

Example 3. Analysis of frame for industrial building

In this lesson you will learn how to:

- perform dynamic analysis of 2D frame;
- perform stability analysis of the structure;
- generate DCL table;
- determine and check steel sections of elements of the frame.

Description:

Model of the frame and its boundary conditions are shown in Fig.3.1.

Sections of elements:

- external columns – box of channel sections No. 24;
- internal columns – channel section No. 24;
- beam of floor slab – I-section No. 36;
- top chord of a truss – two angle sections 120 x 120 x 10;
- bottom chord of a truss – two angle sections 100 x 100 x 10;
- vertical and diagonal elements of a truss – two angle sections 75 x 75 x 6.

Loads:

- load case 1 – dead weight of elements of the model,
- load case 2 – live load,
- load case 3 – wind load,
- load case 4 – harmonic dynamic load,
- load case 5 – earthquake load.

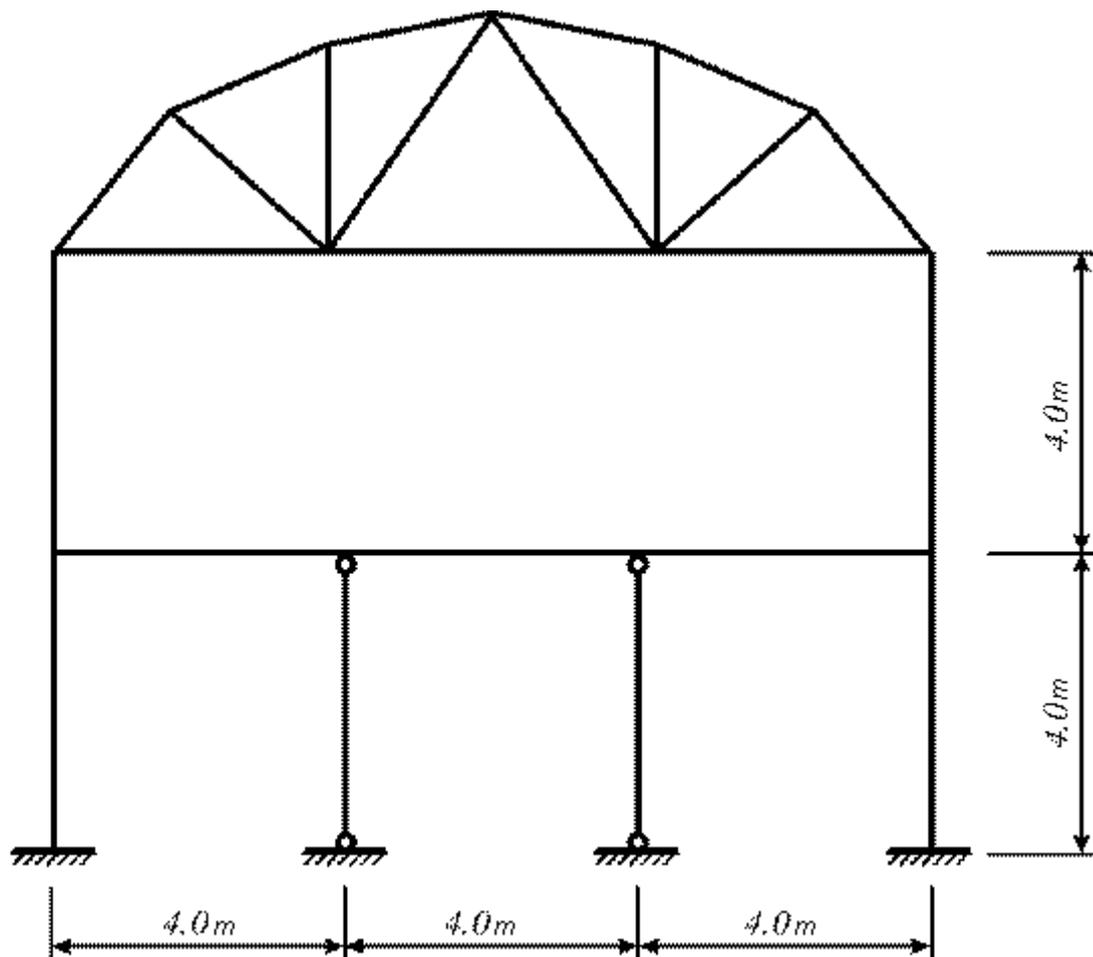


Figure 3.1 Design model

- ⇒ 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.3.2) specify the following data:
 - problem name – **Example3** (problem code by default coincides with the problem name)
 - model type – **2 – Three degrees of freedom per node** (translations X, Z and rotation Uy) X0Z.
- ⇒ Click **OK** .

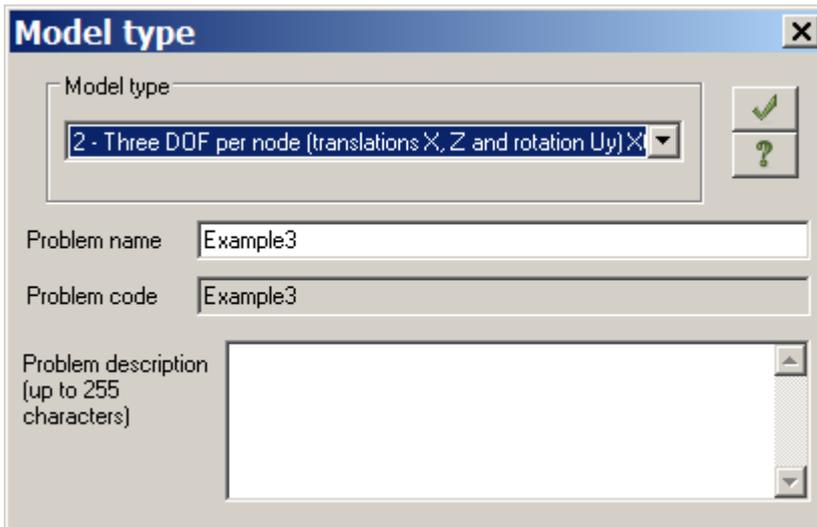


Figure 3.2 **Model type** dialog box



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.

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 first axis: spacing along the second axis:

L(m)	N	L(m)	N
4	3	4	2
 - other parameters remain by default (see Fig.3.3).
- ⇒ Click **Apply** .

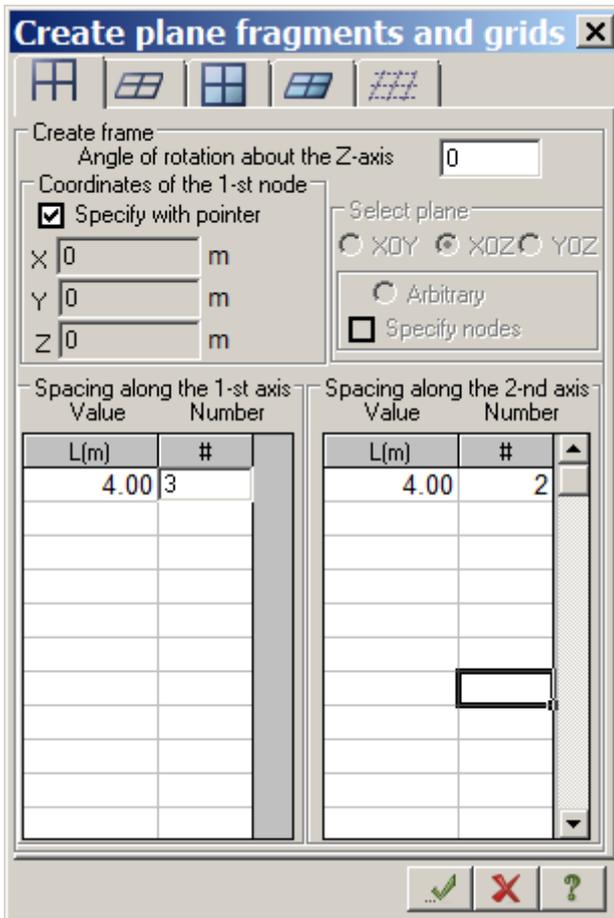


Figure 3.3 Create plane fragments and grids dialog box

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 – **Example3**;
 - location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

To present numbers of nodes and elements 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 **Element numbers** check box on the **Elements** tab.
- ⇒ On the **Nodes** tab, select the **Node numbers** check box.
- ⇒ Click **Redraw** .

To edit the model:

- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Select nodes No.10 and 11 with the pointer (the nodes will be coloured red).



You can select nodes either with a single click or by dragging selection window around appropriate nodes.

- ⇒ To delete selected nodes, on the **Create and edit** ribbon tab, on the **Edit** panel, click **Delete selected objects** . (Note that when you delete nodes, elements adjacent to these nodes will be automatically deleted).
- ⇒ On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .
- ⇒ Select elements No. 3 and 5 with the pointer (elements will be coloured red).



When you select nodes or elements on design model, you will see Contextual Tabs on the Ribbon User Interface. Contextual Tabs expose functionality specific only to the object in focus. They remain hidden when the object it works on is not selected. Contextual Tabs are mentioned to work with nodes or elements of the model. They contain commands to create and edit the model and can't be activated from **Results**, **More results** and **Design** ribbon tabs.

- ⇒ On the **Bars** contextual tab, on the **Edit bars** panel, click **Hinges** .
- ⇒ In the **Hinges** dialog box (see Fig.3.4), define nodes and directions along which there is no stiffness or there is limited stiffness for the restraint between one of the bar ends and model node. To do this, select appropriate check boxes:
 - 2nd node – UY.
- ⇒ Click **Apply** .

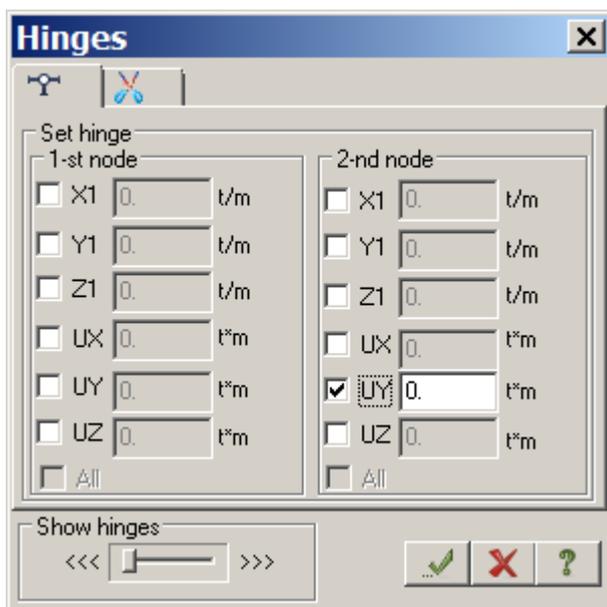


Figure 3.4 Hinges dialog box

To place the truss upon the frame:

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, click **Create truss** .
- ⇒ In the **Generate 2D truss** dialog box, to select the truss by shape of the chord, press the button  with arch (segmental) truss.
- ⇒ Then select web of the truss (in the second tab that becomes active). Click the upper left button.
- ⇒ When you define the web, the third tab of the dialog box is active. Define the following parameters of the truss (see Fig.3.5):
 - $L = 12$ m;
 - $K_f = 6$.
- ⇒ To obtain representation of the truss with all dimensions, click **Draw**.

