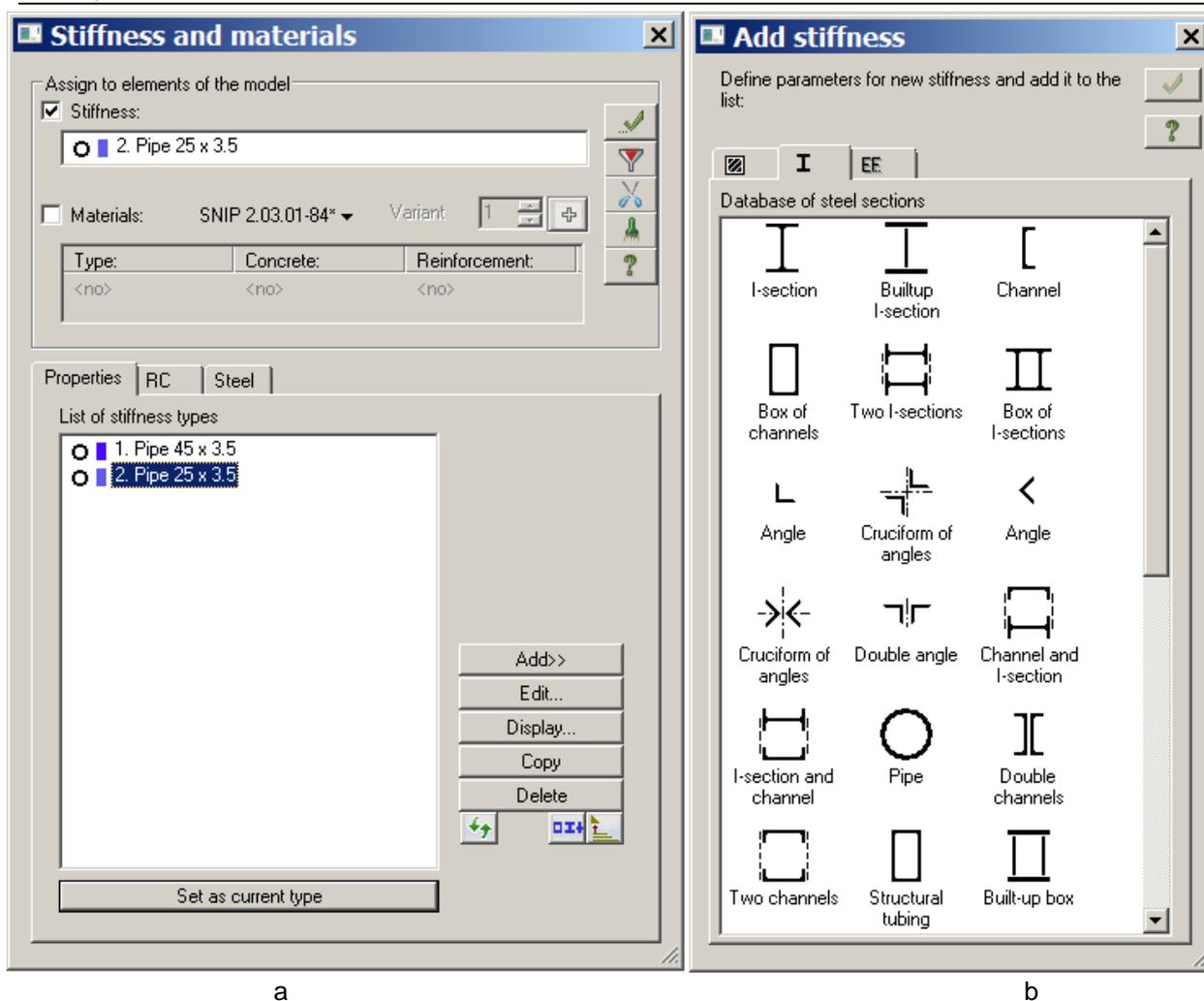
Figure 5.9   Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

⇨   Double-click the **Pipe** 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 (see Fig.5.10), specify the following parameters for pipe section (for main elements - element 1, see Fig.5.1):

▪   in the **Steel table** box, click Труба бесшовная горячекатаная ;

▪   in the **Shape** box, click 45 x 3.5.

⇨   Click **OK**.

⇨   In the **Add stiffness** dialog box (see Fig.5.9b), on the **Database of steel sections** (the second tab), double-click the **Pipe** icon once again.

⇨   In the **Steel cross-section** dialog box, specify the following parameters for pipe section (for elements of lacing - element 2, see Fig.5.1):

▪   in the **Steel table** box, click Труба бесшовная горячекатаная ;

▪   in the **Shape** box, click 25 x 3.5.

⇨   Click **OK**.

⇨   To hide the library of stiffness parameters, click **Add** in the **Stiffness and materials** dialog box.
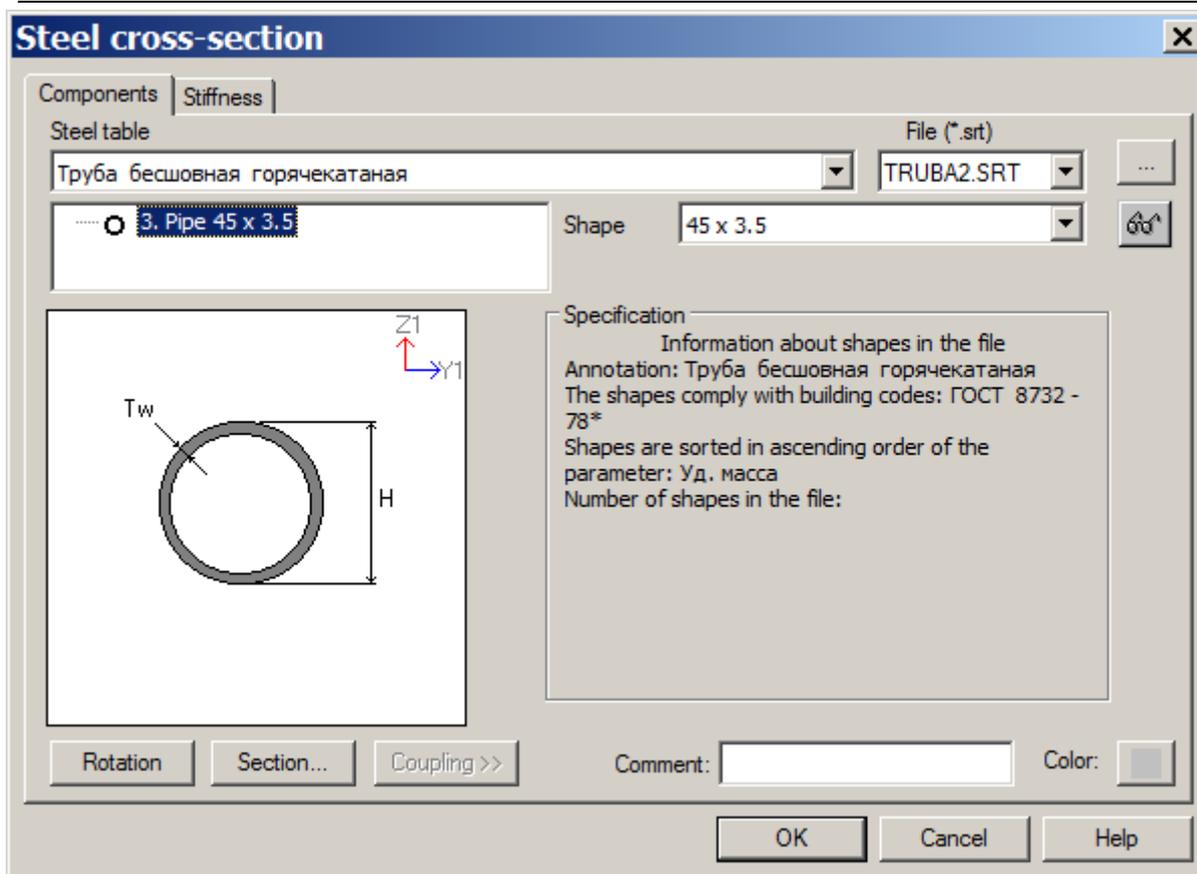
Figure 5.10   **Steel cross-section** dialog box

