

## Example 12. Analysis of steel framework of the structure and preparing data to KM-SAPR module

In this lesson you will learn how to:

- carry out static analysis of 3D frame and calculate DCF;
- select and check steel sections for elements of frame;
- analyse joints.

### Description:

Schematic presentation of the frame and its boundary conditions are presented in Fig.12.1.

Sections of elements:

extreme and middle columns – I-section No. 35K1;

longitudinal beams – I-section No. 30;

transverse beams – built-up I-section;

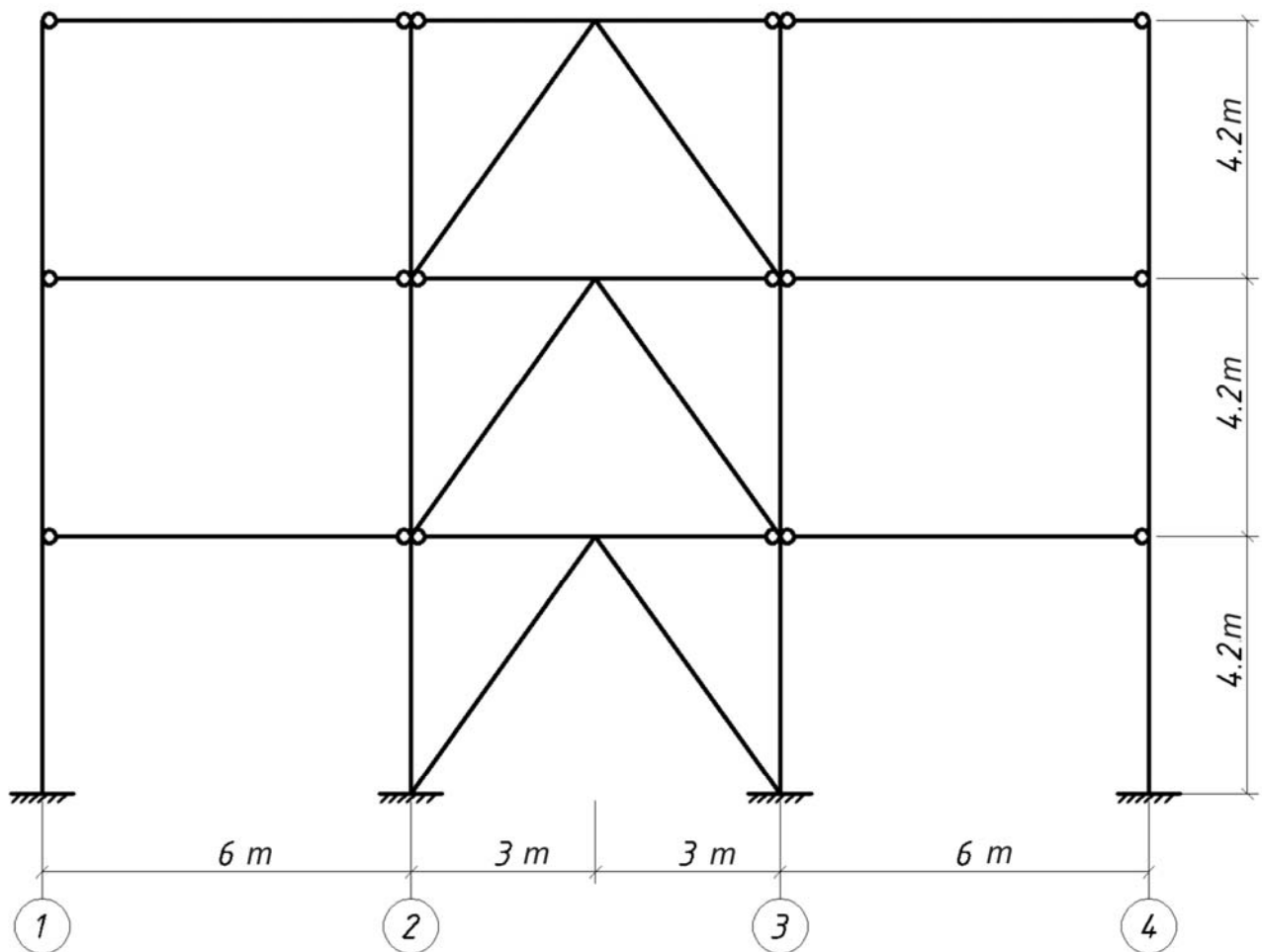
vertical bracing – double angle 75 x 75 x 6.

Loads:

load case 1 – dead weight of elements of model;

load case 2 – uniformly distributed load on beams;

load case 3 – wind load along the X-axis.



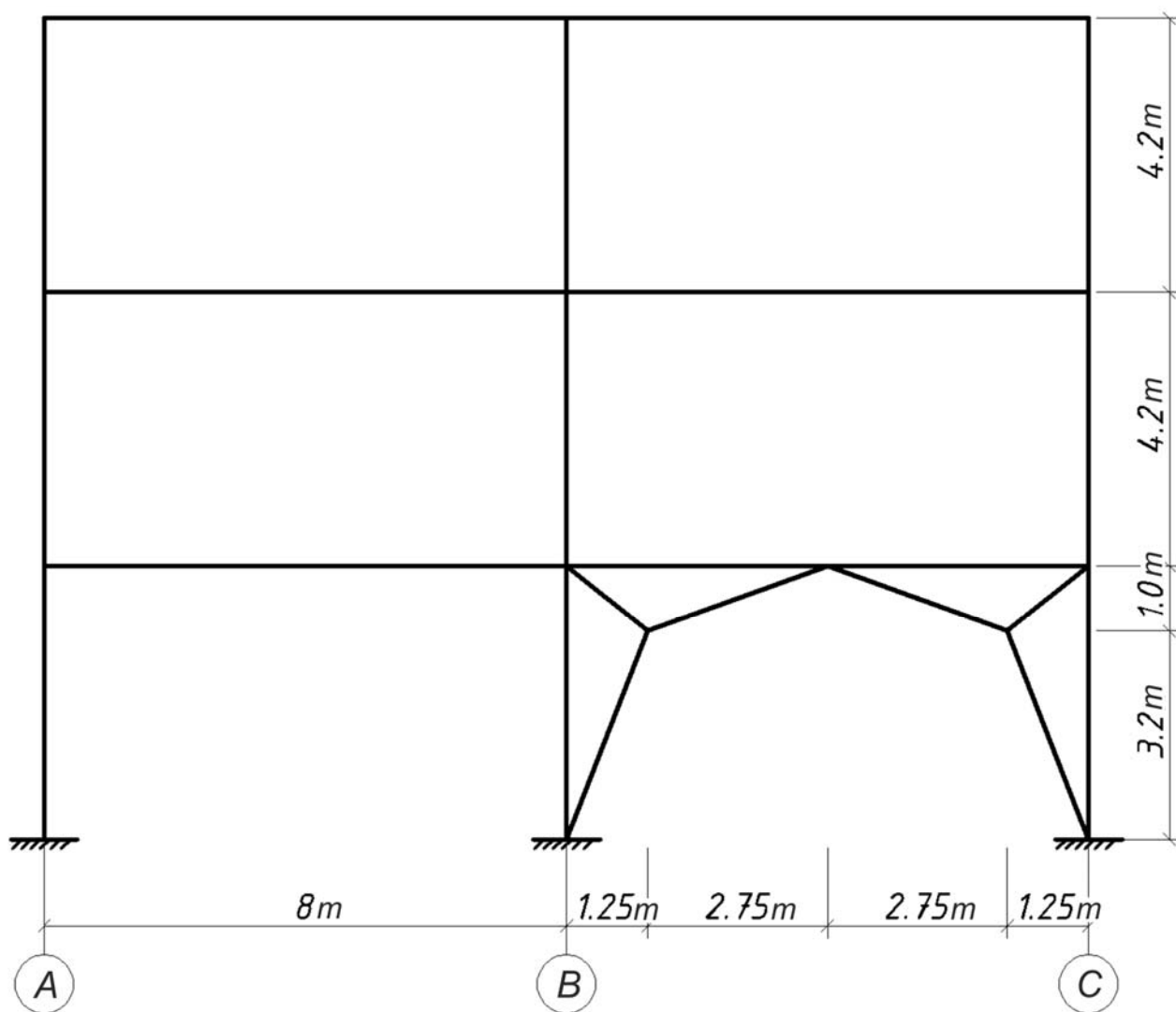




Figure 12.1 Cross-sectional plan of the structure

- ⇒ On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2016** and then click **LIRA-SAPR 2016**.

## Step 1. Creating new problem

- ⇒ On the FILE menu, click **New** (button  on the toolbar).
- ⇒ In the **Model type** dialog box (see Fig.12.2) specify the following data:
- problem name – **Example12**;
  - model type – **5 – Six degrees of freedom per node** (translations X, Y, Z and rotations Ux, Uy, Uz).
- ⇒ Click **OK** .

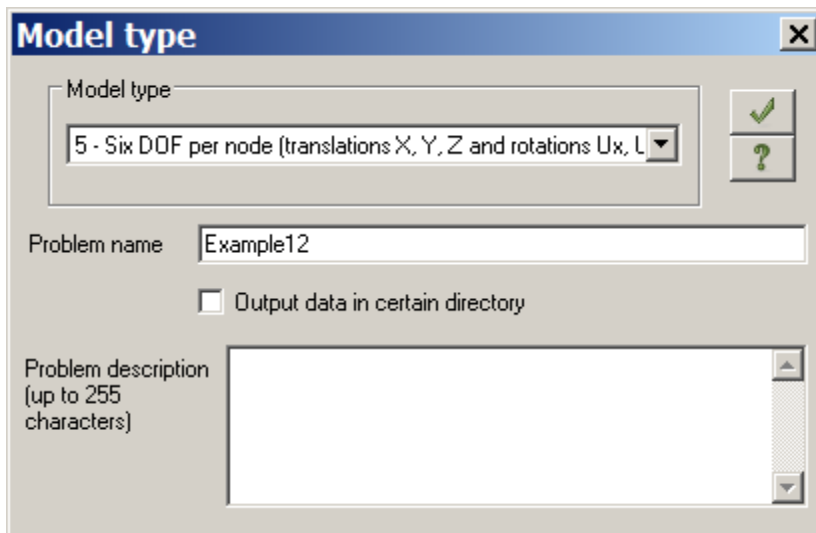




Figure 12.2 **Model type** dialog box



*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 5 (Six 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 5 (Six 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

To generate 3D frame:

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** list and click the **3D frame**  command.

- ⇒ In the **Create plane fragments and grids** dialog box (see Fig.12.3), clear the **Generate floor slabs and divide bars** and the **Generate foundation slab** check boxes.
- ⇒ Then specify the following data for 3D frame:
  - spacing along X:                      spacing along Y:                      spacing along Z:
  - |             |          |
|-------------|----------|
| <b>L(m)</b> | <b>N</b> |
| 8           | 2        |

<b>L(m)</b>	<b>N</b>
6	3

<b>L(m)</b>	<b>N</b>
4.2	3
  - other parameters remain by default.
- ⇒ Click **Apply** .