- ⇒ Under **Coordinates of yellow node of truss**, make sure that the **Select node of fixation** check box is selected, and then select node No. 9 (coordinates of this node will be automatically displayed in the dialog box).
- ⇒ To place the truss upon the frame, click **Apply** .

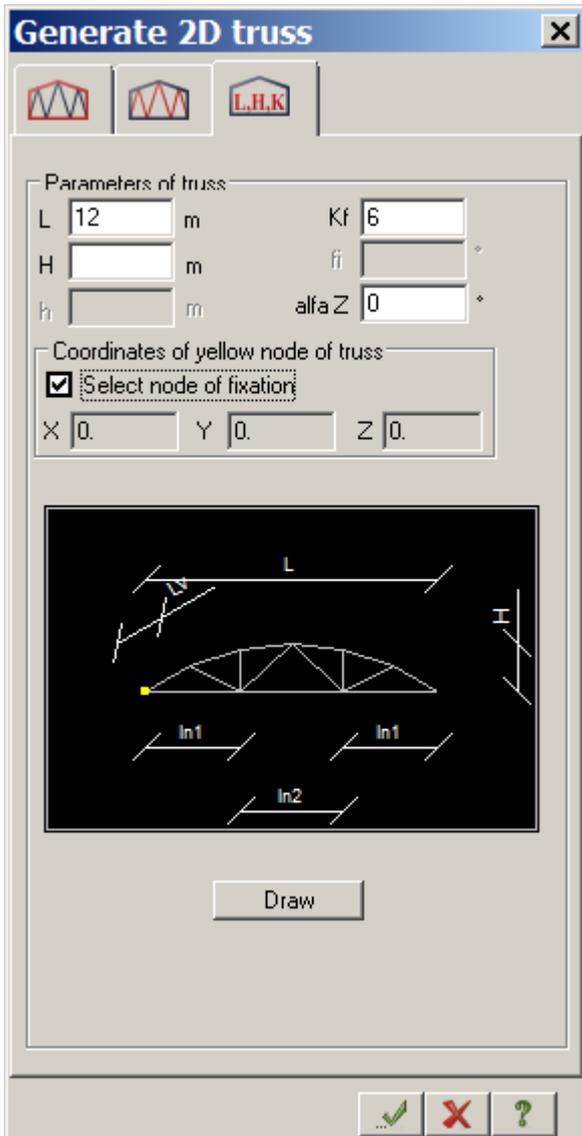


Figure 3.5 Create 2D truss dialog box

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.3.6), 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.

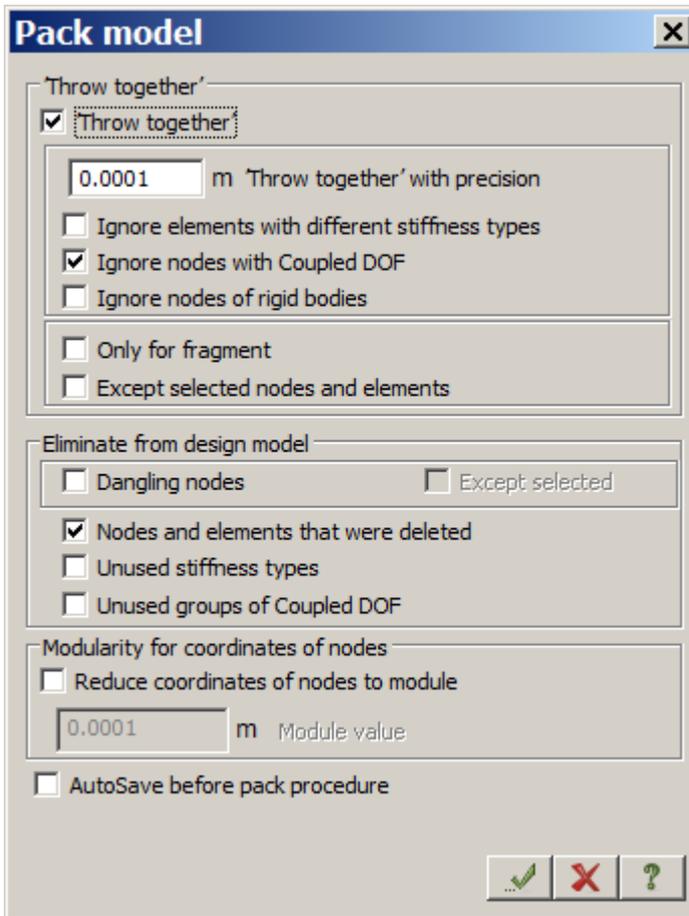


Figure 3.6 Pack model dialog box

The model shown in Fig.3.7 will be displayed on the screen.

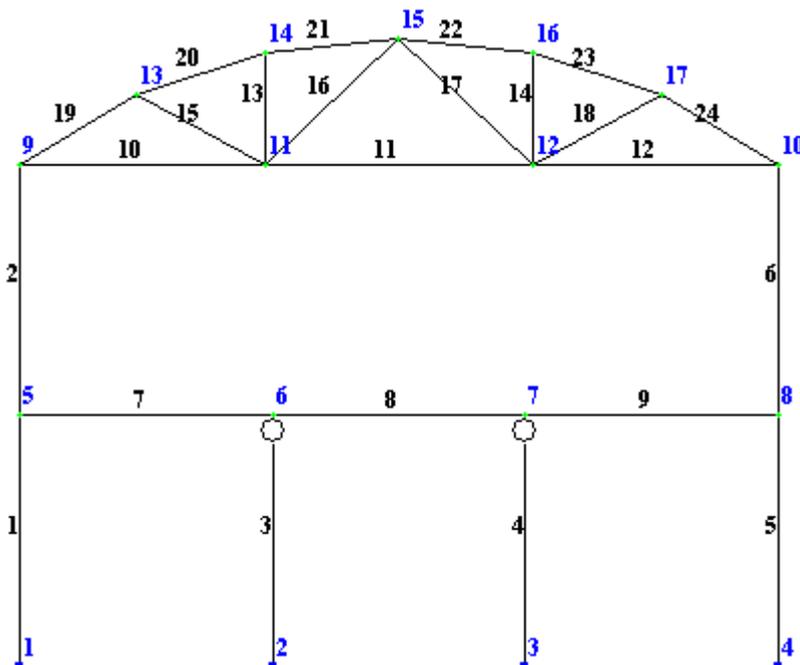


Figure 3.7 Design model of the frame with node and element numbers

Step 3. Defining boundary conditions

- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Select nodes No.1 and 4 with the pointer.

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Restraints** button .
- ⇒ 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**  (the nodes will be coloured blue).
- ⇒ Select nodes No. 2, 3 and impose restraints along X and Z. To do this, simply clear the UY check box.
- ⇒ 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. Defining design options

- ⇒ On the **More edit options** ribbon tab, on the **Design** panel, click **Design options for main model** command .
- ⇒ In the **Design options** dialog box (see Fig.3.8), define parameters for the first design option:
 - in the **Analysis of sections by** list, select **DCL**;
 - other parameters remain by default.
- ⇒ Click **Apply** .

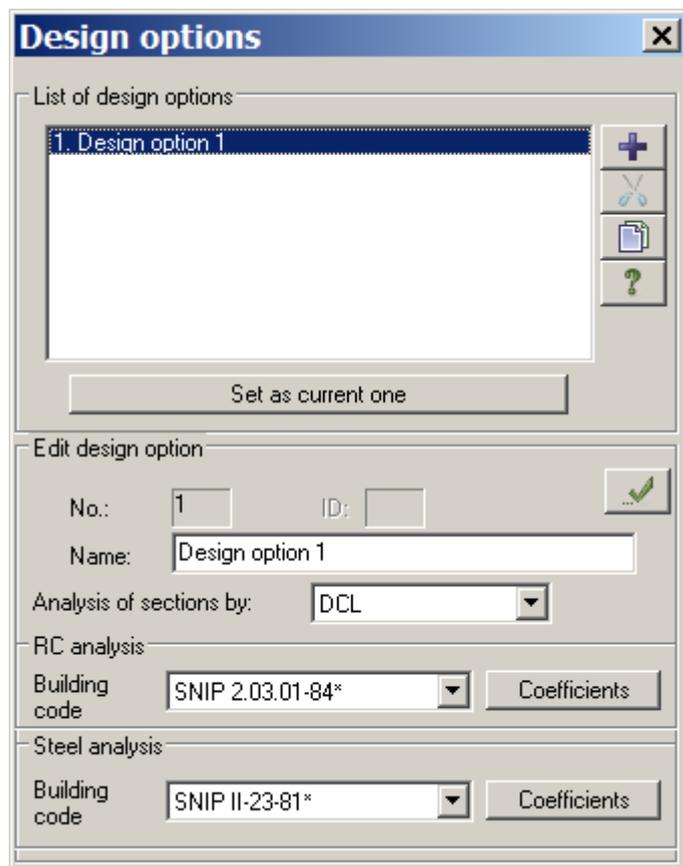


Figure 3.8 **Design options** dialog box

- ⇒ To close the **Design options** dialog box, click the **Close** button.

Step 5. Defining material properties to elements of the frame

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.3.9a), click **Add**. The dialog box expands to display the library of stiffness parameters. In the **Add stiffness** dialog box (see Fig.3.9b), select the **Database of steel sections** tab (the second tab).