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 assign material properties to elements:

⇨   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.5.11).

⇨   Select **By FE numbers** check box and specify numbers of elements 17 – 40.

⇨   Click **Apply**.

Example 5   Tutorial on LIRA-SAPR version 2014
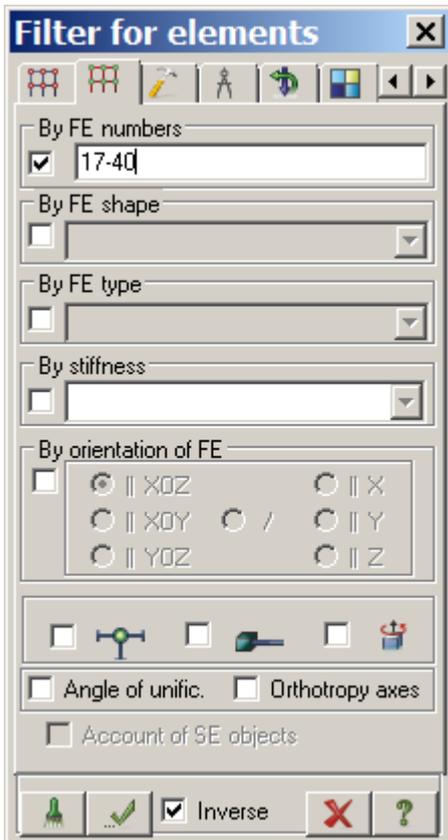
Figure 5.11   **Filter for elements** tab

⇨   In the **Stiffness and materials** dialog box, click **Apply** ⟨✓⟩ . The elements become unselected. It
    indicates that the current stiffness type is assigned to selected elements.

⇨   In the same dialog box, in the **List of stiffness types**, select type '1.Pipe 45 x 3.5' and 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.

⇨   Select elements of type 1 (see Fig.5.1) (elements No.1 – 16) with **PolyFilter** command as described
    above.

⇨   In the **Stiffness and materials** dialog box, click **Apply**.

⇨   In the **Stiffness and materials** dialog box, on the **Properties** tab (the first tab), select '2.Pipe 25 x 3.5' in
    the **List of stiffness types**.

⇨   Click **Set as current type**. This operation is necessary for further procedure (see Step 5).

Step 5. Editing the model

To copy fragment of the model:

⇨   On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button ⟨🔍⟩ .

⇨   On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button ⟨🔍⟩ .

⇨   Select with the pointer all nodes and elements of the model.

⇨   On the **Create and edit** ribbon tab, on the **Edit** panel, point to **Copy** drop-down list and click **Copy by
    rotation** ⟨📋⟩ .

⇨   The **Copy objects** dialog box is presented with the **Copy by rotation** tab open (see Fig.5.12). In this
    dialog box specify the following parameters:

▪   axis about which the fragment will be copied - Z-axis;

- angle of rotation Fi = 90 degrees;

- number of copies N = 3.

⇨   Click **Apply** [✓] .
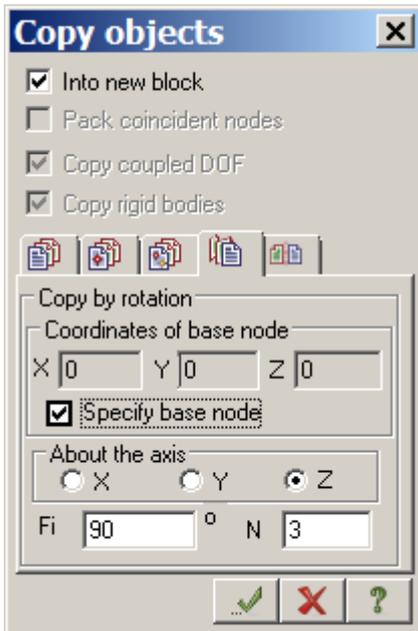
Figure 5.12   **Copy objects** dialog box

To pack the model:

⇨   On the **Create and edit** ribbon tab, on the **Edit** panel, click **Pack model** [▣] .

⇨   In the **Pack model** dialog box (see Fig.5.13), click **Apply** [✓] . It is necessary to pack the model in order to 'throw together' coincident nodes and elements and to eliminate (that is, to remove completely) deleted nodes and elements from design model.

Example 5   Tutorial on LIRA-SAPR version 2014
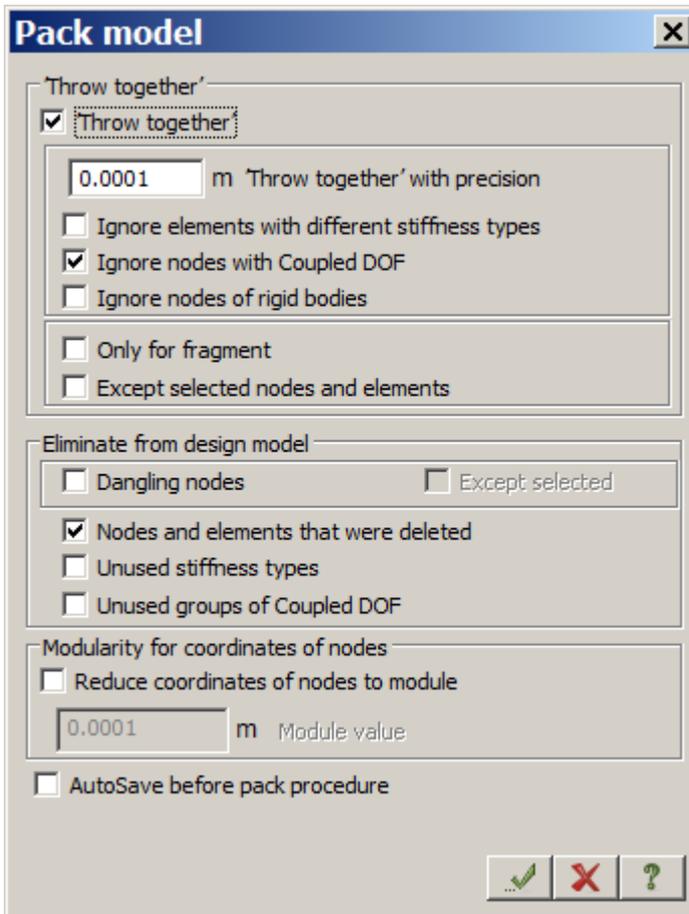
Figure 5.13 **Pack model** dialog box

⇨   To unselect elements, on the **Select** toolbar, click **Unselect all** button .