**3D frame**

☒ Specify with pointer

X  m

Y  m

Z  m

☐ Generate floor slabs and divide bars

☐ Generate foundation slab

☐ Use parameters for mesh of floor slab

☒ Apply restraints

☒ X ☒ Y ☒ Z

☒ UX ☒ UY ☒ UZ

Angle of rotation about the Z-axis

X		Y		Z	
Value	Number	Value	Number	Value	Number
8.00	2	6.00	3	4.20	3

☒ Generate nodes at places of intersection with other FE











Figure 12.3 3D frame 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 – **Example12**;
  - 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** .

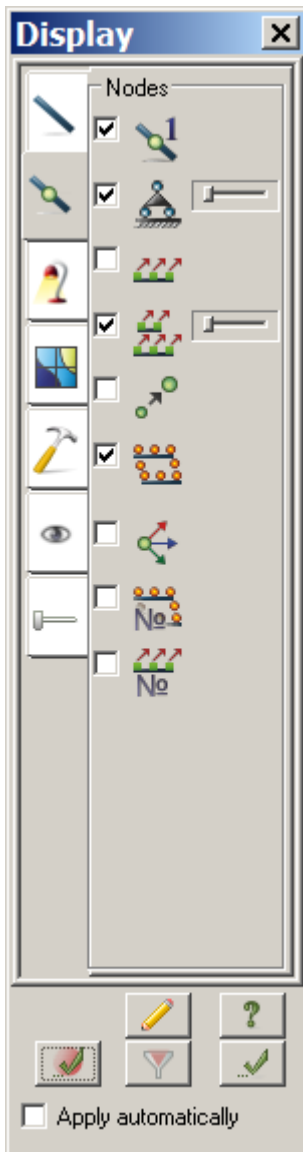




Figure 12.4 **Display** dialog box

To create hinges at nodes where beams connect columns:

- ⇒ 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.12.5).
- ⇒ Select **By orientation of FE** check box and specify option **II Y**.
- ⇒ Click **Apply** .

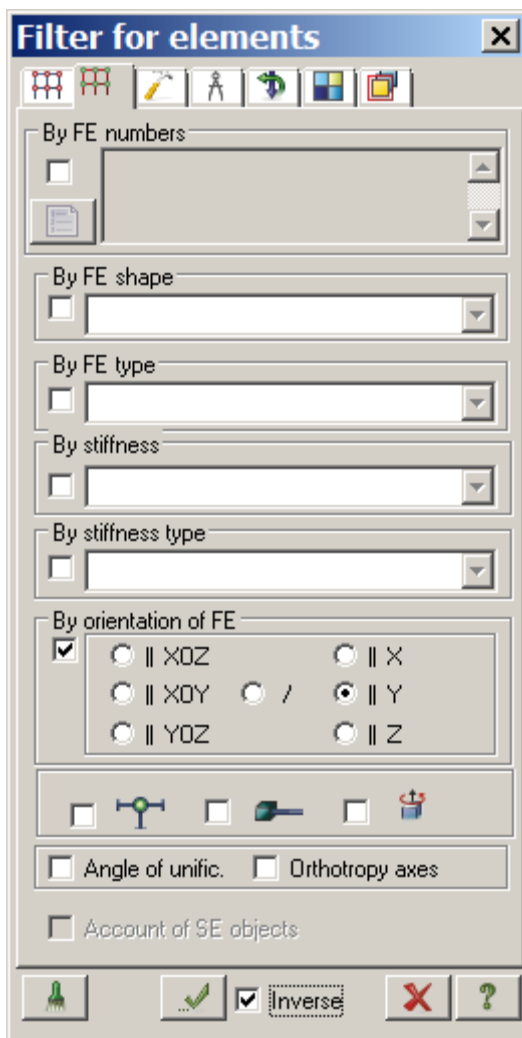




Figure 12.5 Filter for elements tab



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**, **Advanced results** and **Design** ribbon tabs.

- ⇒ On the **Bars** contextual tab, on the **Edit bars** panel, click **Hinges** .
- ⇒ In the **Hinges** dialog box (see Fig.12.6), 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:
  - 1st node – UY, UZ;
  - 2nd node – UY, UZ.
- ⇒ Click **Apply** .

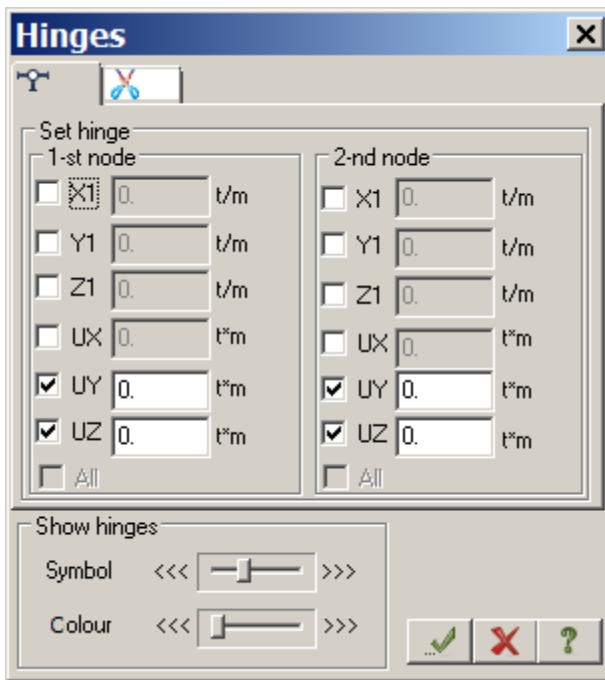





Figure 12.6 Hinges dialog box

⇒ Close the **PolyFilter** dialog box.

To add vertical bracing between axes 2 and 3:

- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **Select elements** (button  on the toolbar).
- ⇒ With the pointer, select the following elements on the model: No. 2, 8, 12, 13, 14, 41, 42, 43, 70, 71, 72 (selected elements will be coloured red).
- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Add element** drop-down list and click **Divide into N equal parts** .
- ⇒ The **Add element** dialog box is presented with the **Divide into N equal parts** tab open (see Fig.12.7).
- ⇒ On this tab, specify N = 2.
- ⇒ Click **Apply** .

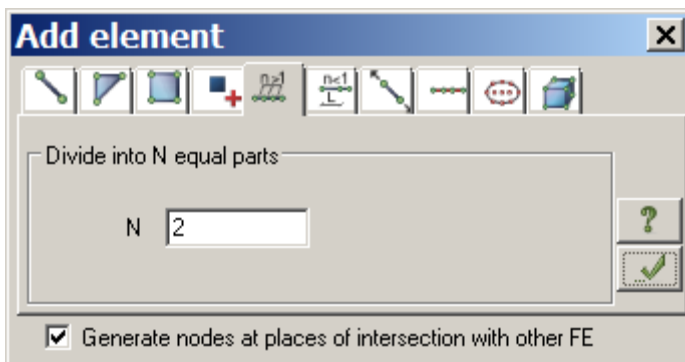




Figure 12.7 Add element dialog box

⇒ In the same dialog box, select the **Add bar** tab.

- ⇒ Make sure that the **Specify nodes with pointer** and **Account of intermediate nodes** check boxes are selected.
- ⇒ To add bar elements between nodes, specify with the pointer the following pairs of nodes in sequence: No. 7 and 51, 4 and 51, 19 and 54, 16 and 54, 31 and 57, 28 and 57 (in this case the rubber-band line is automatically stretched between the nodes that you select).
- ⇒ On the **Select** toolbar, click **Select elements** (button  on the toolbar).
- ⇒ Select new elements No. 110, 111, 112, 113, 114 and 115.
- ⇒ On the **Create and edit** ribbon tab, on the **Edit** panel, point to **Copy** drop-down list and click **Copy by one node** .
- ⇒ The **Copy objects** dialog box is presented with the **Copy by one node** tab open (see Fig.12.8).

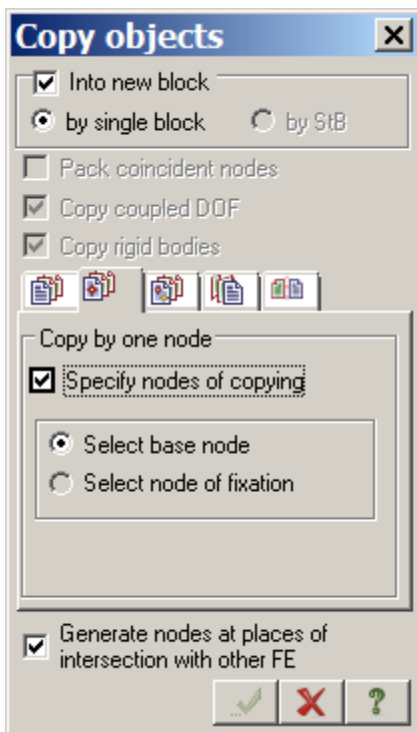




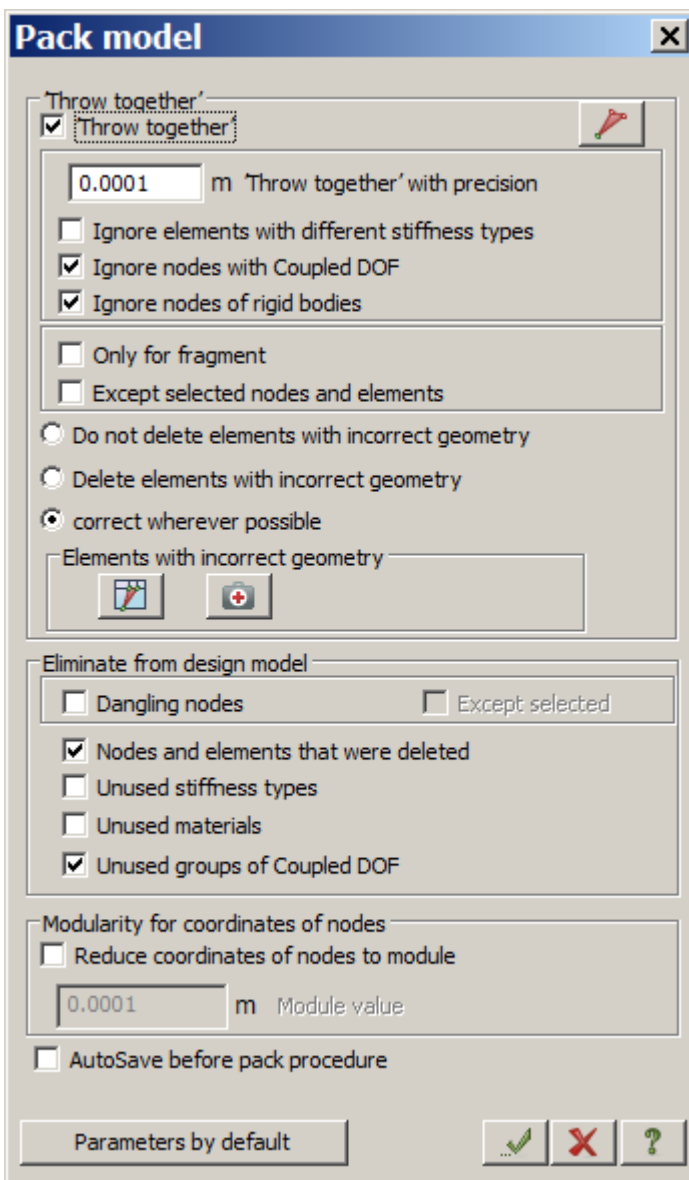
Figure 12.8 **Copy objects** dialog box

- ⇒ Specify with the pointer node where braces adjoin the middle of the beam No. 57 (the node will be coloured pink and in the **Copy objects** dialog box, the **Select base node** option becomes selected automatically).
- ⇒ Then select with the pointer (left mouse button) the nodes that the fragment should be copied to – nodes No. 58 and 59.