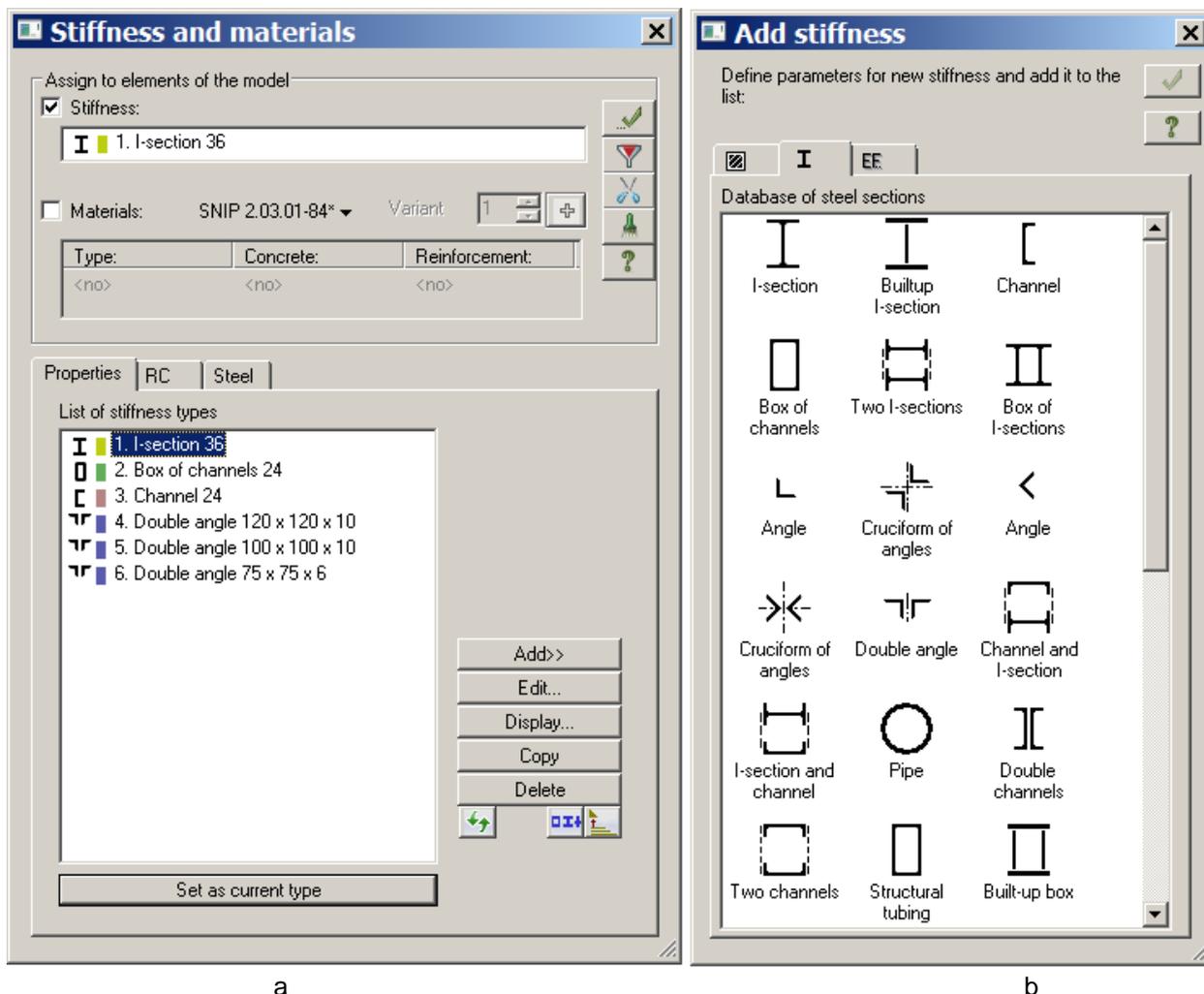


Figure 3.9 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ Double-click the **I-section** 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 specify the following parameters for I-section (see Fig.3.10):
 - in the **Steel table** box, click **Двутавр с непараллельными гранями полок** ;
 - in the **Shape** box, click **36**.
- ⇒ Click **OK**.

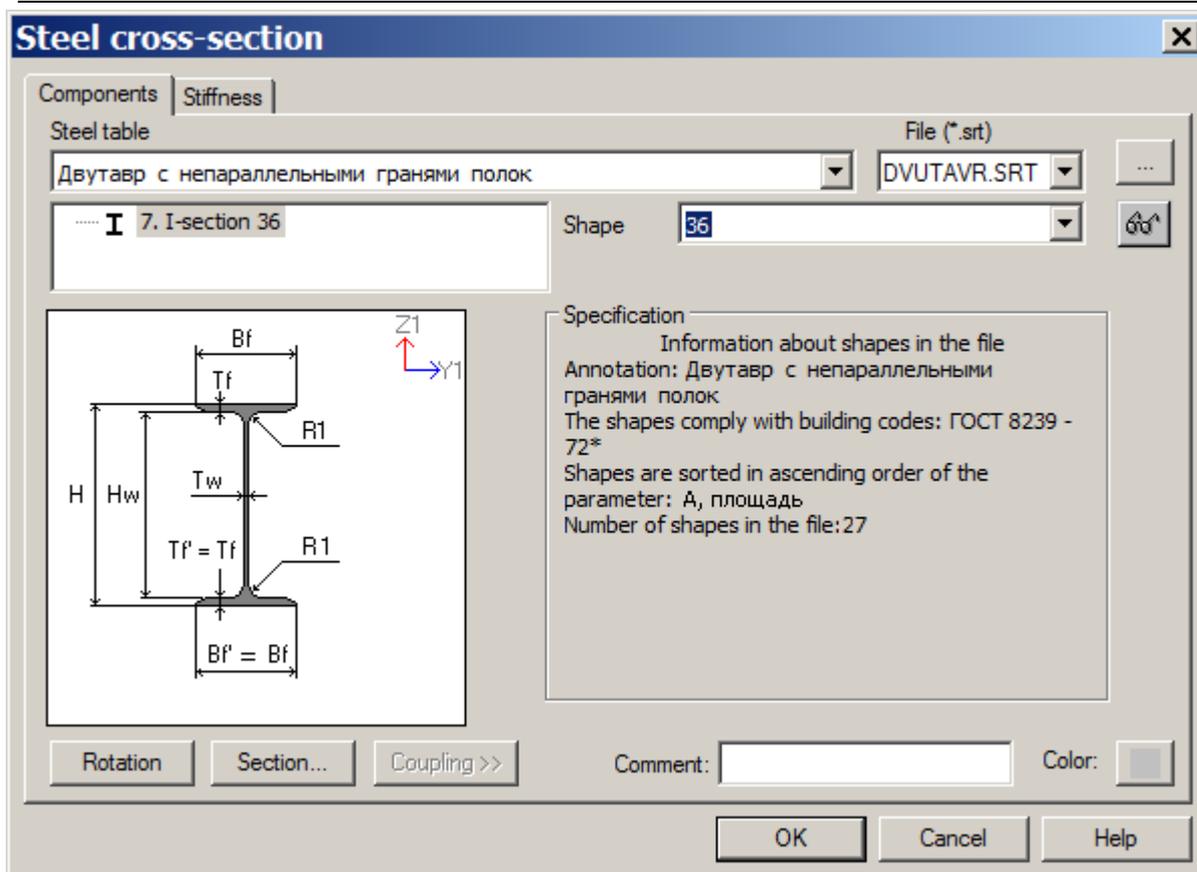


Figure 3.10 Steel cross-section dialog box

- ⇒ In the **Add stiffness** dialog box (see Fig.3.9), on the **Database of steel sections** (the second tab), double-click the **Box of channels** icon in the list.
- ⇒ In the **Steel cross-section** dialog box specify the following parameters for **Box of channels** section:
 - in the **Steel table** box, click **Швеллер с уклоном внутренних граней полок** ;
 - in the **Shape** box, click **24**.
- ⇒ Click **OK**.
- ⇒ In the **Add stiffness** dialog box, double-click the **Channel** icon in the list.
- ⇒ In another **Steel cross-section** dialog box specify the following parameters for **Channel** section:
 - in the **Steel table** box, click **Швеллер с уклоном внутренних граней полок** ;
 - in the **Shape** box, click **24**.
- ⇒ Click **OK**.
- ⇒ In the **Add stiffness** dialog box, double-click the **Double angle** icon in the list.
- ⇒ In another **Steel cross-section** dialog box specify the following parameters for **Double angle** section:
 - in the **Steel table** box, click **Уголок равнополочный** ;
 - in the **Shape** box, click **120 x 120 x 10**.
- ⇒ Click **OK**.
- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select '4. Double angle 120 x 120 x 10'.
- ⇒ Click **Copy** two times.
- ⇒ In the **List of stiffness types**, select '5. Double angle 120 x 120 x 10'.
- ⇒ Click **Edit**.

- ⇒ In another **Steel cross-section** dialog box specify the following parameters:
 - in the **Shape** box, click **100 x 100 x 10**.
- ⇒ Click **OK**.
- ⇒ In the **List of stiffness types**, select '6. Double angle 120 x 120 x 10'.
- ⇒ Click **Edit**.
- ⇒ In another **Steel cross-section** dialog box specify the following parameters:
 - in the **Shape** box, click **75 x 75 x 6**.
- ⇒ Click **OK**.
- ⇒ To hide the library of stiffness parameters, click **Add** in the **Stiffness and materials** dialog box.

To assign material properties to steel structures:



The 'Reinforced concrete and steel structures' mode is mentioned to select and check bar sections of steel elements according to SNIP II-23-81, SP 16.13330.2011, Eurocode 3.1.1 ENV 1993-1-1:1992 and LFRD 2nd edition (AISC). Analysis is carried out for one or several design combinations of forces (DCF), design combinations of loads (DCL) or forces obtained from the static analysis of structure. There are also checks for elements of plane stress state.

The selection and check procedures may be carried out for the following types of sections:

- truss elements and frame braces that are in axial compression and tension;
- beams that are in bending;
- columns that are in combined action of bending and compression.

The selection and check procedures may be carried out in two modes:

- mode for work with the model, in this mode analysis is carried out automatically for all elements defined by the user;
- mode for analysis of a separate element (local mode), in this mode the user could make alternative design for elements of structure, edit section dimensions, grade of steel, pattern of stiffeners, etc. Section dimensions of elements and utilization percentage for bearing capacity of element sections determined according to certain building code will be considered as analysis results.

- ⇒ In the **Stiffness and materials** dialog box, under **List of stiffness types**, click the stiffness type '1.I-section 36'.
- ⇒ Click **Set as current type**. Selected type will be displayed in the **Stiffness** box under **Assign to elements of the model**. You can also specify the current type by double-clicking the necessary type in the **List of stiffness types**.
- ⇒ Then, to assign materials for steel structures, in the **Stiffness and materials** dialog box, select the **Steel** tab (the third tab).
- ⇒ Select **Material** option and click **Add**.
- ⇒ In the **Parameters** dialog box (see Fig.3.11), in the **Steel** list box, define the steel grade **BCr3kп2-1** (it will be applied to all elements).
- ⇒ To confirm the data, click **OK**.