To hide numbers of elements on design model:

⇨   On the **Select** toolbar, click **Flags of drawing** button .

⇨   In the **Display** dialog box, clear the **Element numbers** check box on the **Elements** tab.

⇨   Click **Redraw** .

To add bar elements of lacing:

⇨   On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **PolyFilter** .

⇨   In the **PolyFilter** dialog box, select the **Section and cut off** tab (see Fig.5.14).

⇨   Under **Cutting plane**, click **Arbitrary**.

⇨   Specify with the pointer three nodes that define diagonal of the tower (nodes No.2, 17, 21).

⇨   Click **Apply** in the **PolyFilter** dialog box on the **Section and cut off** tab.
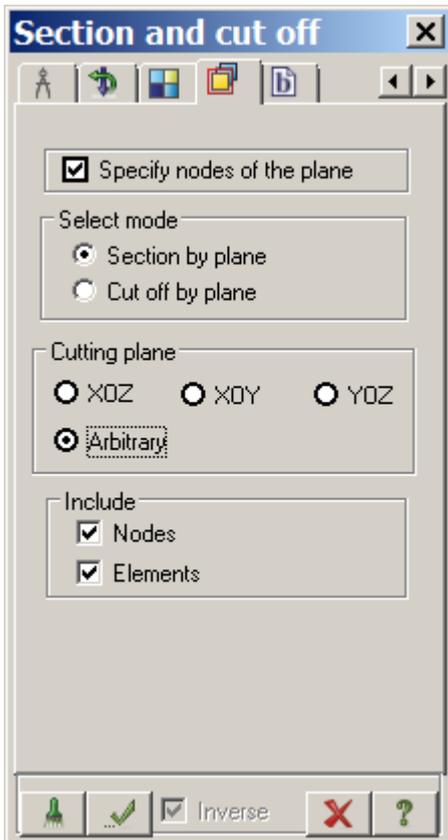
Figure 5.14   **PolyFilter** dialog box (**Section and cut off** tab)

⇨   To present on the screen only selected nodes and elements of the model, on the **Select** toolbar, click
     Fragmentation ▣ .

⇨   To present design model in projection on the XOZ-plane, on the **Projection** toolbar, click **Projection on**
     XOZ-plane ▣ .

⇨   On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click
     Add bar ▨ .

⇨   The **Add element** dialog box is presented with the **Add bar** tab open (see Fig.5.6).

⇨   To add bar elements between nodes, specify with the pointer the following pairs of nodes in sequence:
     No.4 and No.21, No.5 and No.24, No.6 and No.25, etc. up to the top of the tower.

Example 5   Tutorial on LIRA-SAPR version 2014
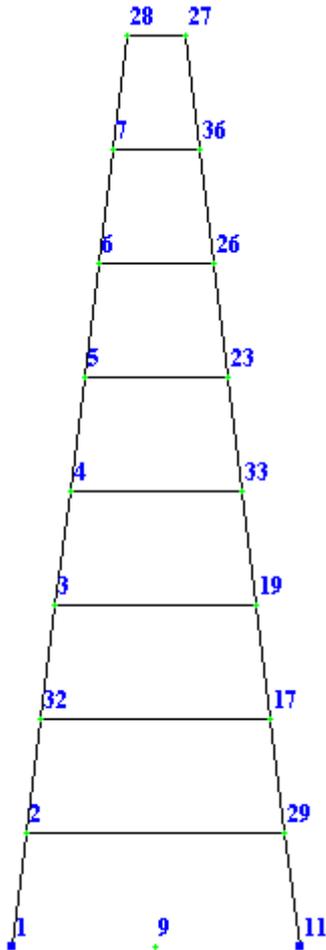
Figure 5.15   Model of the tower in projection on the XOZ-plane

⇨ To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection** .

⇨ To restore design model in initial view after fragmentation, on the **Select** toolbar, click **Restore model** .

⇨ In the **PolyFilter** dialog box (see **Flags of drawing** command), on the **Section and cut off** tab, select the **Specify nodes of the plane** check box.

⇨ Specify with the pointer three nodes that define diagonal of the tower (nodes No.10, 11, 32).

⇨ Click **Apply** .

⇨ To present on the screen only selected nodes and elements of the model, on the **Select** toolbar, click **Fragmentation** .

⇨ To present design model in projection on the XOZ-plane, on the **Projection** toolbar, click **Projection on XOZ-plane** .

⇨ In the **Add element** dialog box, select the **Specify nodes with pointer** check box.

⇨ Specify with the pointer the following pairs of nodes in sequence: No.22 and No.11, No.33 and No.23, etc. up to the top of the tower (as described above).

⇨ To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection** .

⇨ On the **Select** toolbar, click **Restore model** .

⇨ On the **Select** toolbar, click **Flags of drawing** button .

⇨ In the **Display** dialog box, clear the **Node numbers** check box on the **Nodes** tab.

⇨   Click **Redraw** ✏️ .

The model of tower is presented in Fig.5.16.
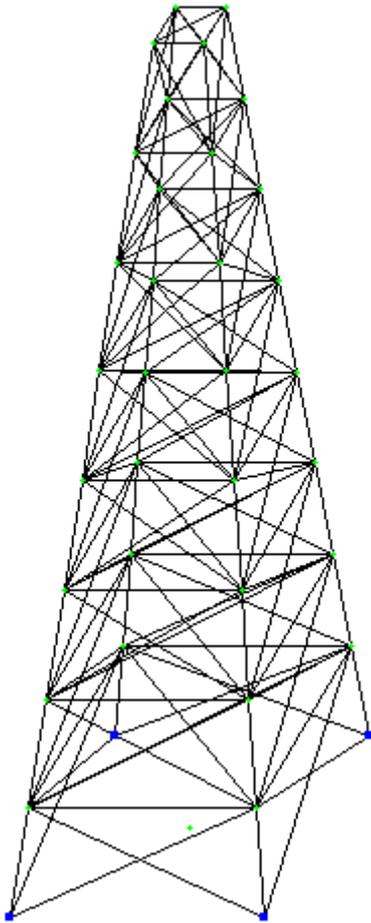
Figure 5.16   Design model of tower

ⓘ   *In the **Stiffness of elements** dialog box, the stiffness type **2. Pipe 25 x 3.5** is assigned as current one. That's why this type of stiffness will be assigned to all bar elements.*

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 – **Example5**;

▪   location where you want to save this file (**Data** folder is displayed by default).

⇨   Click **Save**.