#### 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.12.9), 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 12.9 **Pack model** dialog box

To add vertical bracing between axes B and C:

- ⇒ 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 Fig.12.10), define coordinates for the node:
  - X(m)    Y(m)    Z(m)
  - 9.25    0    3.2.
- ⇒ Click **Apply** .

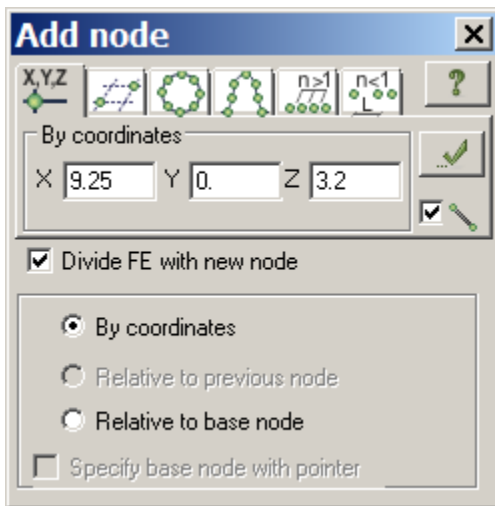



Figure 12.10 Add node dialog box

⇒ Then define new coordinates for the node:

▪ X(m)    Y(m)    Z(m)  
14.75    0        3.2.


⇒ Click **Apply** .

⇒ 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.

⇒ Make sure that the **Specify nodes with pointer** and **Account of intermediate nodes** check boxes are selected.

⇒ To add bar elements between nodes, specify with the pointer the following pairs of nodes in sequence: No. 2 and 60, 14 and 60, 49 and 60, 49 and 61, 15 and 61, 3 and 61 (in this case the rubber-band line is automatically stretched between the nodes that you select).

⇒ On the **Select** toolbar, click **Select elements** (button  on the toolbar).

⇒ Select new elements No. 119 – 124.

⇒ On the **Create and edit** ribbon tab, on the **Edit** panel, point to **Copy** drop-down list and click **Copy by one node** .

⇒ The **Copy objects** dialog box is presented with the **Copy by one node** tab open.





⇒ Specify with the pointer node where braces adjoin the middle of the beam No. 49 (the node will be coloured pink and in the **Copy objects** dialog box, the **Select base node** option becomes selected automatically).

⇒ Then select with the pointer (left mouse button) the nodes that the fragment should be copied to – nodes No. 50.

⇒ Pack the model ones again as described above.

### Step 3. Defining design options

To define the first design option:

- ⇒ 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.12.11), define parameters for the first design option:
  - in the **Analysis of sections by** list, select **DCF**;
  - to select the DCF table, click **Add/Edit DCF table** button .
  - in the new **Design combinations of forces** dialog box, click **OK** .
  - other parameters in the **Design options** dialog box remain by default.
- ⇒ In the **Design options** dialog box, click **Apply** .

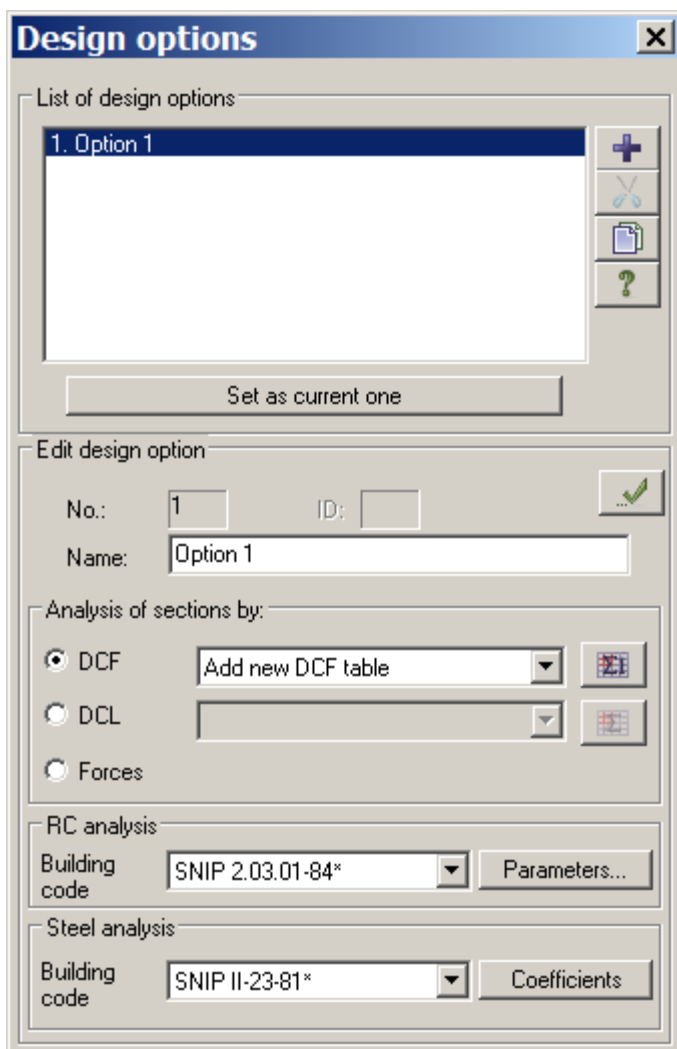



Figure 12.11 **Design options** dialog box

- ⇒ Close the **Design options** dialog box.

## Step 4. Defining material properties to elements of the model

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.12.12a), click **Add**. The dialog box expands to display the library of stiffness parameters. In the **Add stiffness** dialog box (see Fig.12.12b), select the **Database of steel sections** tab (the second tab).

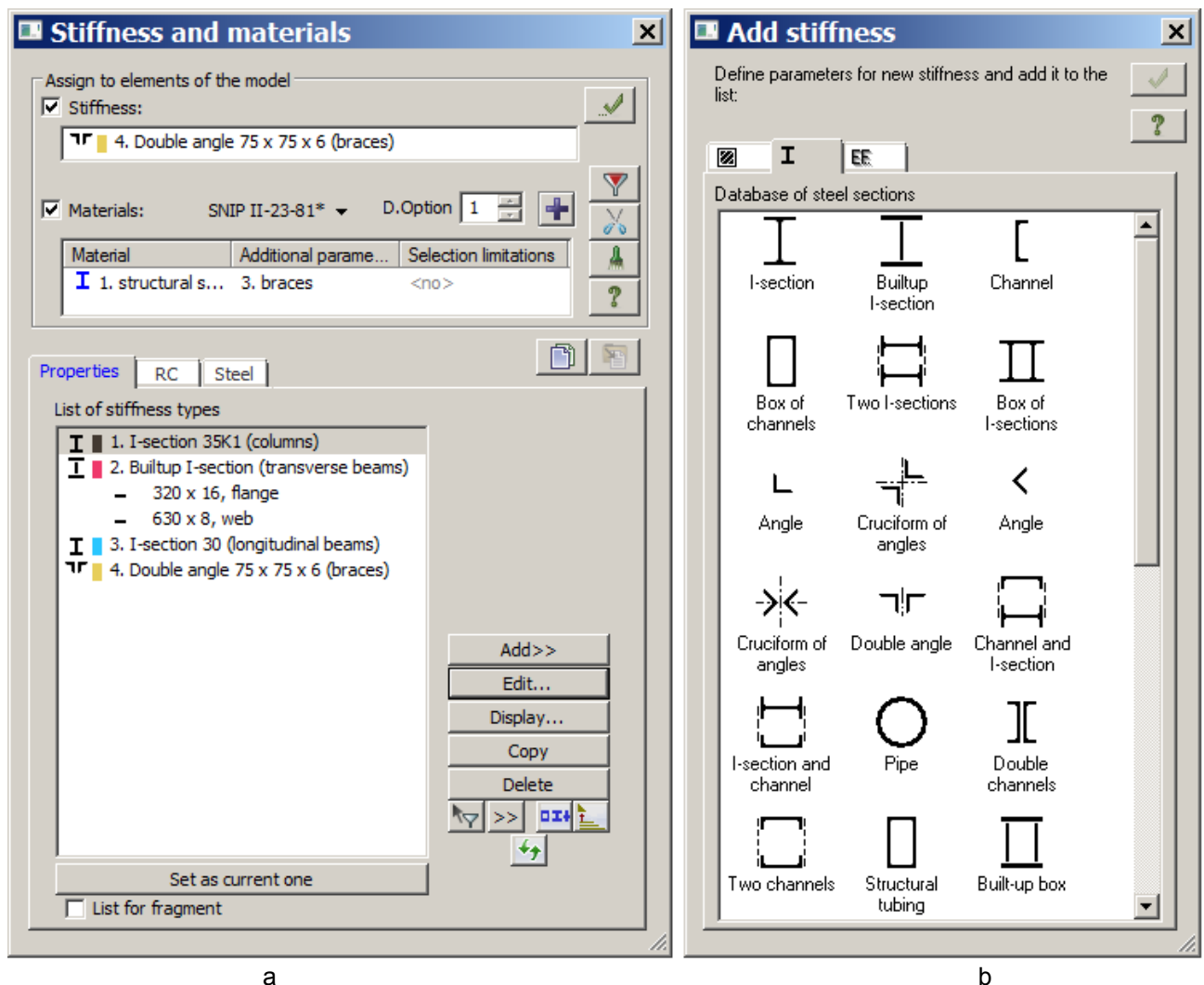


Figure 12.12 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ 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 (see Fig.12.13), specify the following parameters for I-section:
  - in the **Profile** box, click 'Dvutavr s parallel'nyimi granyami polok tipa K(kolonnyj)';
  - in the shape box, click **35K1**.
- ⇒ Click **OK**.