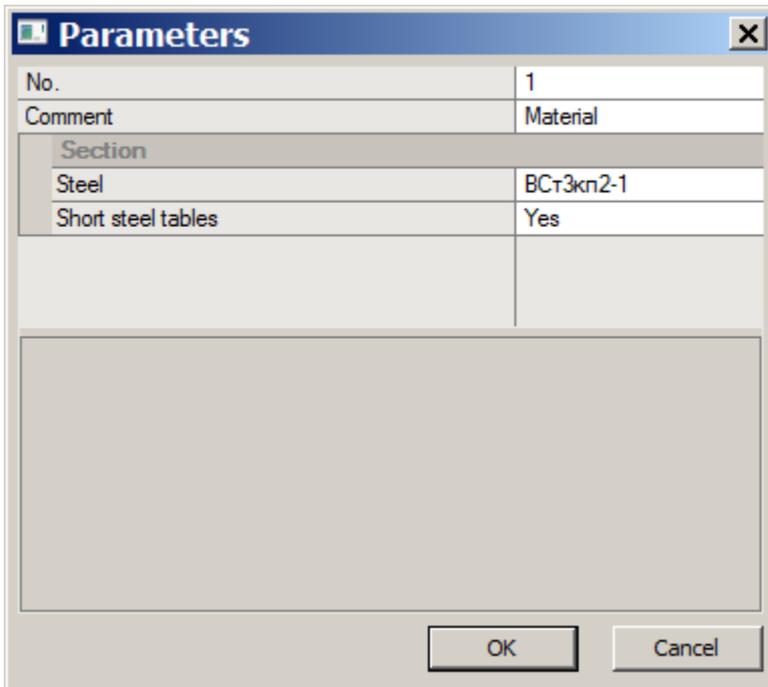


Figure 3.11 **Parameters** (for materials) dialog box

- ⇒ In the **Stiffness and materials** dialog box, select **Additional parameters** option and click **Add**.
- ⇒ In the **Parameters** dialog box (see Fig.3.12), define the following parameters for beams:
 - under **Element type**, click **Beam**;
 - under **Data for buckling analysis**, select the **Use length factors** check box;
 - define the length factor for buckling analysis of beam **Kb=0.33**;
 - in the **End conditions of compression flange of beam** list box, select the **2 or more dividing the span to equal parts** line;
 - under **Deflection analysis**, define maximum allowed deflection - $1/250$;
 - in the **Comment** line, type **Beams**;
 - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.

Parameters	
Building code	SNIP II-23-81*
No.	1
Comment	Beams
Element type	
Truss	<input type="radio"/>
Column	<input type="radio"/>
Beam	<input checked="" type="radio"/>
Service conditions factors and safety factor	
Yc buckling	0.95
Yc strength	1
Yn	1
Analysis is carried out	
within elasticity limits	<input checked="" type="radio"/>
with account of plasticity	<input type="radio"/>
Pure bending	<input type="checkbox"/>
Stiffeners	
place stiffeners	<input type="checkbox"/>
stiff. spacing, m	0
Deflection analysis	
Span length L, m	Auto
Maximum allowed deflection	1/250
Cantilever	<input type="checkbox"/>
Data for buckling analysis	
Kb	0.33
use length factors	<input checked="" type="checkbox"/>
Cantilever	<input type="checkbox"/>
Beam with one axis of symmetry	<input type="checkbox"/>
End conditions of compression flange of beam	2 or more dividing the ...
<div style="border: 1px solid gray; height: 100px; width: 100%;"></div>	
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Figure 3.12 Parameters (for beams) dialog box

- ⇒ In the **Stiffness and materials** dialog box, click **Add** once more.
- ⇒ In the **Parameters** dialog box (see Fig.3.13), define the following parameters for columns:
 - under **Element type**, click **Column**;
 - under **Effective lengths**, select the **Use length factor** check box;
 - define effective length factor relative to Z1-axis $K_z=1$;
 - define effective length factor relative to Y1-axis $K_y=1$;
 - define effective length factor for check by lateral-torsional buckling (calculation of factor F_b) $K_b=0.85$;
 - in the **Comment** line, type **Columns**;
 - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.

Parameters	
Building code	SNIP II-23-81*
No.	2
Comment	Columns
Element type	
Truss	<input type="radio"/>
Column	<input checked="" type="radio"/>
Beam	<input type="radio"/>
Service conditions factors and safety factor	
Yc buckling	1
Yc strength	1
Yn	1
Ultimate slenderness	
principal column	<input checked="" type="radio"/>
secondary column	<input type="radio"/>
other	<input type="radio"/>
In compression	180-60a
In tension	300
Analysis is carried out	
within elasticity limits	<input checked="" type="radio"/>
with account of plasticity	<input type="radio"/>
Effective lengths	
Kz	1
Ky	1
Kb	0.85
use length factors	<input checked="" type="checkbox"/>
Comment Arbitrary text that describes this set of additional parameters	
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Figure 3.13 Parameters (for columns) dialog box

- ⇒ In the **Stiffness and materials** dialog box, click **Add** once more.
- ⇒ In the **Parameters** dialog box (see Fig.3.14), define the following parameters for top chord of the truss:
 - under **Element type**, click **Truss**;
 - under **Effective lengths**, select the **Use length factor** check box;
 - define effective length factor relative to Z1-axis $K_z=1$;
 - define effective length factor relative to Y1-axis $K_y=1$;
 - under **Ultimate slenderness**, click **chord or support diagonal of truss** option;
 - in the **Comment** line, type **Top chord**;
 - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.

Parameters	
Building code	SNIP II-23-81*
No.	3
Comment	Top chord
Element type	
Truss	<input checked="" type="radio"/>
Column	<input type="radio"/>
Beam	<input type="radio"/>
Service conditions factors and safety factor	
Yc buckling	1
Yc strength	1
Yn	1
Additional Yc=0.8	<input type="checkbox"/>
Ultimate slenderness	
chord or support diagonal of truss	<input checked="" type="radio"/>
element of truss web (except support element)	<input type="radio"/>
single element of space truss system on bolts	<input type="radio"/>
other	<input type="radio"/>
In compression	180-60a
In tension	300
Effective lengths	
Kz	1
Ky	1
use length factors	<input checked="" type="checkbox"/>
Comment Arbitrary text that describes this set of additional parameters	
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Figure 3.14 Parameters (for top chord of the truss) dialog box

- ⇒ In the **Stiffness and materials** dialog box, click **Add** once more.
- ⇒ In the **Parameters** dialog box, define the following parameters for bottom chord of the truss:
 - under **Element type**, click **Truss**;
 - under **Effective lengths**, select the **Use length factor** check box;
 - define effective length factor relative to Z1-axis $K_z=0.33$;
 - define effective length factor relative to Y1-axis $K_y=0.33$;
 - under **Ultimate slenderness**, click **chord or support diagonal of truss** option;
 - in the **Comment** line, type **Bottom chord**;
 - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.
- ⇒ In the **Stiffness and materials** dialog box, click **Add** once more.
- ⇒ In the **Parameters** dialog box, define the following parameters for truss web:

- under **Element type**, click **Truss**;
 - under **Effective lengths**, select the **Use length factor** check box;
 - define effective length factor relative to Z1-axis $K_z=1$;
 - define effective length factor relative to Y1-axis $K_y=1$;
 - under **Ultimate slenderness**, click **element of truss web (except support element)** option;
 - in the **Comment** line, type **Truss web**;
 - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.

To assign material properties to elements of the frame:

- ⇒ In the **Stiffness and materials** dialog box, in the list of additional parameters for steel structures, select '**1.Beams**'. In this case stiffness '**1.I-section 36**' should be defined as current type in the **List of stiffness types** on the first tab.
- ⇒ Click **Set as current type**. Selected type of additional parameters will be displayed in the **Materials** box under **Assign to elements of the model**. You can also specify the current type of additional parameters by double-clicking the necessary type in the list.
- ⇒ On the **Select** toolbar, click **Select horizontal bars** button .
- ⇒ Select elements No. 7, 8 and 9 with the pointer. The elements will be coloured red.
- ⇒ 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.
- ⇒ On the **SELECT** toolbar, click **Select horizontal bars**  in order to make this command not active.
- ⇒ In the **Stiffness and materials** dialog box, in the list of additional parameters for steel structures, select '**2.Columns**'.
- ⇒ Click **Set as current type**.
- ⇒ In the **Stiffness and materials** dialog box, on the **Properties** tab (the first tab), select '**2.Box of channels 24**' in the **List of stiffness types**.
- ⇒ Click **Set as current type**.
- ⇒ On the **Select** toolbar, click **Select vertical bars** button .
- ⇒ Select vertical elements No. 1, 2, 5, 6 (external columns) with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the **Stiffness and materials** dialog box, under **List of stiffness types**, click the stiffness type '**3.Channel 24**'.
- ⇒ Click **Set as current type**.
- ⇒ Select vertical elements No. 3, 4 (internal columns) with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the **Stiffness and materials** dialog box, under **List of stiffness types**, click the stiffness type '**4.Double angle 120 x 120 x 10**'.
- ⇒ Click **Set as current type**.
- ⇒ In the same dialog box, select the **Steel** tab (the third tab) and in the list of additional parameters for steel structures select '**3.Top chord**'.
- ⇒ Click **Set as current type**.
- ⇒ To select elements of the top chord, on the **Select** toolbar, click **PolyFilter** .

- ⇒ In the **PolyFilter** dialog box, select the **Filter for elements** tab (the second tab) (see Fig.3.15).
- ⇒ Select **By FE numbers** check box and specify numbers of elements 19 – 24.
- ⇒ Click **Apply**.

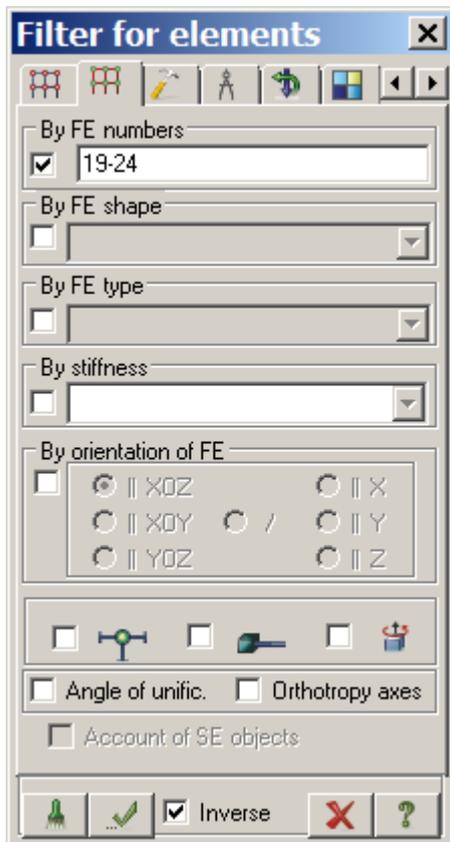


Figure 3.15 **Filter for elements** tab

- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the **Stiffness and materials** dialog box, under **List of stiffness types**, click the stiffness type '5.Double angle 100 x 100 x 10'.
- ⇒ Click **Set as current type**.
- ⇒ In the same dialog box, select the **Steel** tab (the third tab) and in the list of additional parameters for steel structures select '4.Bottom chord'.
- ⇒ Click **Set as current type**.
- ⇒ To select elements of the bottom chord, in the **PolyFilter** dialog box, on the **Filter for elements** tab, select **By FE numbers** check box and specify numbers of elements 10 – 12.
- ⇒ Click **Apply**.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the **Stiffness and materials** dialog box, under **List of stiffness types**, click the stiffness type '6.Double angle 75 x 75 x 6'.
- ⇒ Click **Set as current type**.
- ⇒ In the same dialog box, select the **Steel** tab (the third tab) and in the list of additional parameters for steel structures select '5.Truss web'.
- ⇒ Click **Set as current type**.
- ⇒ To select elements of truss web, in the **PolyFilter** dialog box, on the **Filter for elements** tab, select **By FE numbers** check box and specify numbers of elements 13 – 18.
- ⇒ Click **Apply**.

- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ Close the **PolyFilter** dialog box.

Step 6. Changing FE types for elements of truss

- ⇒ Select all elements of the truss.
- ⇒ On the **More edit options** ribbon tab, on the **Model** panel, click **Change FE type** .
- ⇒ In the **Change FE type** dialog box (see Fig.3.16), in the list of FE types, select **FE type 1 - FE of 2D truss**.
- ⇒ Click **Apply** .

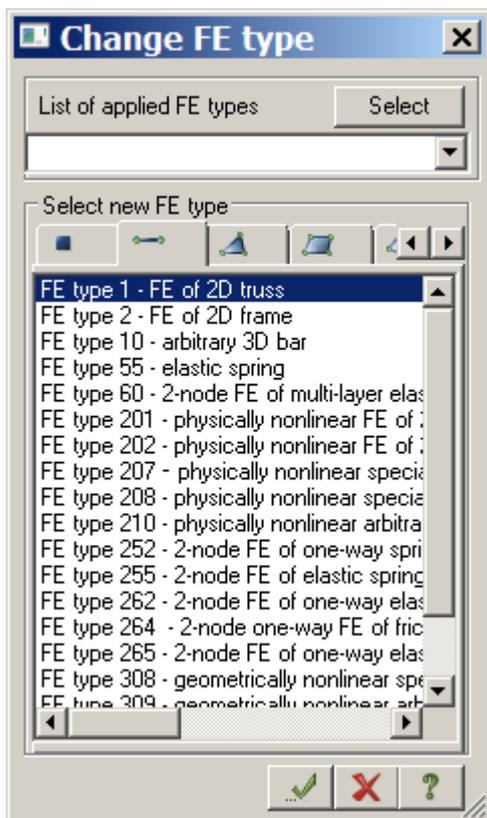


Figure 3.16 **Change FE type** dialog box

Step 7. 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** .
- ⇒ In the **Add dead weight** dialog box (see Fig.3.17), 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**  (uniformly distributed load equal to unit weight of elements is automatically applied to all elements of the structure).

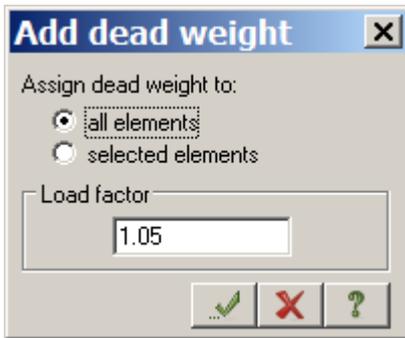


Figure 3.17 Add dead weight dialog box

- ⇒ Select nodes No.7, 8 and 9 with the pointer.
- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel, then select **Load on bars** command  from the **Loads on nodes and elements** drop-down list.
- ⇒ In the **Define loads** dialog box (see Fig.3.18), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

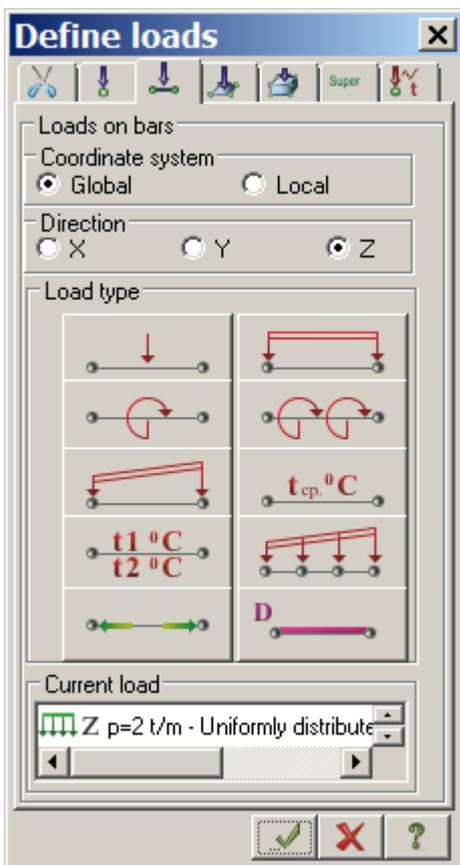


Figure 3.18 Define loads dialog box

- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 2.0$ t/m (see Fig.3.19).
- ⇒ Click **OK** .

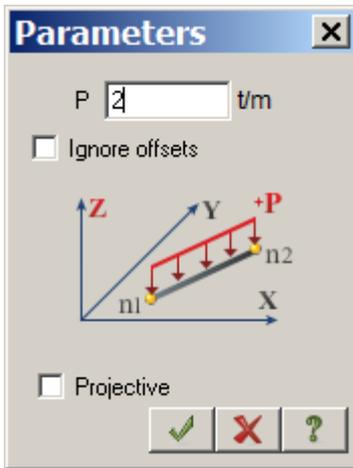
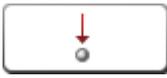


Figure 3.19 Load parameters dialog box

- ⇒ In the **Define loads** dialog box, click **Apply** .
- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Select nodes (where the truss is supported by the column) No.9 and 10 with the pointer.
- ⇒ In the **Define loads** dialog box, select **Loads on nodes** tab.
- ⇒ Specify **Global** coordinate system and direction along the Z-axis.
- ⇒ In the **Load type** area, click the **Concentrated load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 12t$.
- ⇒ Click **OK**.
- ⇒ In the **Define loads** dialog box, click **Apply**.
- ⇒ Select nodes of the top chord (nodes No.13-17) with the pointer. Then define concentrated load equal to $P = 24t$ on these nodes in a similar way.

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.

In a similar way perform the following:

- ⇒ Select elements No. 7, 8, 9 and define uniformly distributed load $P = 2t$ on these nodes.
- ⇒ Select nodes No. 9, 10 and define concentrated load $P = 2t$ on these nodes.
- ⇒ Select nodes No. 13 - 17 and define concentrated load $P = 4t$ on these nodes.

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.

In a similar way perform the following:

- ⇒ Select nodes No. 5, 10 and define concentrated load along global X-axis $P = -1.5t$ on these nodes.
- ⇒ Select node No. 9 and define concentrated load along global X-axis $P = -2t$ on this node.
- ⇒ Select node No. 8 and define concentrated load along global X-axis $P = -1.125t$ on this node.

To create load case No.4:

⇒ Change the number of the current load case for **4**.

To define nodal harmonic load:

⇒ Select node No. 6.

⇒ In the **Define loads** dialog box, select the **Loads on nodes** tab.

⇒ In the **Load type** area, click the **Nodal harmonic load** button .

⇒ In the **Nodal harmonic load** dialog box (see Fig.3.20), define the following parameters of load:

- additional weight of mass at node – 2t;
- direction of load – X;
- law of load variation – cos;
- load magnitude – 0.1t.

⇒ Click **OK** .

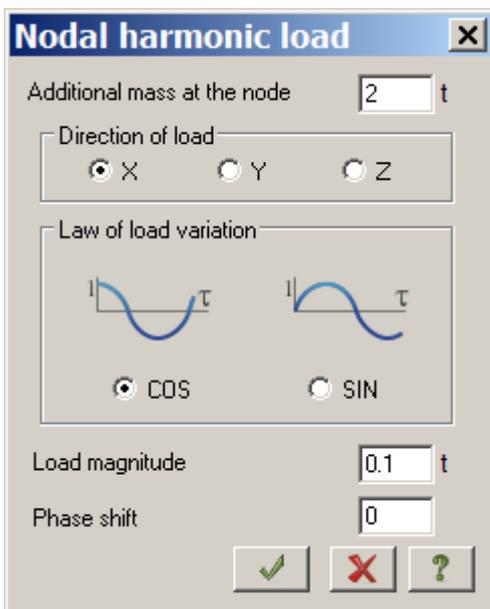


Figure 3.20 **Nodal harmonic load** dialog box

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

⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button  once again in order to make this command not active.

To define detailed information about load cases:

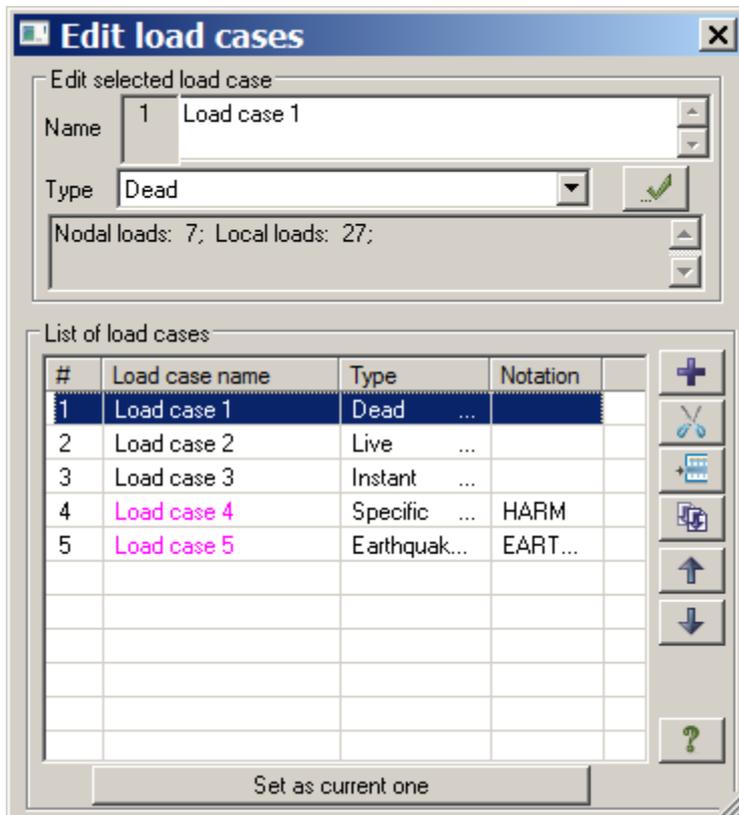
⇒ On the **Create and edit** ribbon tab, on the **Loads** panel, click **Edit load cases** . The **Edit load cases** dialog box is displayed on the screen (see Fig.3.21).

⇒ 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 **Live** and click **Apply** .

⇒ For load case 3 – in the **Edit selected load case** area, in the **Type** box, select **Instant** and click **Apply** .

- ⇒ For load case 4 – in the **Edit selected load case** area, in the **Type** box, select **Specific / Occasional** and click **Apply** .
- ⇒ To add the fifth 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 **Earthquake** and click **Apply** .

Figure 3.21 **Edit load cases** dialog box

Define parameters for dynamic analysis of the frame

Step 8. Generating dynamic load cases from the static ones

To generate table for account of static load cases for harmonic load:

- ⇒ 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.3.22), 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 – 0.9.
- ⇒ 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 – 0.8.
- ⇒ Click **Add**.