Step 6. Applying loads

To create load case No.1:

⇨   To define load from dead weight, on the **Create and edit** ribbon tab, select the **Loads** panel and click

Add dead weight 🗑️ .

Example 5   Tutorial on LIRA-SAPR version 2014

⇨ In the **Add dead weight** dialog box (see Fig.5.17), 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** ![checkmark] (uniformly distributed load equal to unit weight of elements is automatically applied to all elements of the structure).
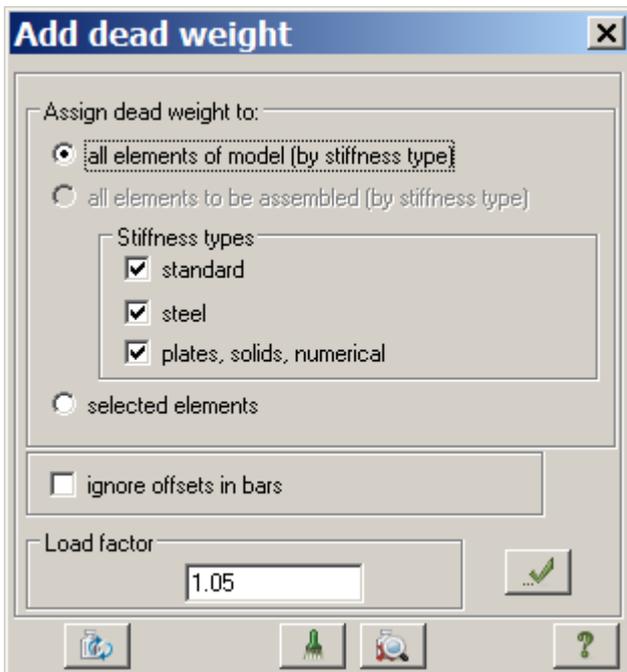

Figure 5.17   **Add dead weight** dialog box

⇨ On the **Select** toolbar, click **Select horizontal bars** button ![icon] .

⇨ Select horizontal elements of the upper platform of the tower.

⇨ On the **Create and edit** ribbon tab, select the **Loads** panel, then select **Load on bars** command ![icon] from the **Loads on nodes and elements** drop-down list.

⇨ In the **Define loads** dialog box (see Fig.5.18), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).
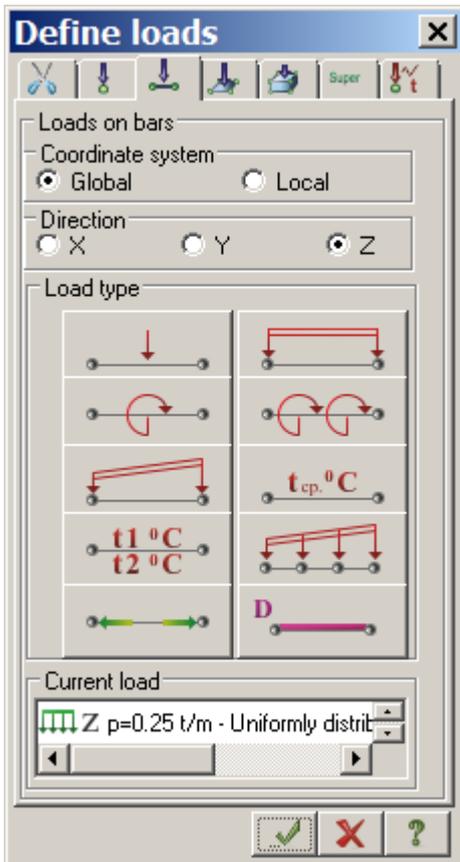
Figure 5.18   **Define loads** dialog box

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

⇨   In the **Load parameters** dialog box specify P = 0.25 t/m (see Fig.5.19).

⇨   Click **OK** .
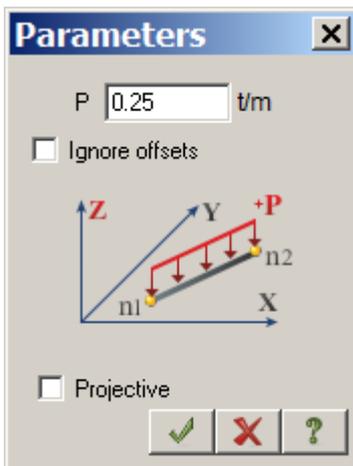
Figure 5.19   **Load parameters** dialog box

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

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, point to **Select elements** drop-down list and click **Select elements** button .

Example 5   Tutorial on LIRA-SAPR version 2014

⇨   Select all elements of the tower with the pointer.

⇨   In the **Define loads** dialog box, when uniformly distributed load P = 0.25 t/m is defined as current load and direction is defined along the global Z-axis, click **Apply**.

To create load case No.3:

⇨   To change the number of the current load case, click the **Next load case** button located on the Status bar or on the toolbar.

ⓘ
    Static wind pressure may be specified as the sum of windward and leeward side pressures because every level of tower is considered as rigid body.

⇨   On the **Projection** toolbar, click **Projection on XOZ-plane** .

⇨   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).

⇨   Select **By stiffness** check box and specify the line '1.Pipe 45x3.5'.

⇨   Select projection of five lower elements of right edge of tower with 'selection window' as presented in Fig.5.20.

ⓘ
   *Uniformly distributed wind load is applied up to 10m.*
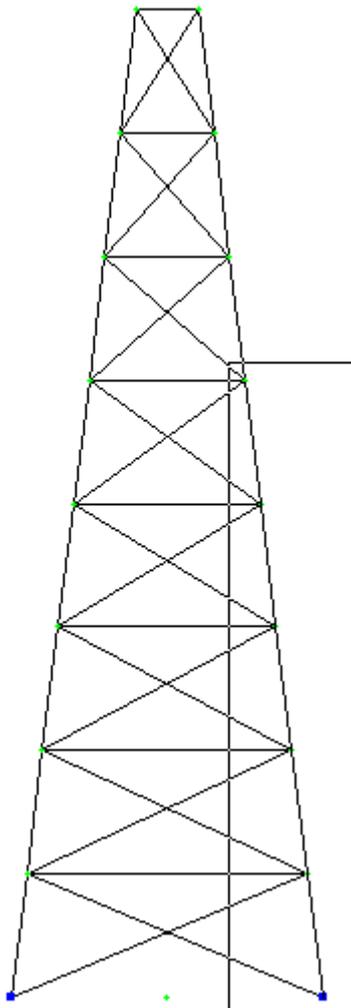
Figure 5.20   Specifying elements with 'selection window'

⇨   To change direction of load, in the **Define loads** dialog box, specify direction along the **X**-axis.

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

⇨   In the **Load parameters** dialog box specify P = 0.1 t/m.

⇨   Click **OK** .

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

⇨   Select projection of three upper elements of the right edge of the tower with 'selection window' in a similar way as shown in Figure 5.21.

⇨   In the **Define loads** dialog box, click **Trapezoidal load on group of bars** button .
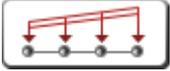
⇨   In the **Non-uniformly distributed load** dialog box (see Fig.5.21), specify P1 = 0.1 t/m, P2 = 0.12 t/m and direction along which the load is changed (Z-axis).