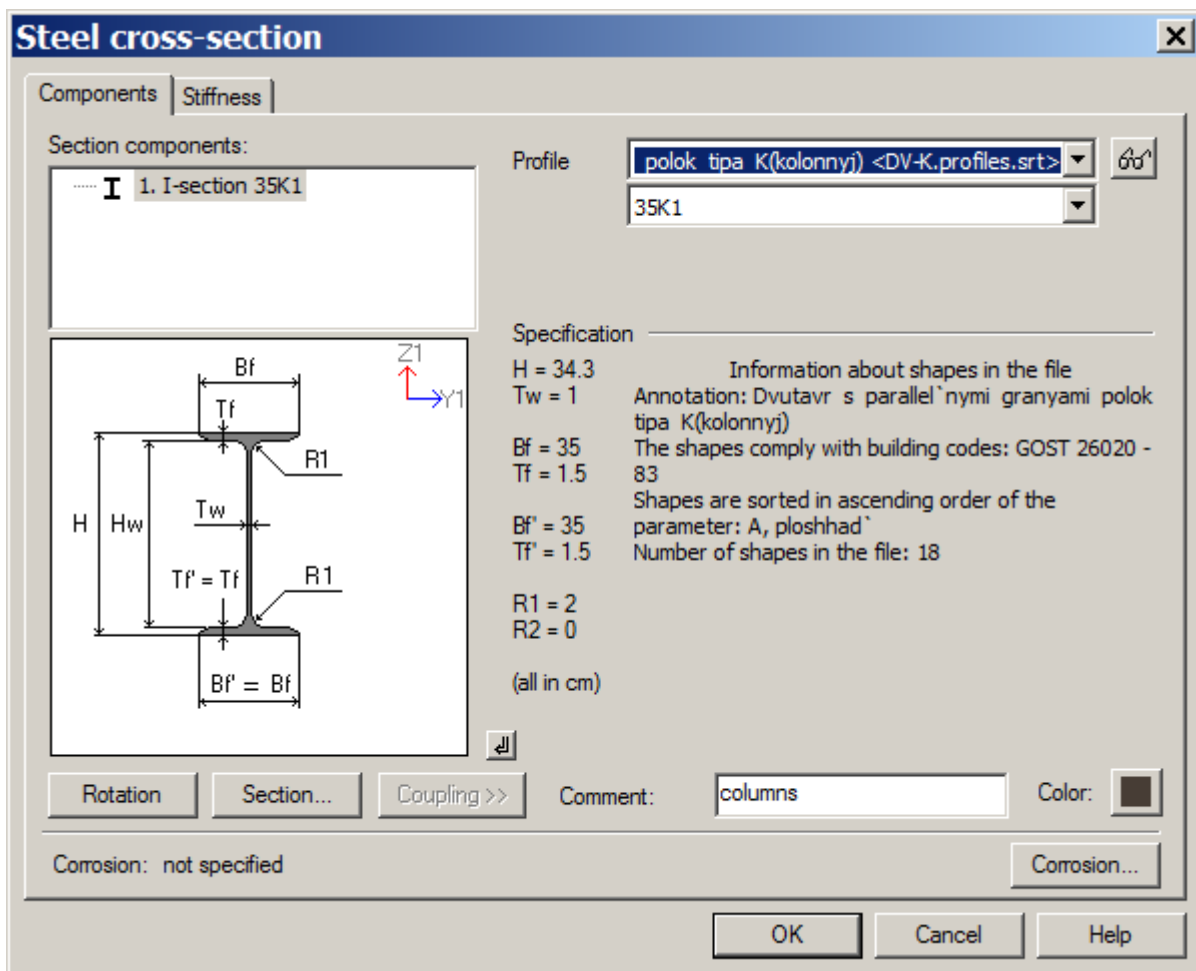


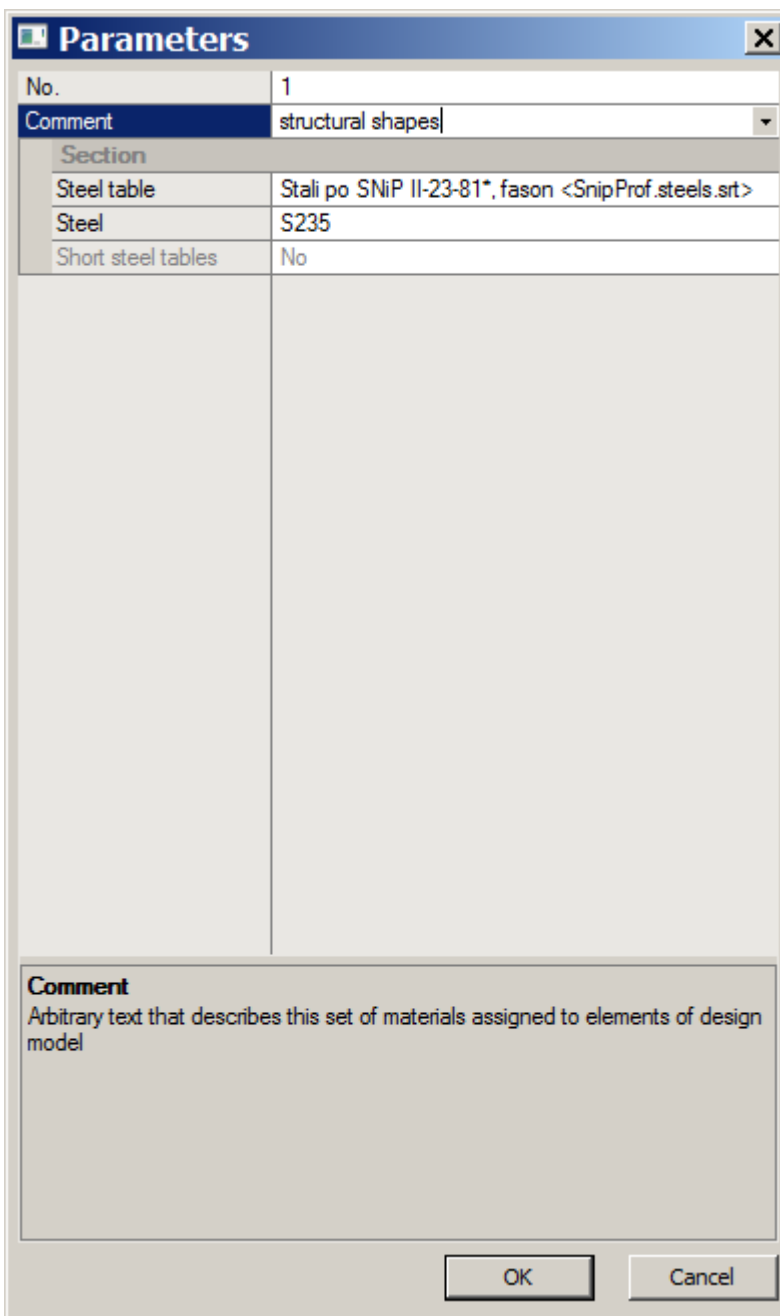
Figure 12.13 Steel cross-section dialog box

- ⇒ In the **Add stiffness** dialog box, double-click **Built-up I-section** icon.
- ⇒ In the new **Steel cross-section** dialog box, define the following data for flanges of the section:
  - in **Section components** area, select **flange**;
  - in the **Profile** list, select 'Prokat listovoj goryachekatanyj tolshhinoj 2.5...25 mm' ;
  - in the next list define the **shape** – 320 x 16.
- ⇒ Define the following data for web of the section:
  - in **Section components** area, select **web**;;
  - in the **Profile** list, select 'Prokat listovoj goryachekatanyj tolshhinoj 2.5...25 mm' ;
  - in the next list define the **shape** – 630 x 8;
  - in the **Comment** box, type **Transverse beams**.
- ⇒ To confirm the data, click **OK**.
- ⇒ In the **Add stiffness** dialog box, double-click **I-section** icon.
- ⇒ In the new **Steel cross-section** dialog box, define the following data for the I-section:
  - Profile – 'Dvutavr s neparallel'nymi granyami polok';
  - Shape – 30;
  - in the **Comment** box, type **Longitudinal beams**.

- ⇒ To confirm the data, click **OK**.
- ⇒ In the **Add stiffness** dialog box, double-click **Double angle** icon.
- ⇒ In the new **Steel cross-section** dialog box, define the following data for the double section:
  - Profile – 'Ugolok ravnopolochnyj';
  - Shape – 75 x 75 x 6;
  - in the **Comment** box, type **Braces**.
- ⇒ To confirm the data, click **OK**.

To define 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.12.14), define the following data:
  - in the **Steel table** list box, define the 'Stali po SNIP II-23-81\*, fason' row;
  - in the **Steel** list box, select grade **S235**;
  - in the **Comments** box, type 'Structural shapes'.
- ⇒ To confirm the data, click **OK**.



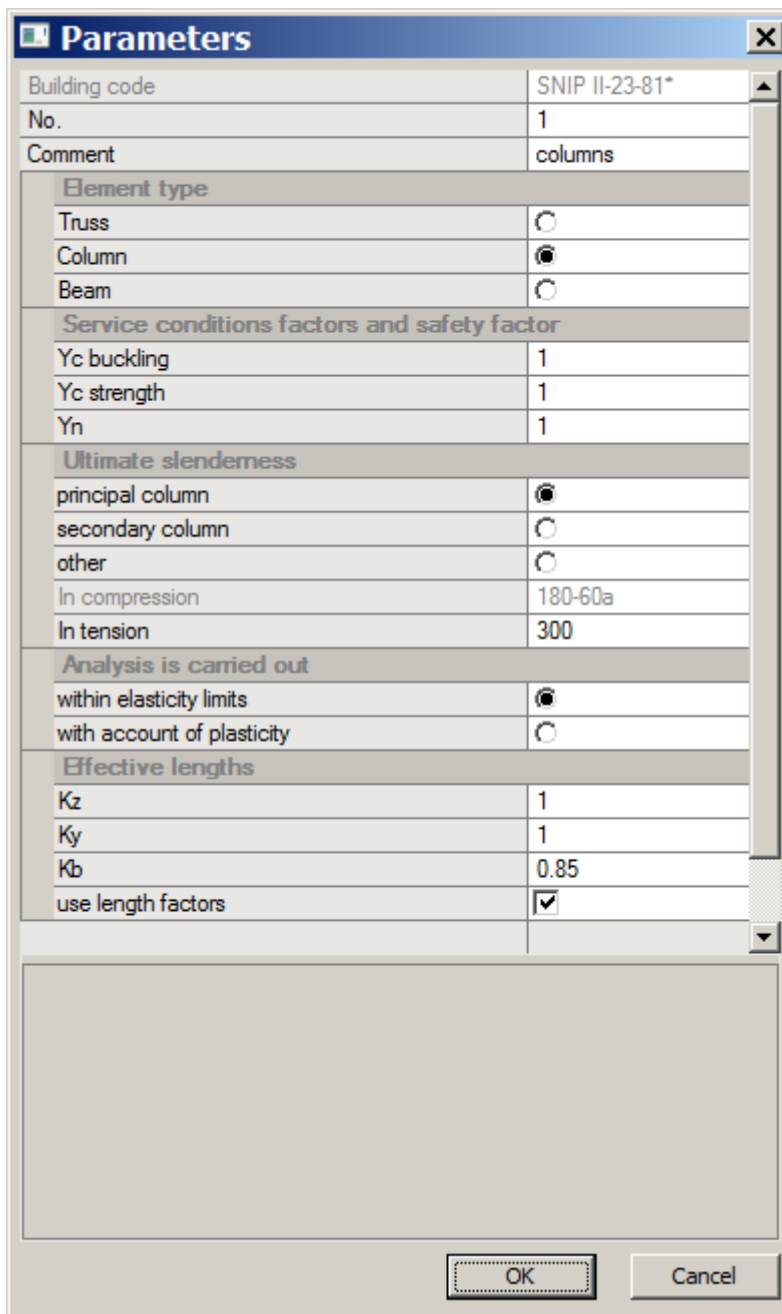
The image shows a 'Parameters' dialog box with a title bar containing a close button. The dialog is divided into several sections. At the top, there are two input fields: 'No.' with the value '1' and 'Comment' with the text 'structural shapes'. Below these is a 'Section' section containing a table with four rows: 'Steel table' with the value 'Stali po SNiP II-23-81\*, fason <SnipProf.steels.srt>', 'Steel' with 'S235', and 'Short steel tables' with 'No'. The fourth row is empty. At the bottom of the dialog is a larger 'Comment' text area with the placeholder text 'Arbitrary text that describes this set of materials assigned to elements of design model'. At the very bottom are 'OK' and 'Cancel' buttons.