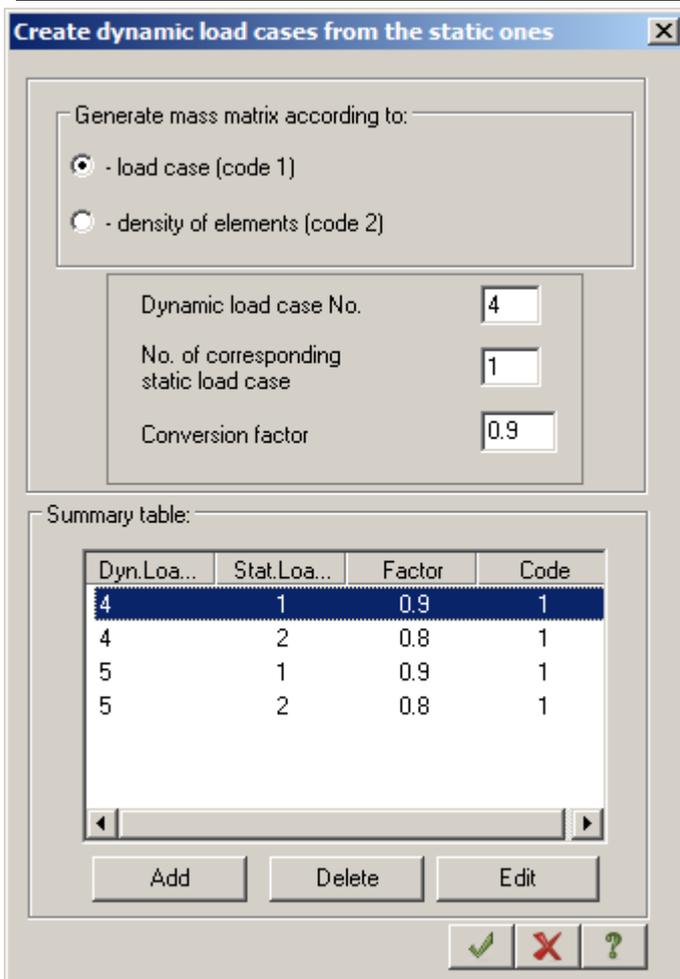


Figure 3.22 **Create dynamic load cases from the static ones** dialog box

To generate table for account of static load cases for earthquake load:

- ⇒ To create the third line of the summary table, in the **Create dynamic load cases from the static ones** dialog box, specify the following data:
 - dynamic load case No. – 5;
 - No. of corresponding static load case – 1;
 - conversion factor – 0.9.
- ⇒ Click **Add**.
- ⇒ To create the fourth line of the summary table, specify the following data:
 - dynamic load case No. – 5;
 - No. of corresponding static load case – 2;
 - conversion factor – 0.8.
- ⇒ Click **Add** and then **OK** .



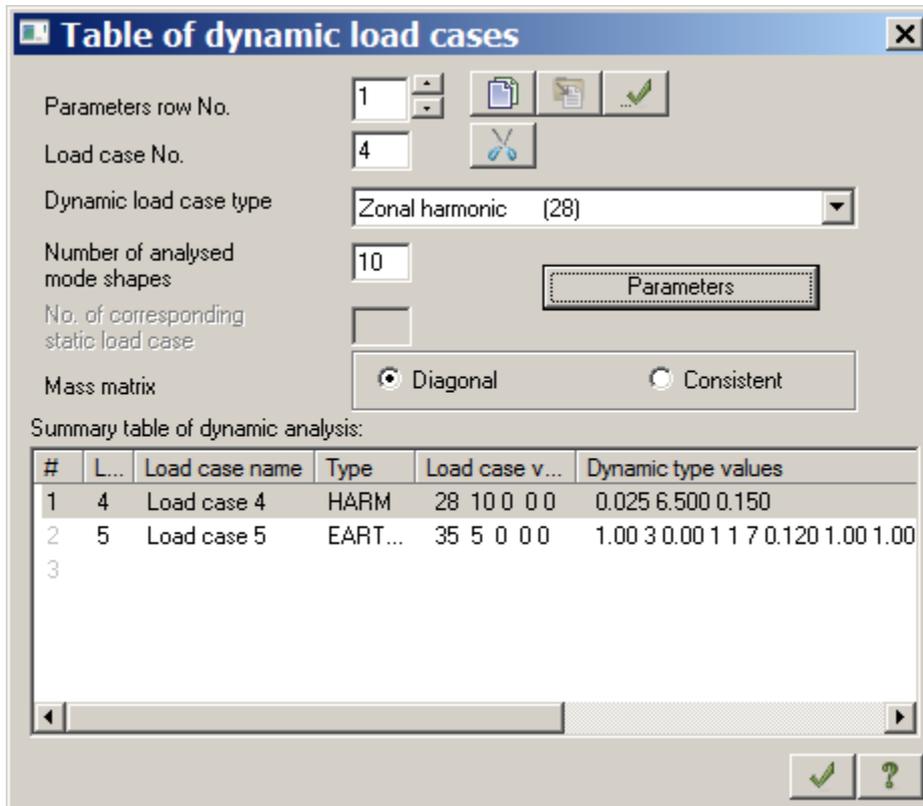
Specified static load cases generate weights of masses for dynamic load cases.

Step 9. Generating table of dynamic load cases

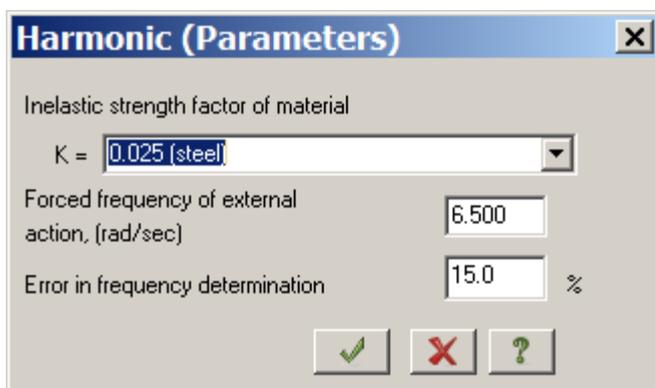
To define parameters for load case No.4:

- ⇒ 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.3.23), define the following data:
 - load case No. – 4;
 - dynamic load type – **Zonal harmonic (28)**;
 - number of analysed mode shapes – 10.
- ⇒ Click **Parameters**.

Figure 3.23 **Table of dynamic load cases** dialog box

- ⇒ In the **Harmonic (Parameters)** dialog box (see Fig.3.24), define the following data:
 - inelastic strength factor – $K = 0.025$ (steel);
 - forced frequency of external action – 6.5 rad/sec;
 - error in frequency determination – 15%.
- ⇒ Click **OK** .

Figure 3.24 **Harmonic (Parameters)** dialog box

To define parameters for load case No.5:

- ⇒ In the **Table of dynamic load cases** dialog box (see Fig.3.23), define the following data:

- load case No. – 5;
 - dynamic load type – **Earthquake /01.01.2000/SP 14.13330.2011/ (35)**;
 - number of analysed mode shapes – 5.
- ⇒ Click **Parameters**.
- ⇒ In the **Earthquake analysis parameters** dialog box (see Fig.3.25), define the following data:
- in the **Coefficients from table 3 SNIP II-7-81*** area, select the line $K1=0.12$;
 - direction cosines of earthquake load resultant in global coordinate system – $CX = 1$;
 - other parameters remain by default.
- ⇒ Click **OK** .

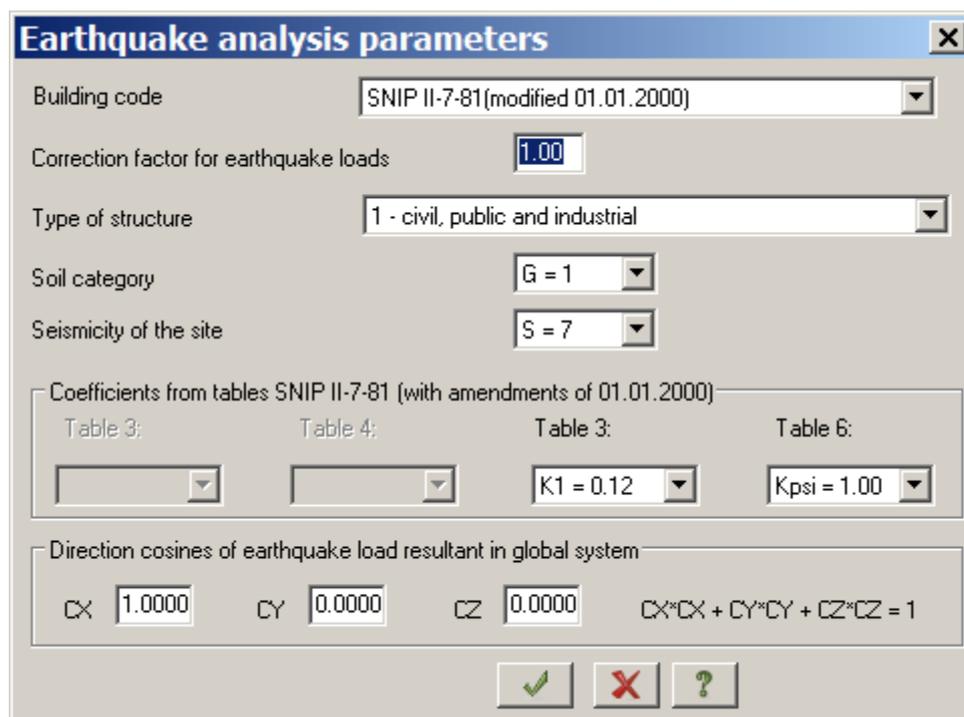


Figure 3.25 **Earthquake analysis parameters** dialog box

- ⇒ In the **Table of dynamic load cases** dialog box, click **OK**.

Step 10. Defining design sections for beams

- ⇒ Select horizontal elements No.7 – 9.



When you select nodes or elements on design model, you will see Contextual Tabs on the Ribbon User Interface. Contextual Tabs expose functionality specific only to the object in focus. They remain hidden when the object it works on is not selected.

*Contextual Tabs are mentioned to work with nodes or elements of the model. They contain commands to create and edit the model and can't be activated from **Results**, **More results** and **Design** ribbon tabs.*

- ⇒ On the **Bars** contextual tab, on the **Edit bars** panel, click **Design sections of bars** button .
- ⇒ In the **Design sections** dialog box (see Fig.3.26), specify number of sections $N = 5$.
- ⇒ Click **Apply**  (to analyse the structure according to serviceability limit states, it is necessary to define at least three design sections) and close the dialog box.



Figure 3.26 Design sections dialog box

Step 11. Defining structural elements



Structural element (STE) is a set of several finite elements that during design procedure will be considered as a single unit. Elements that form the part of the structural element should have no gaps, have the same stiffness type, should not be included into other structural elements and unification groups, have common nodes and belong to the same line.