⇨   Click **OK**.
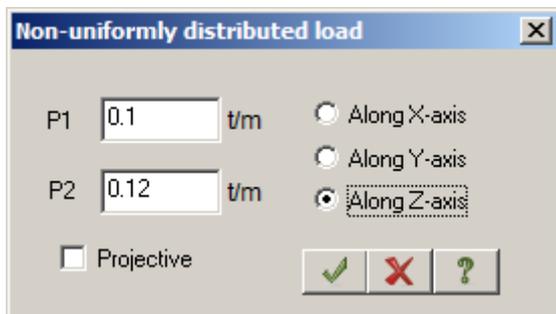
Figure 5.21   **Non-uniformly distributed load** dialog box

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

⇨   To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection** .

⇨   Close the **PolyFilter (Filter for elements)** dialog box.

To define detailed information about load cases:

⇨   On the **Create and edit** ribbon tab, on the **Loads** panel, click **Edit load cases** . The **Edit load cases** dialog box is displayed on the screen (see Fig.5.22).

⇨   For load case 1 – in the **Edit selected load case** area, in the **Type** box, select **Dead** and click **Apply** .

⇨   For load case 2 – in the **Edit selected load case** area, in the **Type** box, select **Short-term** and click **Apply** .

⇨   For load case 3 – in the **Edit selected load case** area, in the **Type** box, select **Inactive** (wind static for pulsation) and click **Apply** .

⇨   To add the fourth load case, in the **List of load cases** area, click **Add load case (to the end)** .

⇨   For load case 4 – in the **Edit selected load case** area, in the **Type** box, select **Instant** and click **Apply** .

Example 5   Tutorial on LIRA-SAPR version 2014
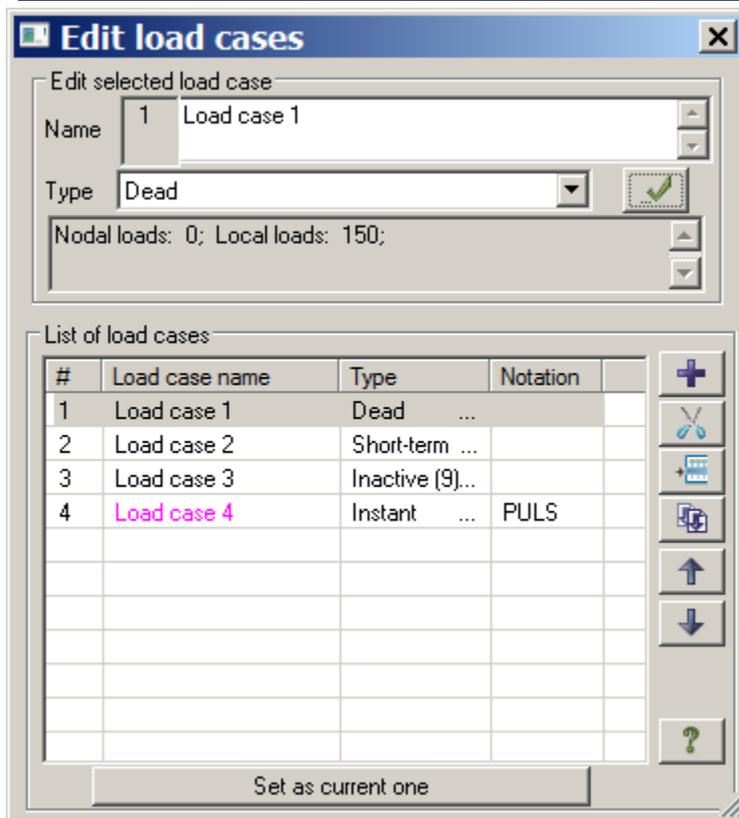
Figure 5.22   **Edit load cases** dialog box

Defining parameters for analysis of tower on wind pulsation

Step 7. Creating dynamic load cases from the static ones

⇨   On the **Analysis** ribbon tab, on the **Dynamics** panel, click **Account of static load cases** .

⇨   In the **Create dynamic load cases from the static ones** dialog box (see Fig.5.23), under **Generate mass matrix according to**, click **Load case (code 1)** and to create the first line of the summary table, specify the following data:

  ▪ dynamic load case No. – 4;

  ▪ No. of corresponding static load case – 1;

  ▪ conversion factor – 1.

⇨   Click **Add**.

⇨   To create the second line of the summary table, specify the following data:

  ▪ dynamic load case No. – 4;

  ▪ No. of corresponding static load case – 2;

 ▪ conversion factor – 0.9.

⇨   Click **Add** and then click **OK**.
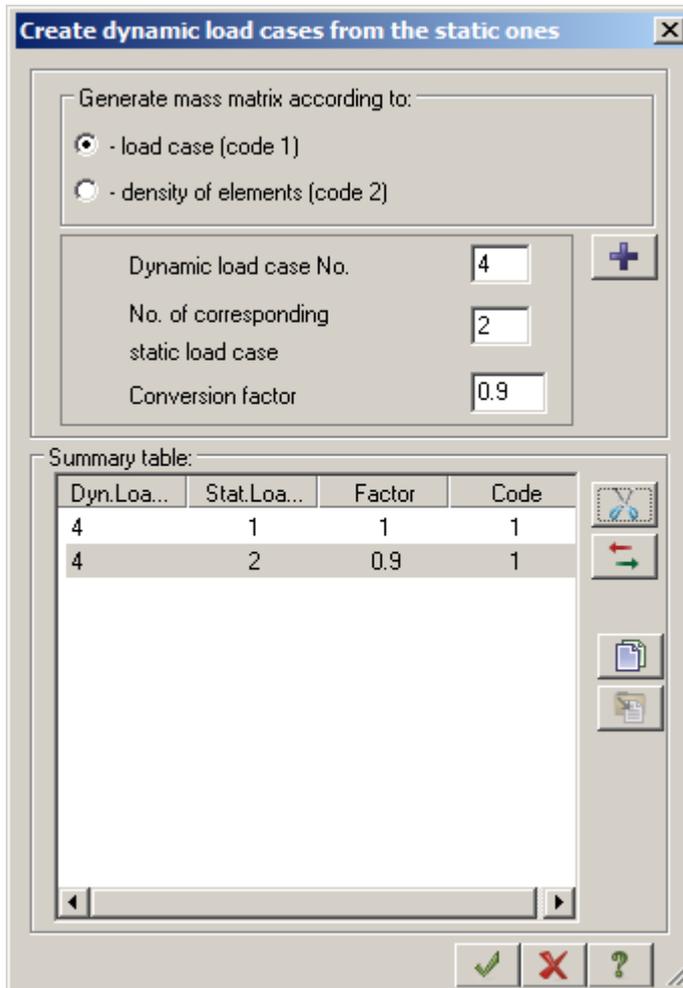
Figure 5.23   **Create dynamic load cases from the static ones** dialog box

Step 8. Generating table of dynamic load cases

⇨   On the **Analysis** ribbon tab, select the **Dynamics** panel and click **Table of dynamic load cases** button .