No.	1
Comment	structural shapes
<b>Section</b>	
Steel table	Stali po SNiP II-23-81*, fason <SnipProf.steels.srt>
Steel	S235
Short steel tables	No
<b>Comment</b> Arbitrary text that describes this set of materials assigned to elements of design model	

Figure 12.14 **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.12.15), 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**.



The image shows a 'Parameters' dialog box for columns. It contains several sections with input fields and checkboxes.

Parameters (for columns)	
Building code	SNIP II-23-81*
No.	1
Comment	columns
<b>Element type</b>	
Truss	<input type="radio"/>
Column	<input checked="" type="radio"/>
Beam	<input type="radio"/>
<b>Service conditions factors and safety factor</b>	
Yc buckling	1
Yc strength	1
Yn	1
<b>Ultimate slenderness</b>	
principal column	<input checked="" type="radio"/>
secondary column	<input type="radio"/>
other	<input type="radio"/>
In compression	180-60a
In tension	300
<b>Analysis is carried out</b>	
within elasticity limits	<input checked="" type="radio"/>
with account of plasticity	<input type="radio"/>
<b>Effective lengths</b>	
Kz	1
Ky	1
Kb	0.85
use length factors	<input checked="" type="checkbox"/>

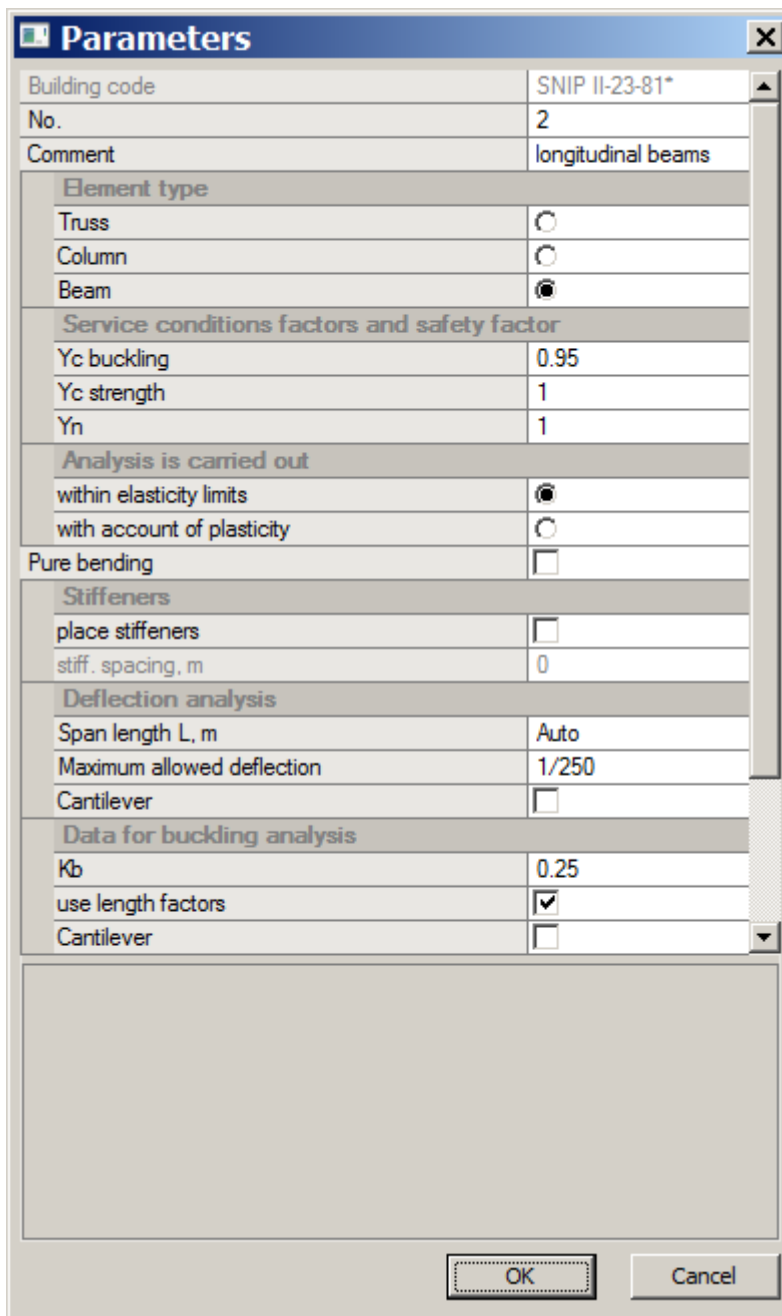
At the bottom of the dialog box are 'OK' and 'Cancel' buttons.

Figure 12.15 **Parameters** (for columns) dialog box

- ⇒ In the **Stiffness and materials** dialog box, click **Add**.
- ⇒ In the **Parameters** dialog box (see Fig.12.16), define the following parameters for longitudinal beams:
- under **Element type**, click **Beam**;
  - in the **Data for buckling analysis** area, select the **use length factors** check box;
  - define effective length factor of beam for buckling analysis  $K_b = 0.25$ ;
  - in the **End conditions of compression flange of beam** list, select the row '2 or more dividing the span to equal parts';
  - in the **Deflection analysis** area, define max allowed deflection –  $1/250$ ;



- in the **Comment** line, type **Longitudinal beams**;
  - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.



The image shows a 'Parameters' dialog box with the following fields and settings:

Building code	SNIP II-23-81*
No.	2
Comment	longitudinal beams
<b>Element type</b>	
Truss	<input type="radio"/>
Column	<input type="radio"/>
Beam	<input checked="" type="radio"/>
<b>Service conditions factors and safety factor</b>	
Yc buckling	0.95
Yc strength	1
Yn	1
<b>Analysis is carried out</b>	
within elasticity limits	<input checked="" type="radio"/>
with account of plasticity	<input type="radio"/>
Pure bending	<input type="checkbox"/>
<b>Stiffeners</b>	
place stiffeners	<input type="checkbox"/>
stiff. spacing, m	0
<b>Deflection analysis</b>	
Span length L, m	Auto
Maximum allowed deflection	1/250
Cantilever	<input type="checkbox"/>
<b>Data for buckling analysis</b>	
Kb	0.25
use length factors	<input checked="" type="checkbox"/>
Cantilever	<input type="checkbox"/>

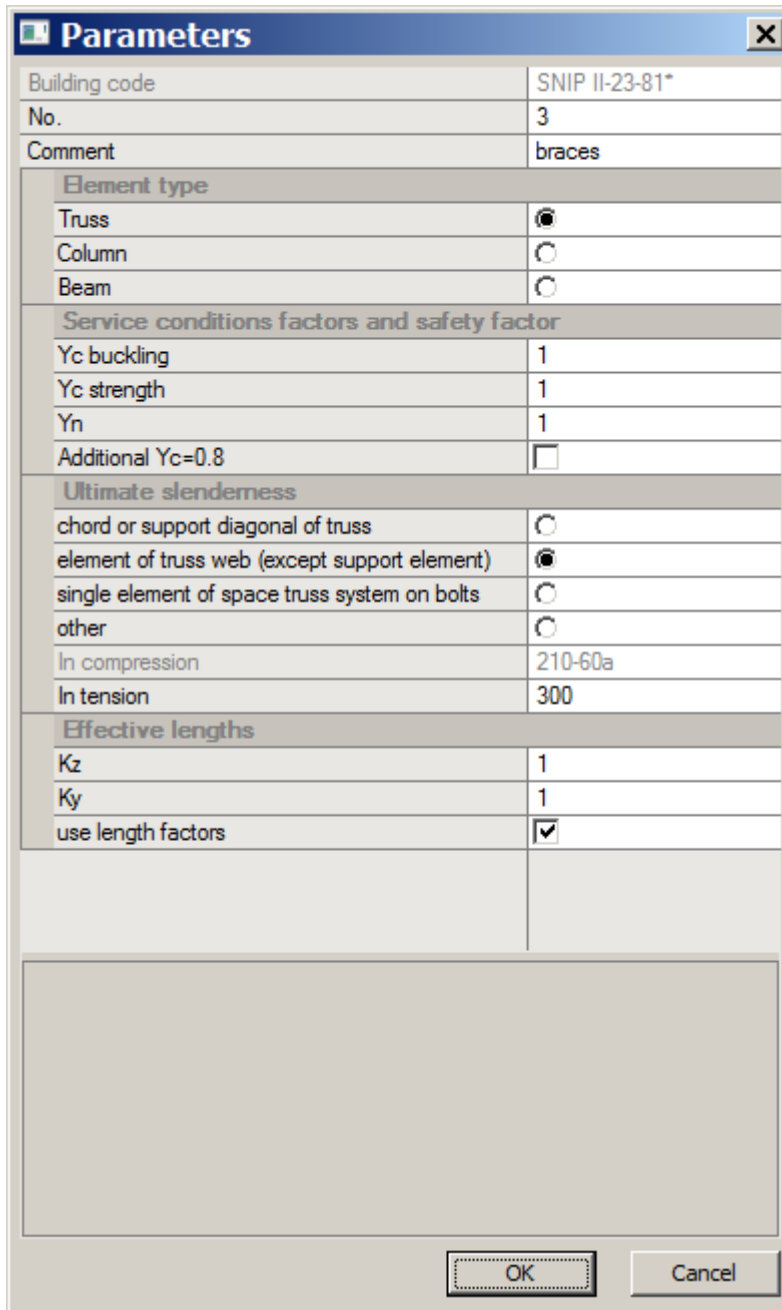
At the bottom of the dialog box are 'OK' and 'Cancel' buttons.