To define structural element BEAM:

- ⇒ Select horizontal elements No. 7, 8 and 9.
- ⇒ To define structural elements, on the **More edit options** ribbon tab, select the **Design** panel and click **Structural elements** button .
- ⇒ In the **Structural elements** dialog box (see Fig.3.27), under **Edit structural elements**, click **Create**.

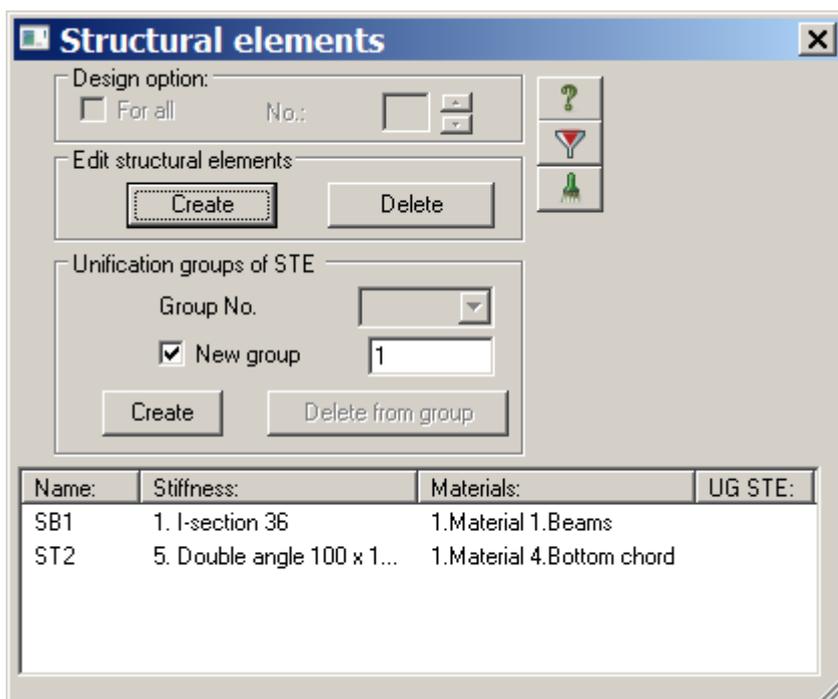


Figure 3.27 Structural elements dialog box

To define structural element TRUSS:

- ⇒ Select elements No. 10, 11 and 12.
- ⇒ In the **Structural elements** dialog box, under **Edit structural elements**, click **Create**.

Step 12. Defining deflection fixities at nodes of bending elements

- ⇒ Select elements No. 7, 8 and 9.
- ⇒ On the **More edit options** ribbon tab, on the **Design** panel, click **Deflection fixities** command .

- ⇒ In the **Deflection fixities** dialog box (see Fig.3.28), select **Create at nodes with NotSLA bars** option in the list. Then select Y1 and Z1 check boxes.
- ⇒ Click **Apply** (deflection of element sections is determined relative to the line that connects fixities at the ends of the element).

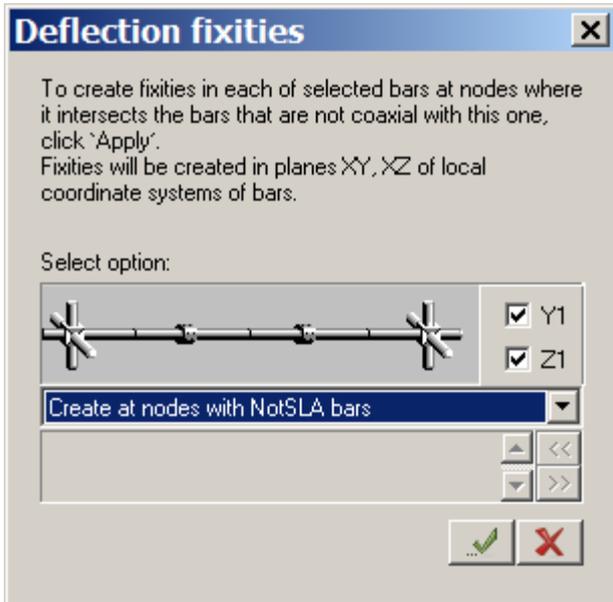


Figure 3.28 **Deflection fixities** dialog box

- ⇒ To close the **Deflection fixities** dialog box, click **Close**.

Step 13. Generating DCL table

- ⇒ On the **Analysis** ribbon tab, select the **More calculations** panel and click **DCL** button .



*As the type of load cases was defined in the **Edit load cases** dialog box (see Fig.3.21), the DCL table is generated automatically with parameters accepted by default for every load case. Now you have to modify parameters for the fourth and the fifth load cases and define combinations.*

- ⇒ In the **Design combinations of loads** dialog box (see Fig.3.29), select building code **SNIP 2.01.07-85*** and for load cases 4 and 5, double-click the **Sign variable** cell and define **+/-**.

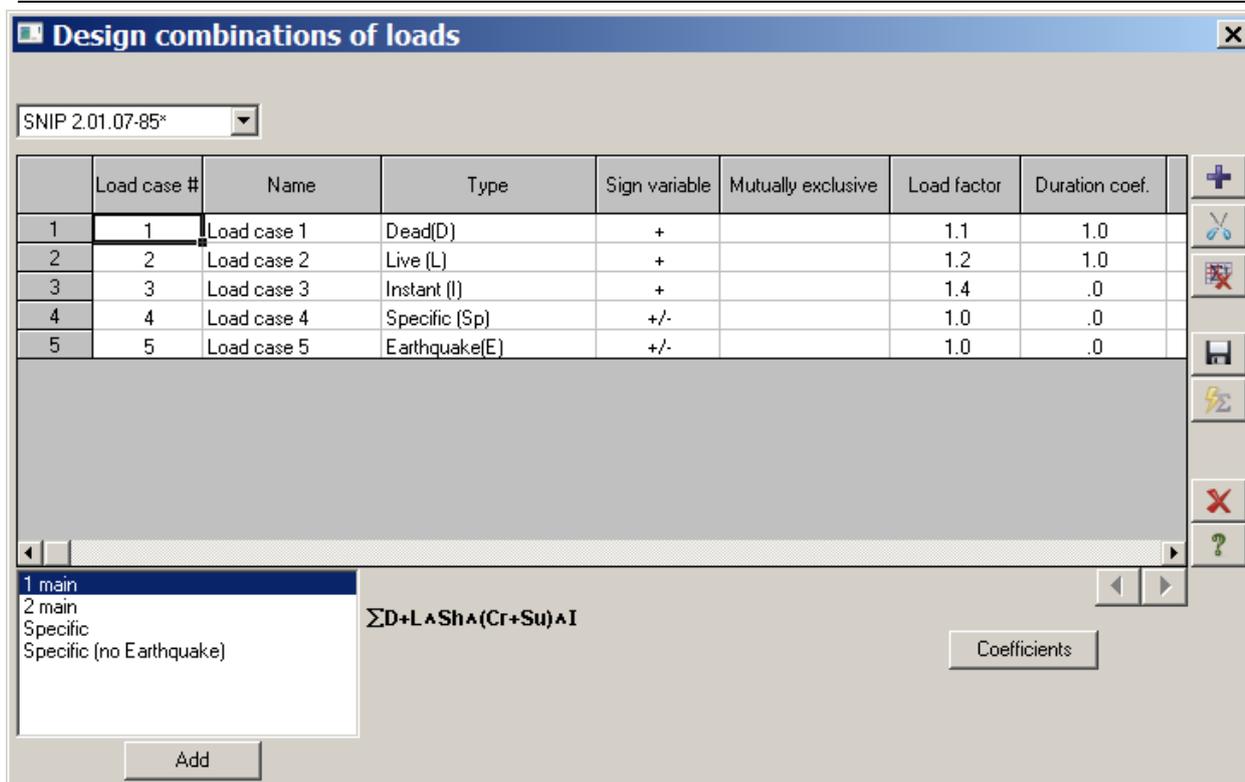


Figure 3.29 Design combination of loads dialog box

⇒ To define combinations, do the following steps:

- in the list of combinations, select the first row (**1 main**) and click **Add**.
- then in the list of combinations, select the second row (**2 main**) and click **Add**.
- in the list of combinations, select the row **Specific (Earthquake)** and click **Add**.
- in the list of combinations, select the row **Specific (no Earthquake)** and click **Add**. The columns with coefficients according to applied formulas for combinations by SNIP 2.01.07-85 will be displayed in the table.

⇒ To save defined data, click **Save data** .

⇒ To close the **Design combinations of loads** dialog box, click **Close**.



Design combinations of loads (DCL) are calculated as the sum of appropriate values of nodal displacements and forces (stresses) in elements. These values are added according to building codes (unlike calculation of DCF where extreme values of stresses at specific points of bar sections are used as criterion for determination of dangerous combinations).

Step 14. Defining parameters for stability analysis of the frame

⇒ To carry out stability analysis of the frame from DCL combinations, on the **Analysis** ribbon tab, select the **More calculations** panel and click **Stability** button .

⇒ In the **Stability** dialog box (see Fig.3.30), define the following parameters:

- select **Stability analysis** check box;
- to define the type of analysis, click **by DCL** option button;
- under **Load cases**, select **All combinations** check box;
- in the **Number of buckling modes to be calculated** box, define number **3**.

⇒ Click **OK** .

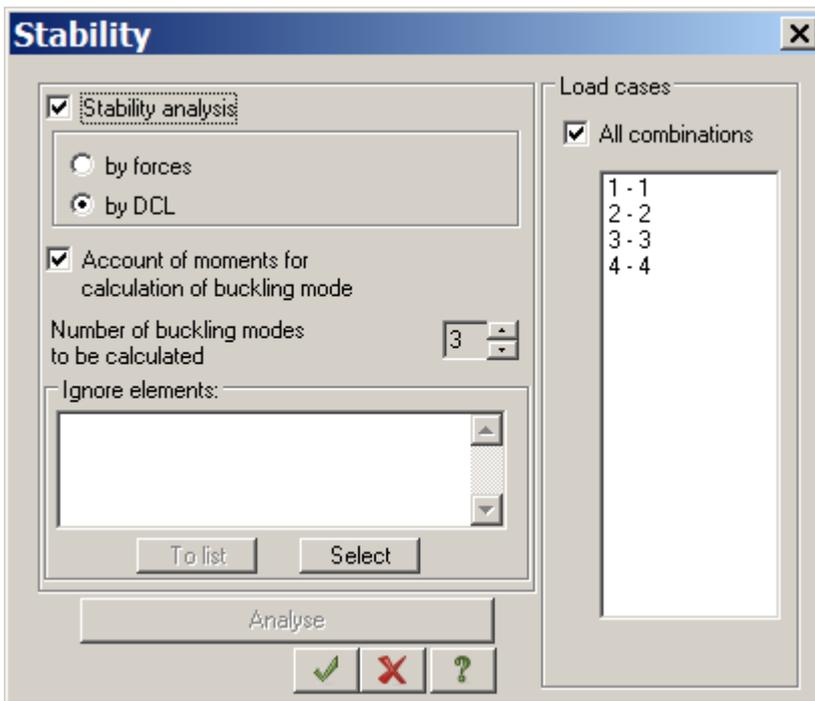


Figure 3.30 **Stability** dialog box

Step 15. Complete analysis of frame

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

Step 16. Review and evaluation of static & dynamic analyses results



*When analysis procedure is complete, to review and evaluate results of 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 (see Fig.3.31). To display the model without nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button .