⇨   In the **Table of dynamic load cases** dialog box (see Fig.5.24), define the following data:

- load case No. – 4;

- dynamic load type – **Pulsation (21)**;

- number of analysed mode shapes – 8;

- number of corresponding static load case – 3;

- in the **Mass matrix** field, click **Diagonal**.

⇨   Click **Parameters**.

Example 5   Tutorial on LIRA-SAPR version 2014
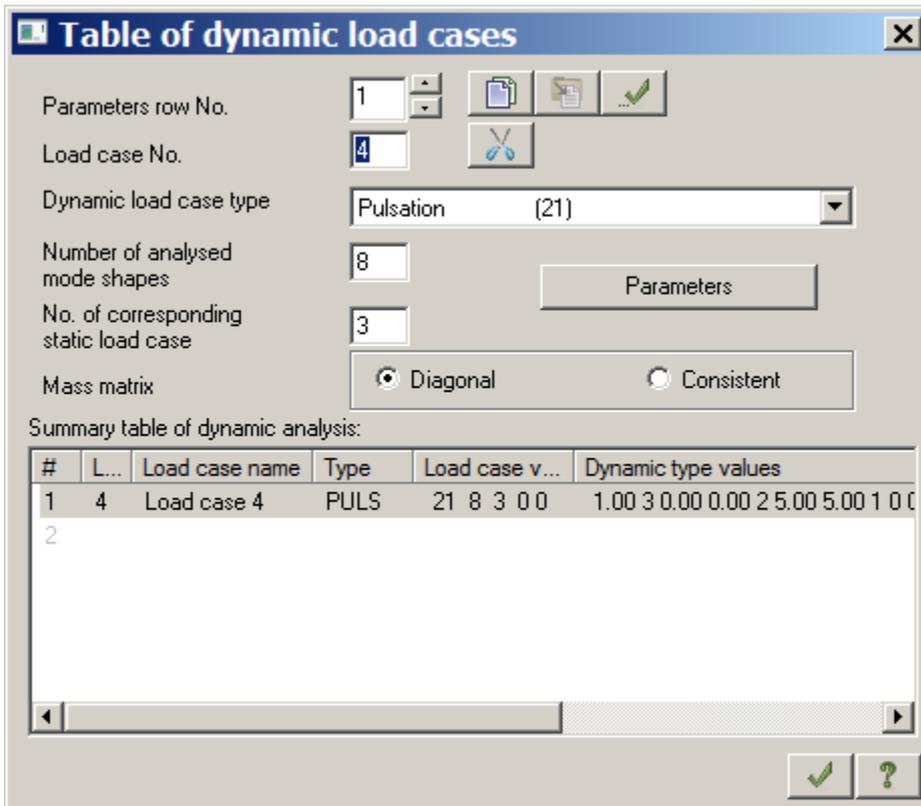
Figure 5.24   **Table of dynamic load cases** dialog box

⇨   In the **Wind analysis parameters with pulsation** dialog box (see Fig.5.25), when SNIP 2.01.07-85* is selected, define the following data:

▪   in the **Wind region of the site** box select **Zone 2**;

▪   length of structure along X-axis – 5 m;

▪   length of structure along Y-axis – 5 m;

▪   logarithmic decrement of vibration – DCR = 0.15 (steel structures);

▪   other parameters remain by default.

⇨   Click **OK** .

⇨   In the **Table of dynamic load cases** dialog box, click **OK**.
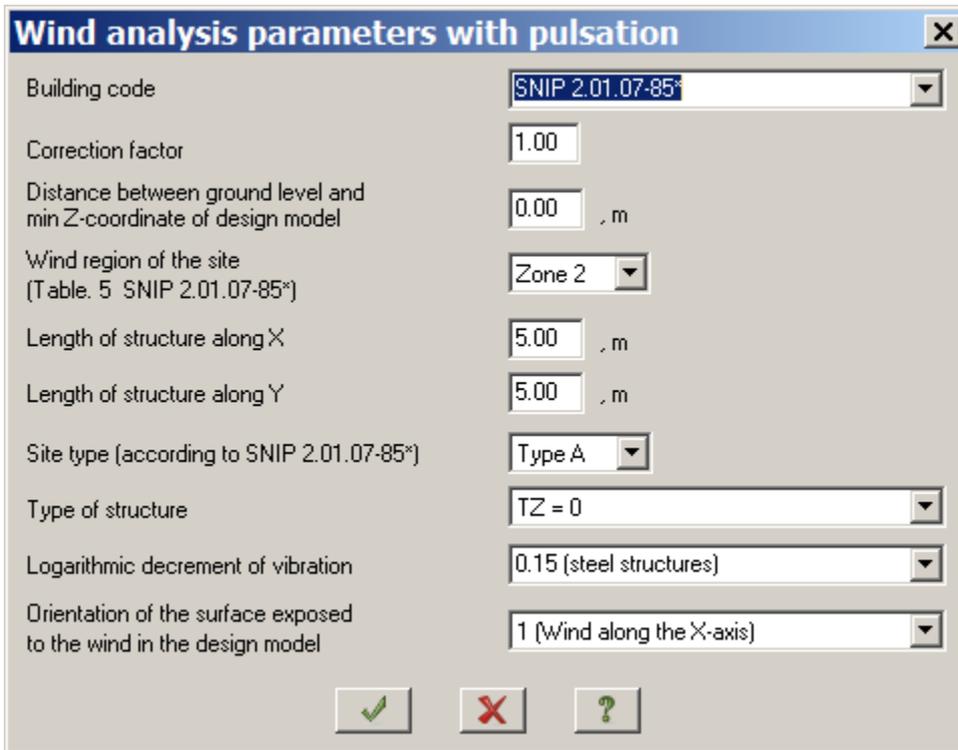
Figure 5.25   **Wind analysis parameters with pulsation** dialog box

Step 9. Generating DCF table

⇨   On the **Analysis** ribbon tab, select the **DCF** panel and click **DCF table** button ⊞ .

ⓘ   *As the type of load cases was defined in the **Edit load cases** dialog box (see Fig.5.22), the DCF table is generated automatically with parameters accepted by default for every load case. Now you have to modify parameters for the first, second and the fourth load cases.*

⇨   In the **Design combinations of forces** dialog box (see Fig.5.26), select building code **SNIP 2.01.07-85*** and specify the following data:

-   for Load case 1 – in the **Load factor** box specify **1.05** and then click **Apply** ✓ ;

-   for Load case 2 – in the **Load factor** box specify **1.3** and then click **Apply** ✓ ;

-   for Load case 4 – select the **Account of sign variability** box and then click **Apply** ✓ .

⇨   Click **OK** ✓ .