Figure 12.16 **Parameters** (for longitudinal beams) dialog box

- ⇒ In the **Stiffness and materials** dialog box, click **Add** once more.
- ⇒ In the **Parameters** dialog box (see Fig.12.17), define the following parameters for braces:
- in the **Element type** area, click **Truss**;
  - in the Effective lengths area, select the **use length factors** check box;
  - effective length factor relative to Z1-axis  $K_z = 1$ ;

- effective length factor relative to Y1-axis  $K_y = 1$ ;
- in the **Ultimate slenderness** area, select 'element of truss web (except support element)';
- in the **Comment** line, type **Braces**;
- other parameters remain by default.

⇒ To confirm the data, click **OK**.



The image shows a 'Parameters' dialog box with the following fields and values:

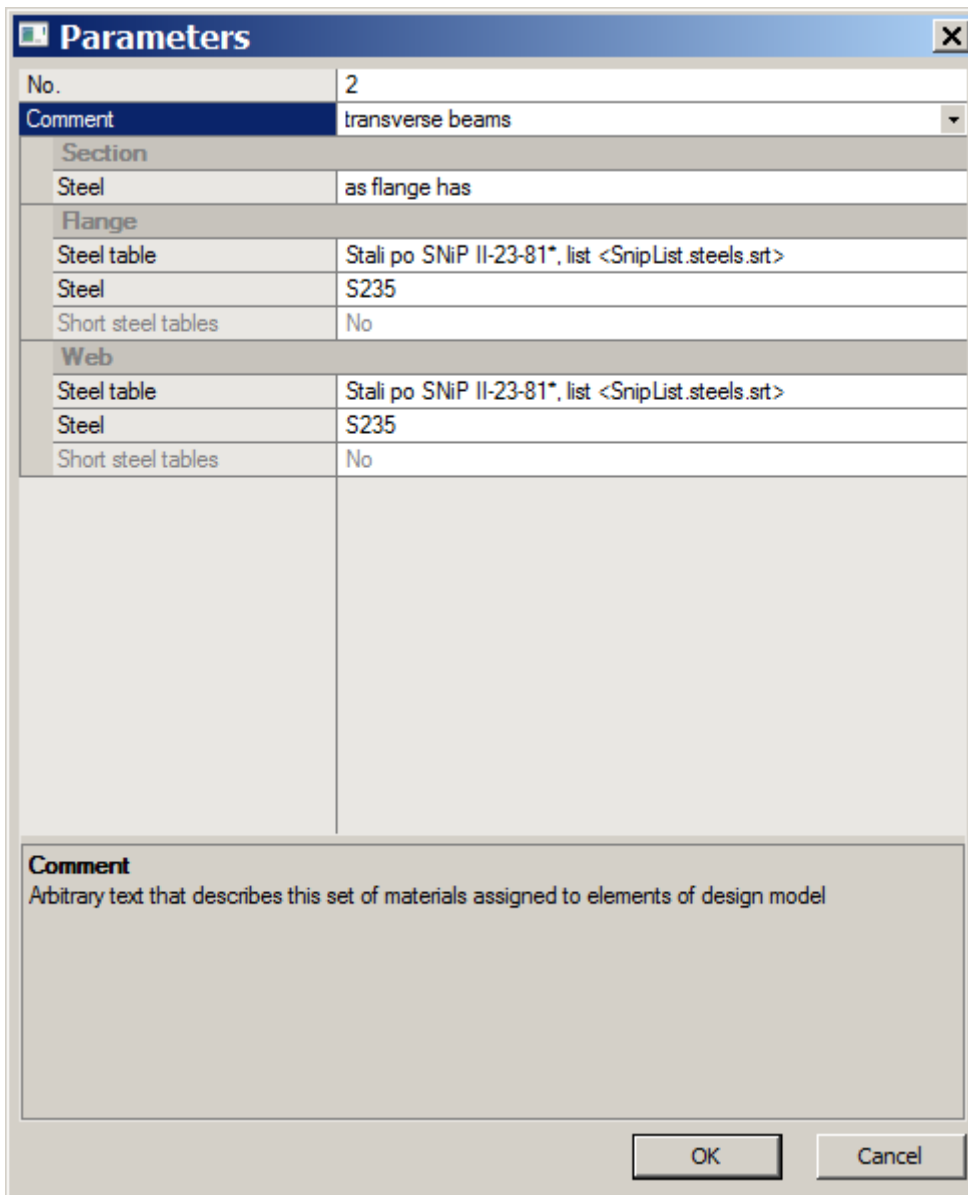
Building code	SNIP II-23-81*
No.	3
Comment	braces
<b>Element type</b>	
Truss	<input checked="" type="radio"/>
Column	<input type="radio"/>
Beam	<input type="radio"/>
<b>Service conditions factors and safety factor</b>	
Yc buckling	1
Yc strength	1
Yn	1
Additional Yc=0.8	<input type="checkbox"/>
<b>Ultimate slenderness</b>	
chord or support diagonal of truss	<input type="radio"/>
element of truss web (except support element)	<input checked="" type="radio"/>
single element of space truss system on bolts	<input type="radio"/>
other	<input type="radio"/>
In compression	210-60a
In tension	300
<b>Effective lengths</b>	
Kz	1
Ky	1
use length factors	<input checked="" type="checkbox"/>

At the bottom of the dialog box are 'OK' and 'Cancel' buttons.

Figure 12.17 **Parameters** (for braces) dialog box

- ⇒ In the **Stiffness and materials** dialog box, select the first tab **Properties**. In the **List of stiffness types**, select type '2. Builtup I-section' 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.

- ⇒ Then, to assign materials for steel structures, in the **Stiffness and materials** dialog box, select the **Steel** tab (the third tab).
- ⇒ Select **Additional parameters** option and click **Add**.
- ⇒ In the **Parameters** dialog box, define the following data for transverse 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=0.25**;
  - 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;
  - under **Stiffeners**, select the **Place stiffeners** check box and define spacing for stiffeners as equal to 1 m;
  - in the **Comment** line, type **Transverse beams**;
  - other parameters remain by default.
- ⇒ To confirm the data, click **OK**.
  
- ⇒ In the **Stiffness and materials** dialog box, select **Material** option and click **Add**.
- ⇒ In the **Parameters** dialog box (see Fig.12.18), define the following data for transverse beams:
  - under **Section**, in the **Steel** list, select 'as flange has';
  - under **Flange**, in the **Steel table** drop-down list, select 'Stali po SNIP II-23-81\*', list' and then in the **Steel** list, select the steel grade S235;
  - under **Web**, in the **Steel table** drop-down list, select 'Stali po SNIP II-23-81\*', list' and then in the **Steel** list, select the steel grade S235;
  - in the **Comments** box, type 'Transverse beams'.
- ⇒ To confirm the data, click **OK**.






The dialog box is titled "Parameters" and contains the following fields and sections:








No.	2
Comment	transverse beams
<b>Section</b>	
Steel	as flange has
<b>Flange</b>	
Steel table	Stali po SNiP II-23-81*, list <SnipList.steels.srt>
Steel	S235
Short steel tables	No
<b>Web</b>	
Steel table	Stali po SNiP II-23-81*, list <SnipList.steels.srt>
Steel	S235
Short steel tables	No
<b>Comment</b> Arbitrary text that describes this set of materials assigned to elements of design model	

At the bottom right, there are "OK" and "Cancel" buttons.


Figure 12.18 **Parameters** (for transverse beams) dialog box



To assign material properties to elements of the model:

- ⇒ 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 orientation of FE** check box and specify option **II X**.
- ⇒ Click **Apply** .
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** . In this case, make sure that stiffness **2.Builtup I-section** is defined as current one in the list of stiffness types and in the list of current materials the following data should be defined as current one: material – **2.transverse beams**, additional parameters – **4.transverse beams**.
- ⇒ In the **Warning** box, click **No**. (The elements become unselected. It indicates that the current stiffness type is assigned to selected elements.)

- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, specify option **II Y** in the **By orientation of FE** area.
- ⇒ Click **Apply** .
- ⇒ In the **Stiffness and materials** dialog box, in the list of parameters for steel structures (see **Steel** tab), select the row '1.structural shapes'.
- ⇒ Click **Set as current one**. 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.
- ⇒ Click **Additional parameters** option, then select the row '2.longitudinal beams' and click **Set as current one**.
- ⇒ In the **Stiffness and materials** dialog box, select the first tab **Properties**. In the **List of stiffness types**, select type '3. I-section 30' and click **Set as current type**.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ The **Warning** box is displayed. Click **No**.
- ⇒ In the **PolyFilter** dialog box, on the **Filter for elements** tab, specify option **I** (for inclined elements) in the **By orientation of FE** area.
- ⇒ Click **Apply** .
- ⇒ In the **Stiffness and materials** dialog box, select the first tab **Properties**. In the **List of stiffness types**, select type '4. Double angle 75 x 75 x 6' and click **Set as current type**.
- ⇒ In the same dialog box, in the list of parameters for steel structures (see **Steel** tab), click **Additional parameters** option.
- ⇒ Select the row '3.braces' in the list and click **Set as current one**.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ On the **Select** toolbar, click **Select vertical bars** button .
- ⇒ Select all vertical elements of the model with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, set as current ones the additional parameters for steel structures as '1.columns' and stiffness type as '1.I-sectionh 35K1'.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ On the **Select** toolbar, click **Select vertical bars** button  once again in order to make this command not active.

## Step 5. Changing FE types for elements of braces

- ⇒ In the **Stiffness and materials** dialog box, clear the **Materials** check box in the **Assign to elements of the model** area.
- ⇒ Select the stiffness type '4. Double angle 75x75x6' as the current type of stiffness.
- ⇒ In the same dialog box, to select elements with this stiffness type on the model, click the **Select on model** button .

- ⇒ On the **Advanced edit options** ribbon tab, on the **Model** panel, click **Change FE type** .
- ⇒ In the **Change FE type** dialog box (see Fig.12.19), in the list of FE types, select **FE type 4 - FE of 3D truss**.
- ⇒ Click **Apply** .