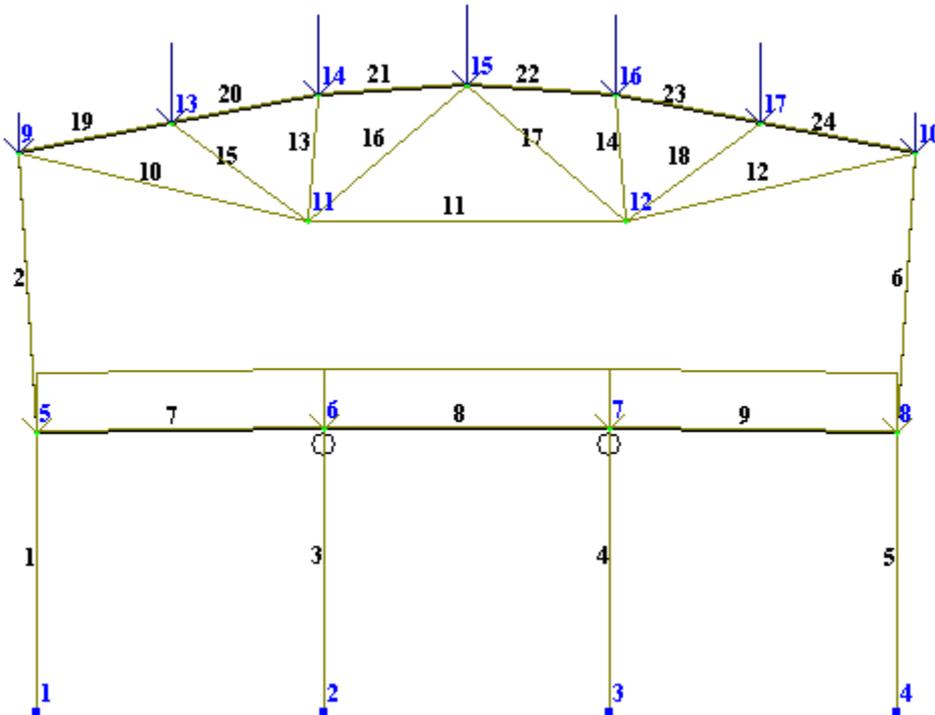


Figure 3.31 Design model with account of nodal displacements

To present diagrams of internal forces:

- ⇒ To display diagram M_y , on the **Results** tab, select **Forces in bars** panel and click **Moment diagrams (M_y)** button .
- ⇒ 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 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 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 mode shape, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button  to make this command not active.
- ⇒ To display the 3rd mode shape of the 5th load case, on the Status bar (displayed at the bottom of the screen), change the number of active load case for **5** (as described above). Then in the **Mode shape No. (component, period)** list, change number of mode shape for **3** and click **Apply** .

To animate the 3rd mode shape of the 5th load case:

- ⇒ 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 3rd mode shape of the 5th 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.3.32), click **Play** button .
- ⇒ To close the **Animate mode shape** dialog box, click **Close**.

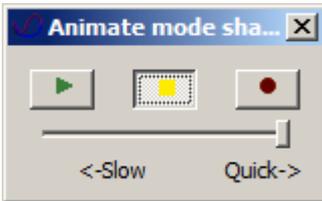


Figure 3.32 **Animate mode shape** dialog box

- ⇒ 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 generate and review tables of analysis results:

- ⇒ To present table with periods of vibrations, 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.3.33), select **Periods of vibrations** in the list.
- ⇒ Click **Apply** (to generate tables in HTML format, select appropriate option). To generate table and work further in [Document Maker \(DOC-SAPR module\)](#), select RPT format.

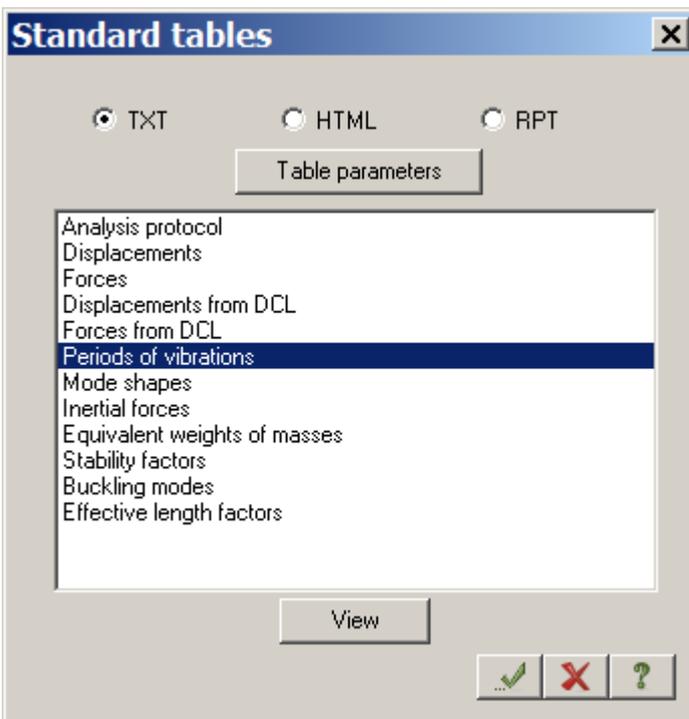


Figure 3.33 **Standard tables** dialog box

- ⇒ To close the table, on the FILE menu, click **Close**.
- ⇒ To present table with equivalent weights of masses at nodes of design model, in the **Standard tables** dialog box, select appropriate row in the list.
- ⇒ Click **Apply** .
- ⇒ In the **Choose load case No.** dialog box (see Fig.3.34), select **All load cases** option and click **OK** .

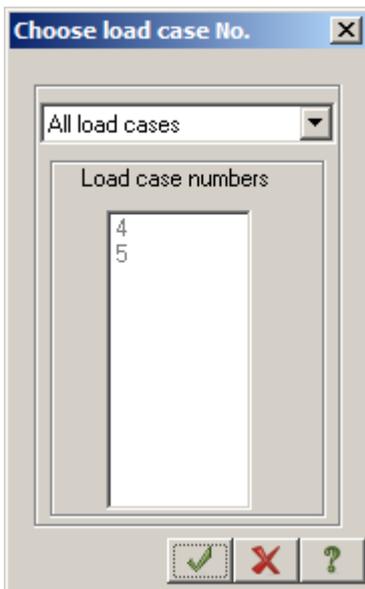


Figure 3.34 **Choose load case No.** dialog box

⇒ Close the **Standard tables** dialog box.

To evaluate analysis results by DCL:

- ⇒ To switch to results of static analysis, on the **Deformations** panel, select the **Deformed shape** command  in the **Stress strain state** drop-down list.
- ⇒ To present analysis results by DCL, on the Status bar (displayed at the bottom of the screen), click **Results by DCL** button .
- ⇒ Diagrams of internal forces are presented on the screen in the same way as described above. The generation of tables with analysis results by DCL is also the same.
- ⇒ To switch to another DCL number, on the Status bar in the **Load case No.** list, select appropriate row and click **Apply** .

To generate tables with stability factors:

- ⇒ To present table with stability factors, in the **Standard tables** dialog box, select appropriate row in the list.
- ⇒ Click **Apply** .

To evaluate results of stability analysis of the frame:

- ⇒ To present the buckling mode on the screen, on the **Deformations** panel, select the **Buckling mode** command  in the **Stress strain state** drop-down list.
- ⇒ To switch to another DCL number, on the Status bar in the **Load case No.** list, select appropriate row and click **Apply** .
- ⇒ To present the next buckling mode, on the Status bar, in the **Mode shape No. (component, period)** list, select appropriate number and click **Apply** .
- ⇒ To present effective length factor on the screen, on the **More results** ribbon tab, on **Stability** panel, click **Ly factors** button .

Step 17. Review and evaluate results from steel analysis



When analysis procedure is complete, to review and evaluate results of steel analysis, select the **Design** ribbon tab (for standard ribbon interface).

To present mosaic plots for the check of assigned sections of steel bars:

- ⇒ To present mosaic results (for assigned cross-sections, check for ultimate limit state), on the **Design** ribbon tab, on the **Steel: check and select** panel, click **Check, ULS** button .
- ⇒ To present mosaic results (for assigned cross-sections, check for local buckling), on the **Design** ribbon tab, on the **Steel: check and select** panel, click **Check, LB** button .

To generate the table with results (assigned sections):

- ⇒ To generate tables with analysis results, on the **Results** ribbon tab, select **Tables** panel and click **Analysis results tables for steel**  in the **Documents** drop-down list.
- ⇒ In the **Table of results** dialog box (see Fig.3.35), select the **Check** option.
- ⇒ Click **Apply**  (to generate tables in HTML format, select appropriate option). To generate table and work further in **Document Maker** (**DOC-SAPR** module), select RPT format. It is also possible to present tables in Excel format.

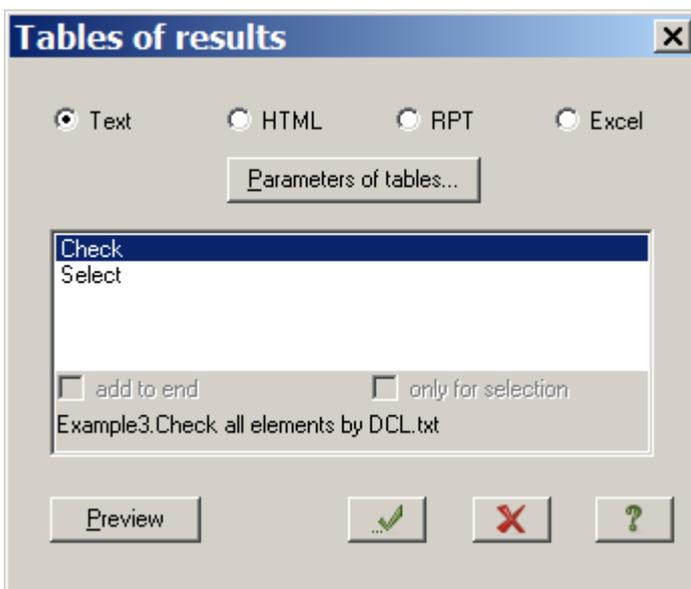


Figure 3.35 **Tables of results (steel analysis)** dialog box

- ⇒ To close the table, on the FILE menu, click **Close**.

To generate the table with results (selected sections):

- ⇒ In the **Table of results** dialog box (see Fig.3.35), define the **Select** option.
- ⇒ Click **Apply** .