Example 5   Tutorial on LIRA-SAPR version 2014
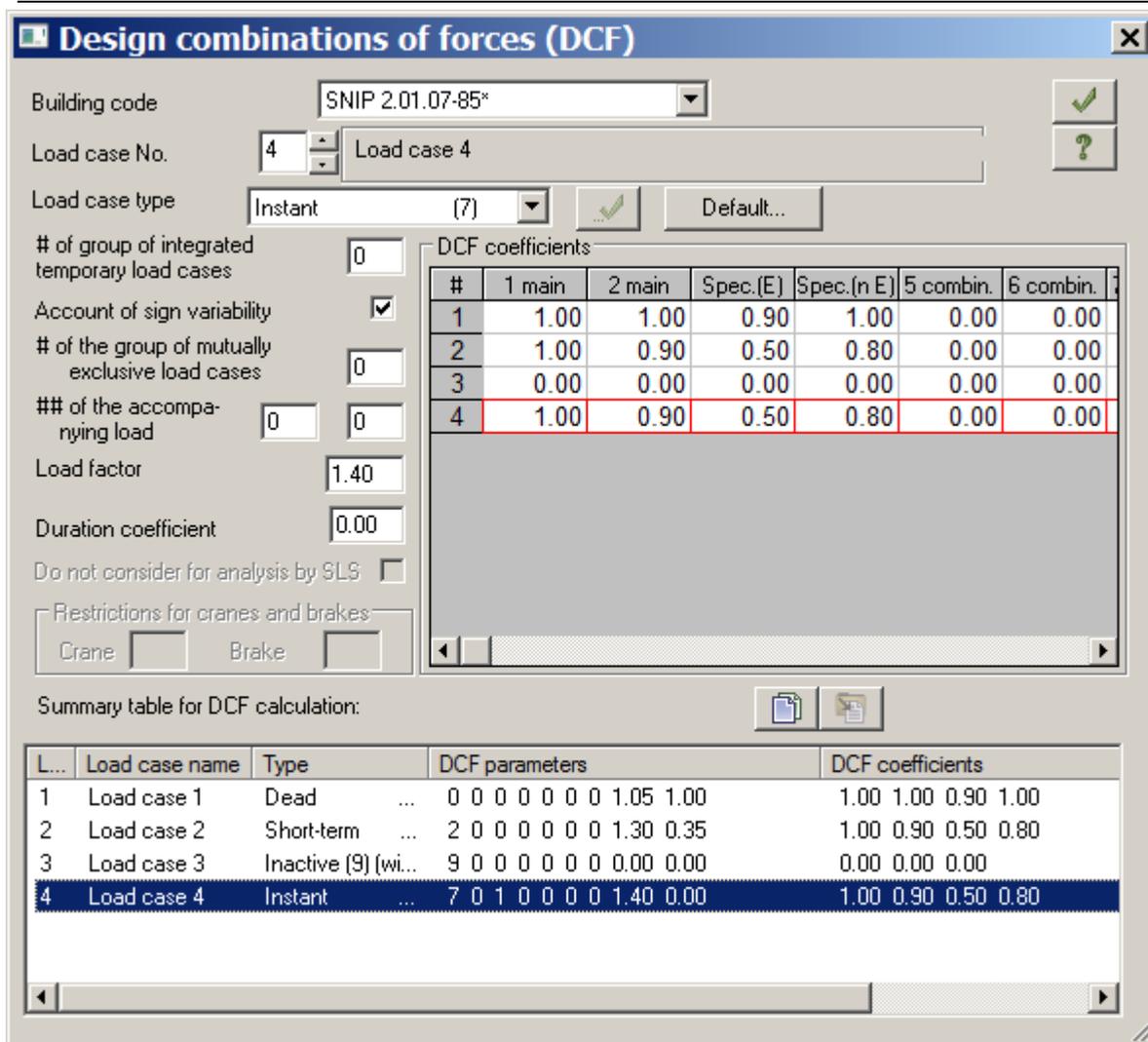
Figure 5.26   **Design combinations of forces** dialog box

Step 10. Static analysis of tower

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

Step 11. Review and evaluation of static analysis results

ⓘ   *When analysis procedure is complete, to review and evaluate results for static and dynamic analyses, select the **Results** ribbon tab.*

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

To hide presentation of loads on design model:

⇨   On the **Select** toolbar, click **Flags of drawing** button . In the **Display** dialog box, clear the **Loads** check box on the **General** tab.

⇨   Click **Redraw** .

To present diagrams of internal forces:

⇨ To display diagram **N**, on the **Results** tab, select **Forces in bars** panel and click **Axial force diagrams (N)** button ▣ .

⇨ 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 change number of active load case:

⇨ On the status bar (displayed at the bottom of the screen), in the **Load case No.** list, select No. **2** and click **Apply** ▣ .

To present mode shapes of the structure:

⇨ To change the number of active load case, on the Status bar (displayed at the bottom of the screen), in the **Load case No.** list, select No. **4** and click **Apply** ▣ .

⇨ To display the model with account of nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button ▣ to make this command not active.

⇨ To hide mosaic plots **N**, on the **Results** tab, select **Forces in bars** panel and click **Mosaic plot N** command ▣ to make this command not active.

⇨ To display the first mode shape, on the **Results** ribbon tab, on the **Deformations** panel, select the **Mode shapes** command ▣ in the **Stress strain state** drop-down list.

⇨ To display the 2nd mode shape of the 4th load case, on the Status bar (displayed at the bottom of the screen), in the **Mode shape No. (component, period)** list, change number of mode shape for **2** and click **Apply** ▣ .

To animate the 2nd mode shape:

⇨ To switch to the mode of 3D visualization, either select **3D model** command on the **Application menu** or click the **3D model** button ▣ on the Quick Access Toolbar.

⇨ To animate the 2nd mode shape of the 4th load case, on the **3D view** ribbon tab, on the **Animation** panel, click **Animate mode shape** button ▣ .

⇨ In the **Animate mode shape** dialog box (see Fig.5.27), click **Play** button ▣ .

⇨ To close the **Animate mode shape** dialog box, click **Close**.
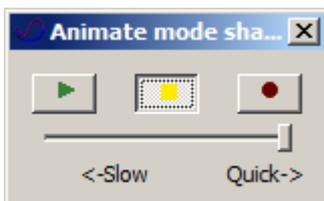
Figure 5.27   **Animate mode shape** dialog box

⇨ To return to the mode of analysis results visualization, on the **3D view** ribbon tab, on the **Back** panel, click **Finite element model** button ▣ .

To present numbers of elements on the screen:

Example 5   Tutorial on LIRA-SAPR version 2014

⇨   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 **Element numbers** check box on the **Elements** tab.

⇨   Click **Redraw** .

To generate and review tables of analysis results:

⇨   On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .

⇨   Select element No. 1 (support element of tower) with the pointer.

⇨   To present table with design combinations of forces in selected element of the model, 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.5.28), select **DCF, design values** in the list.

⇨   To generate tables in HTML format, select appropriate option.