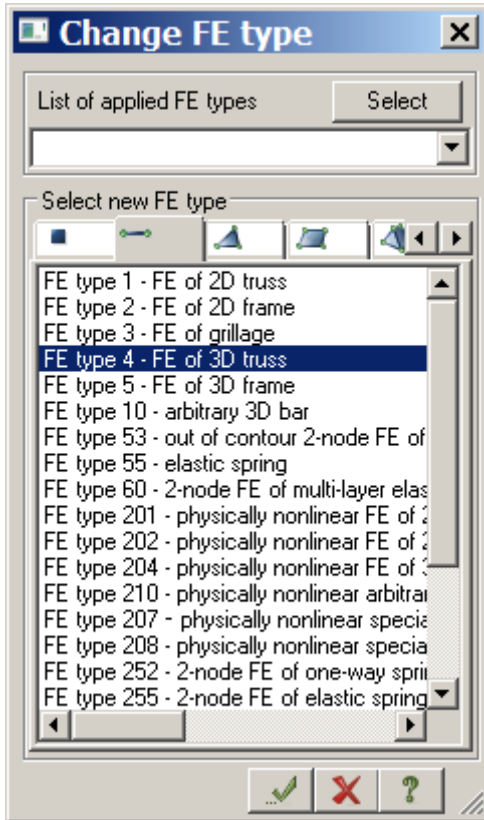




Figure 12.19 **Change FE type** dialog box

## 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** .
- ⇒ In the **Add dead weight** dialog box (see Fig.12.20), 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).

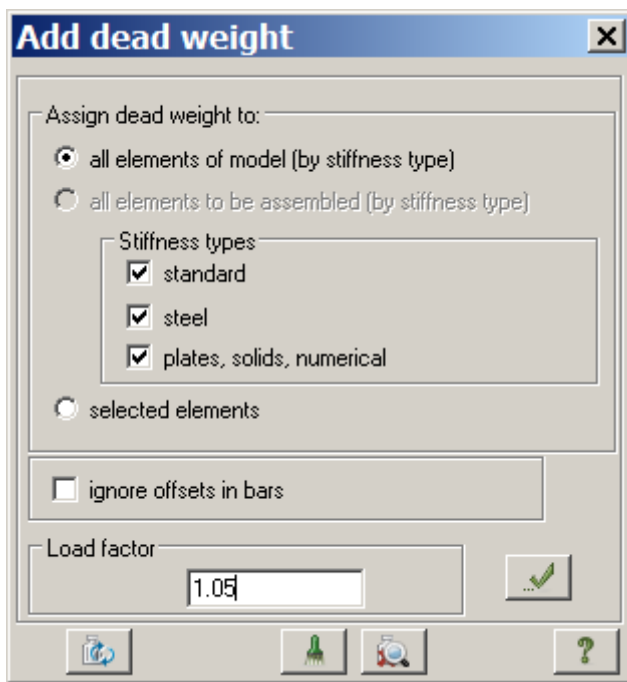






Figure 12.20 Add dead weight dialog box

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 **Projection** toolbar (by default, it is displayed at the bottom of the screen), click **Projection on XOZ-plane** .
- ⇒ On the **Select** toolbar, click **Select horizontal bars** button . Then select (with selection window - from left to right) elements of the central row of columns.
- ⇒ 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.12.21), specify **Global** coordinate system and direction along the **Z-axis** (default parameters).

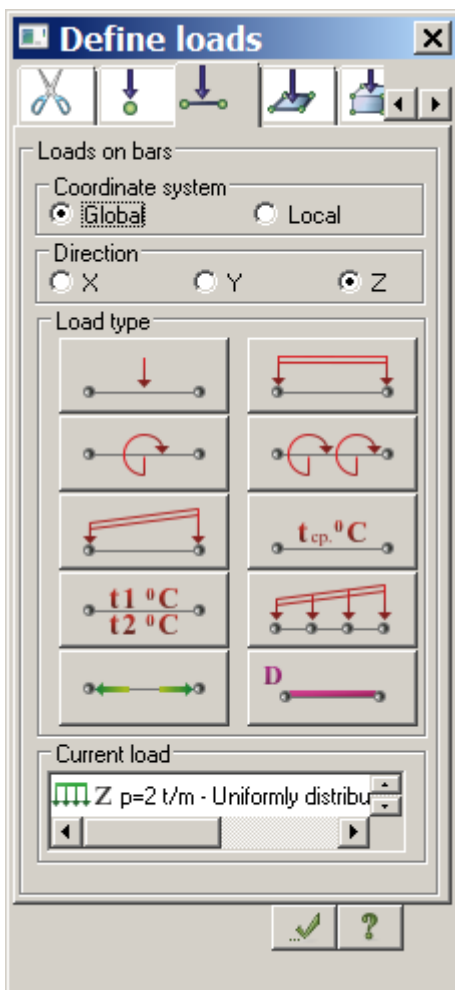




Figure 12.21 Define loads dialog box

- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify  $P = 0.9 \text{ t/m}$  (see Fig.12.22).
- ⇒ Click **OK** .

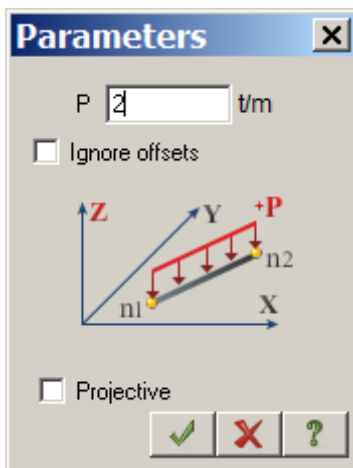







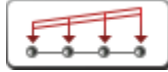


Figure 12.22 Load parameters dialog box



- ⇒ Select elements of extreme rows of columns with the help of selection window (from left to right).
- ⇒ In the **Load type** area, click **Uniformly distributed load** button  once more.
- ⇒ In the **Load parameters** dialog box specify  $P = 0.45 \text{ t/m}$ .
- ⇒ Click **OK** .
- ⇒ To present design model in projection on the YOZ-plane, on the **Projection** toolbar, click **Projection on YOZ-plane** .
- ⇒ Select (in turn) elements of middle rows of columns with the help of selection window (from left to right).
- ⇒ Define for these elements uniformly distributed load  $p = 1.44 \text{ t/m}$ .
- ⇒ Then select elements of extreme rows of columns with the help of selection window (from left to right).
- ⇒ Define for these elements uniformly distributed load  $p = 0.72 \text{ t/m}$ .
- ⇒ On the **Select** toolbar, click **Select horizontal bars** button  once again in order to make this command not active.

#### 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.
- ⇒ On the **Projection** toolbar (by default, it is displayed at the bottom of the screen), click **Projection on XOZ-plane** .
- ⇒ On the **Select** toolbar, click **Select vertical bars** button . Then select extreme left row of columns.
- ⇒ In the **Define loads** dialog box, on the **Load on bars** tab, specify direction along the **X**-axis.
- ⇒ In the **Load type** area, click **Trapezoidal load on group of bars** button .
- ⇒ In the **Non-uniformly distributed load** dialog box (see Fig.12.23), specify  $P1 = -0.065 \text{ t/m}$ ,  $P2 = -0.1 \text{ t/m}$  and direction along which the load is changed (Z-axis).
- ⇒ Click **OK**.

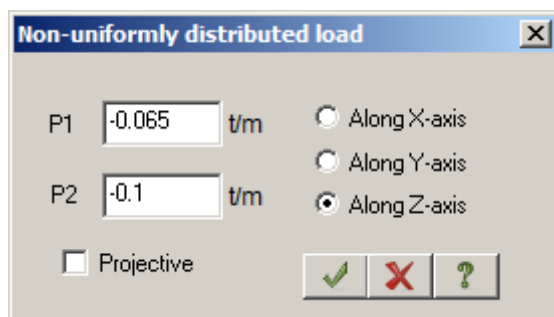
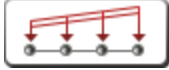


Figure 12.23 **Non-uniformly distributed load** dialog box

- ⇒ Select (with selection window - from left to right) elements of the right row of columns.

- ⇒ In the **Define loads** dialog box, in the **Load type** area, click **Trapezoidal load on group of bars** button







- ⇒ In the **Non-uniformly distributed load** dialog box, specify  $P1 = -0.05 \text{ t/m}$ ,  $P2 = -0.075 \text{ t/m}$  and direction along which the load is changed (Z-axis).
- ⇒ Click **OK**.

- ⇒ On the **Select** toolbar, click **Select vertical bars** button  once again in order to make this command not active.

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

To define detailed information about load cases:

- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel and click **Edit load cases** . The **Edit load cases** dialog box is displayed on the screen (see Fig.12.24).
- ⇒ 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** .