⇨   Click **Apply**. Table 5.1 will be displayed in MS Internet Explorer.
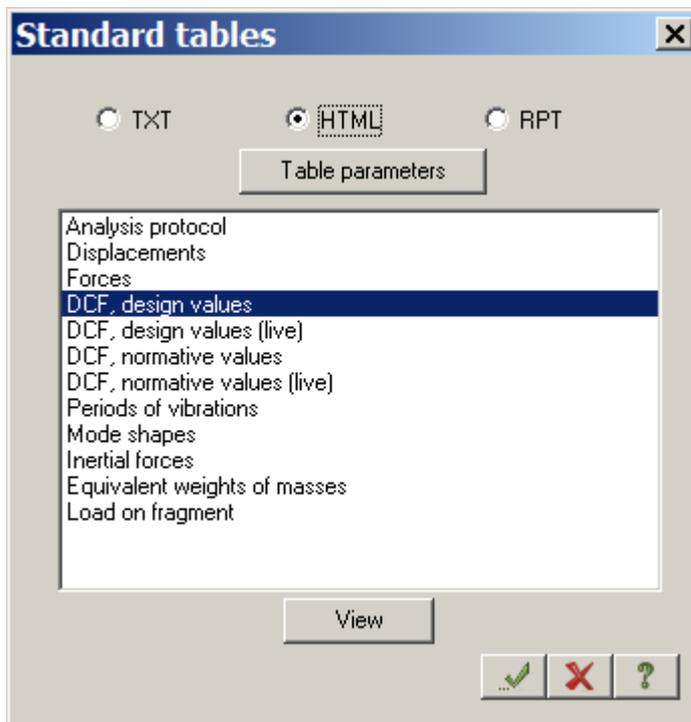
Figure 5.28   **Standard tables** dialog box

Table 5.1. Design combinations

| ELM	SN	CRT	CN	CE	G	N	MK	MY	QZ	MZ	QY	LOAD CASES.
1	1	2	1		A1	-26.673	0	0	.04485	0	0	1,2,
		1	2		B1	4.5777	0	0	.00066	0	0	1,-4,
		2	2		B1	-29.295	0	0	.04043	0	0	1,2,4,
		2	1		A2	-.64635	0	0	.00066	0	0	1,
		2	1		B2	-26.673	0	0	.04485	0	0	1,2,
		1	2		C2	4.5777	0	0	.00066	0	0	1,-4,
		2	2		C2	-29.295	0	0	.04043	0	0	1,2,4,
1	2	2	1		A1	-26.166	0	0	-.04485	0	0	1,2,
		1	2		B1	4.5852	0	0	-.00066	0	0	1,-4,
		2	2		B1	-28.837	0	0	-.04043	0	0	1,2,4,

ⓘ   *The following notation is accepted in the table:*
*column 1 – **ELM** – number of element on design model;*
*column 2 – **SN** – number of section in bar element;*
*column 3 – **CRT** – criterion for DCF selection;*
*column 4 – **CN** – number of column of DCF coefficients in the DCF table;*
*column 5 – **CE** – index of crane and earthquake loads in case these loads are included in DCF;*
*column 6 – **A** and **B** indexes (A1, B1, C1, D1, A2, B2, C2, D2) differentiate DCF groups according to duration of loads that are included in the load case.*
*Internal groups for ULS – groups A1, B1, C1, D1 – are generated according to criteria calculated by total design values of forces. Index **A1** indicates the DCF that consists of load cases with duration. Index **B1** shows the DCF that consists of all load cases regardless of duration except for earthquake and other specific load cases. Index **C1** indicates that DCF includes group B1 plus earthquake load case. Index **D1** indicates that DCF includes B1 group plus specific (no earthquake) load case.*
Internal groups for SLS are generated in two ways:
- groups A2, B2 – from criteria calculated according to long-live part of normative (characteristic) forces.
- groups C2, D2 – from on total normative forces.
Group A2 – contains only dead and live load cases.
Group B2 – contains dead, live and short-term load cases (except instant).
Group C2 – contains all defined load cases regardless of their duration, except earthquake load case and other specific ones.
Group D2 – contains group C2 and earthquake load case.

⇨   To close the table, on the FILE menu, click **Exit**.

⇨   To close the **Standard tables** dialog box, click **Close**.

⇨   To switch to results of static analysis, on the **Results** ribbon tab, on the **Deformations** panel, select the

Deformed shape command ⬚ in the **Stress strain state** drop-down list.

Step 12. Analysis of load on fragment

To present numbers of nodes on the screen:

⇨   On the **Select** toolbar, click **Flags of drawing** button ⬚ .

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

⇨   Click **Redraw** ⬚ .

To analyse load on fragment:

Example 5   Tutorial on LIRA-SAPR version 2014

ⓘ   *Load on fragment is calculated according to the following data:*
      - numbers of nodes where this load should be calculated;
      - numbers of elements that transfer the load on these nodes;
      - angles of rotation of nodes about the Z-axis of global coordinate system.

⇨   On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button 🔍 .

⇨   Select elements No. 2 and 10 with the pointer.

⇨   To select elements that transfer load to selected nodes, on the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements adjacent to selected nodes** button 🔍 .

⇨   On the **More results** ribbon tab, on the **Fragment** panel, click **Calculate load on fragment** button 🔣 .

⇨   In the **Analysis of loads on fragment** dialog box (see Fig.5.29), specify the following parameters:

   ▪ in the **List of elements** box, click **Refresh** in order to input numbers of selected elements to appropriate field;

   ▪ to create new group of adjacent nodes, under **Adjacent nodes for fragment**, click **Create**;

   ▪ in the **List of elements** box, click **Refresh** in order to input numbers of selected elements to appropriate field.

⇨   Click **Analyse**.

Figure 5.29   **Analysis of loads on fragment** dialog box

ⓘ   *It is also possible to define data for analysis of load on fragment when the model generation is complete, before analysis of problem. In this case the procedure for selecting nodes and elements remains the same and to activate the **Analysis of loads on fragment** dialog box, on the **Analysis** ribbon tab, on the **More calculations** panel, click **Input data - Load on fragment** button 🖵 . To input data, in the dialog box click **Apply**.*

To generate and review table of analysis results for load on fragment:

➪   To present table with loads on fragment, on the **Results** ribbon tab, select **Tables** panel and click

Standard tables in the **Documents** drop-down list.

➪   In the **Standard tables** dialog box, select **Load on fragment** in the list.

➪   Click **Apply** .

➪   To close the table, on the FILE menu, click **Exit**.

➪   To close the **Standard tables** dialog box, click **Close**.

To edit flags of drawing:

➪   On the **Select** toolbar, click **Flags of drawing** button .

➪   In the **Display** dialog box, clear the **Node numbers** check box on the **Nodes** tab.

➪   Clear the **Element numbers** check box on the **Elements** tab.

➪   Select the **Load magnitudes** check box on the **General** tab.

➪   Click **Redraw** .

To display values of load on fragment at nodes of design model:

➪   On the status bar (displayed at the bottom of the screen), in the **Load case No.** list, select No. **1** and

click **Apply** .

➪   To display values of forces at nodes of the fragment along the Z-axis, on the **More results** ribbon tab, on

the **Fragment** panel, click **Load along Z** button ΣP_z .

➪   To display values of forces at nodes of the fragment along the X-axis, on the **More results** ribbon tab, on

the **Fragment** panel, click **Load along X** button ΣP_x .