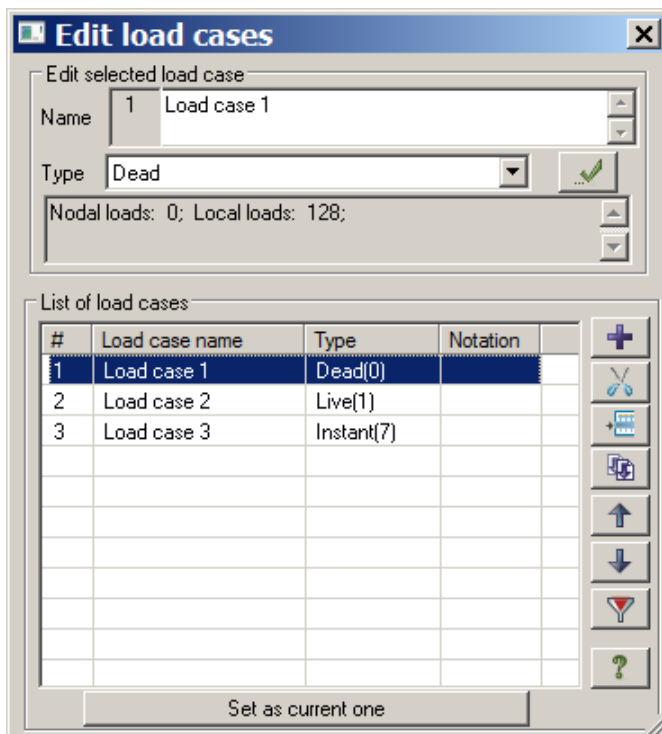




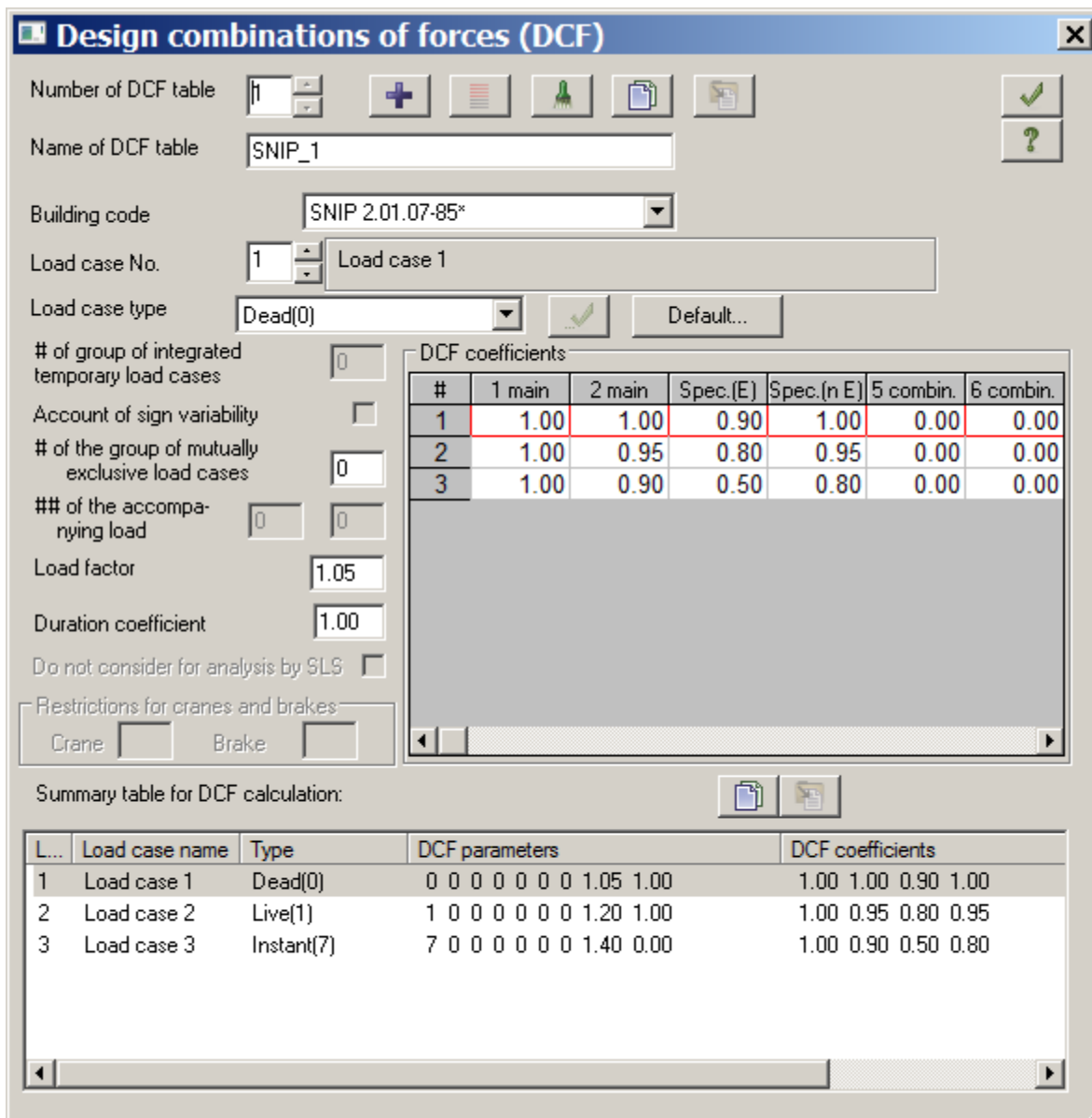


Figure 12.24 **Edit load cases** dialog box







- ⇒ Close the **Edit load cases** dialog box.


## Step 7. Generating DCF table

- ⇒ On the **Analysis** ribbon tab, select the **DCF** panel and click **DCF table** button .
- ⇒ In the **Design combinations of forces** dialog box (see Fig.12.25), to generate DCF table with values accepted by default for every load case, click **Default DCF values** button .
- ⇒ Then make sure that building code **SNIP 2.01.07-85\*** is selected and specify the following data:
  - for Load case 1 – in the **Summary table for DCF calculation**, define **Load factor** as equal to 1.05 and then click **Apply** .
- ⇒ Click **OK** .




**Design combinations of forces (DCF)**

Number of DCF table:       

Name of DCF table:  

Building code:

Load case No.:  Load case 1

Load case type:  

# of group of integrated temporary load cases:

Account of sign variability: ☐

# of the group of mutually exclusive load cases:

## of the accompanying load:

Load factor:

Duration coefficient:

Do not consider for analysis by SLS: ☐

Restrictions for cranes and brakes:  
Crane ☐ Brake ☐

DCF coefficients


#	1 main	2 main	Spec.(E)	Spec.(n E)	5 combin.	6 combin.
1	1.00	1.00	0.90	1.00	0.00	0.00
2	1.00	0.95	0.80	0.95	0.00	0.00
3	1.00	0.90	0.50	0.80	0.00	0.00

Summary table for DCF calculation:

L...	Load case name	Type	DCF parameters	DCF coefficients
1	Load case 1	Dead(0)	0 0 0 0 0 0 1.05 1.00	1.00 1.00 0.90 1.00
2	Load case 2	Live(1)	1 0 0 0 0 0 1.20 1.00	1.00 0.95 0.80 0.95
3	Load case 3	Instant(7)	7 0 0 0 0 0 1.40 0.00	1.00 0.90 0.50 0.80



Figure 12.25 Design combinations of forces dialog box

## Step 8. Defining design sections for beams

- ⇒ On the **Select** toolbar, click **Select horizontal bars** button .
- ⇒ Select all horizontal elements of the model with the pointer.



*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**, **Advanced 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.12.26), specify number of sections  $N = 5$ .
- ⇒ Click **Apply**  (to carry out analysis on serviceability limit states, it is necessary to calculate forces in three or more sections) and close the dialog box.

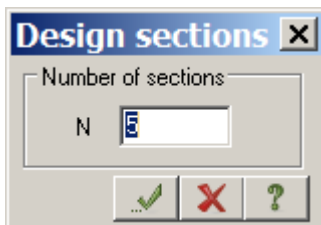




Figure 12.26 **Design sections** dialog box

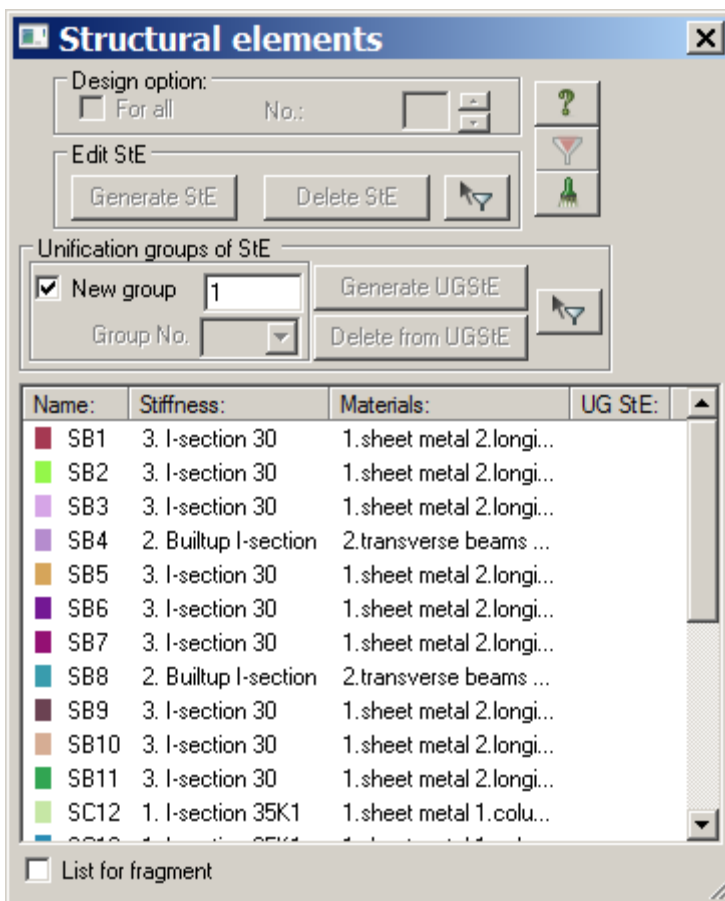
## Step 9. Defining structural elements



*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. In this version it is possible to select all elements of the model and unite them into structural ones.*



To define structural element BEAM:

- ⇒ On the **Select** toolbar, click **Select horizontal bars** button .
- ⇒ Select with the pointer horizontal elements to which braces adjoin (No. 77 - 98).
- ⇒ 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.12.27), under **Edit structural elements (StE)**, click **Generate StE**.



Figure 12.27 **Structural elements** dialog box

- ⇒ On the **Select** toolbar, click **Select horizontal bars** button  once again in order to make this command not active.

To define structural element COLUMN:

- ⇒ On the **Select** toolbar, click **Select vertical bars** button .
- ⇒ In the **Structural elements** dialog box, under **Edit structural elements (StE)**, click **Generate StE**.
- ⇒ On the **Select** toolbar, click **Select vertical bars** button  once again in order to make this command not active.

## Step 10. Defining deflection fixities at nodes of bending elements

- ⇒ On the **Select** toolbar, click **Select horizontal bars** button .
- ⇒ Select all horizontal elements of the model with the pointer.
- ⇒ On the **More edit options** ribbon tab, on the **Design** panel, click **Deflection fixities** command .
- ⇒ In the **Deflection fixities** dialog box (see Fig.12.28), select **Create at ends of structural elements** option in the list.

- ⇒ Then select Y1 and Z1 check boxes and click **Apply** (deflection of element sections is determined relative to the line that connects fixities at the ends of the element).

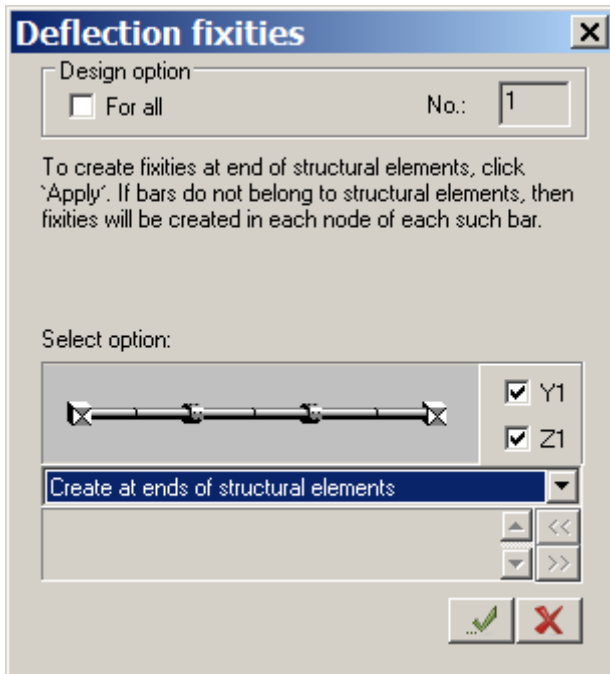


Figure 12.28 **Deflection fixities** dialog box

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

## Step 11. Complete analysis of the model

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



## Step 12. Review and evaluation of static analysis results





*When 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.

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  $Q_z$ , on the **Results** tab, select **Forces in bars** panel and click **Shear force diagrams** ( $Q_z$ ) 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 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 generate and review tables of analysis results:

- ⇒ To present table with design combinations of forces in elements 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.12.29), select **DCF, design values** in the list.
- ⇒ Click **Apply**.

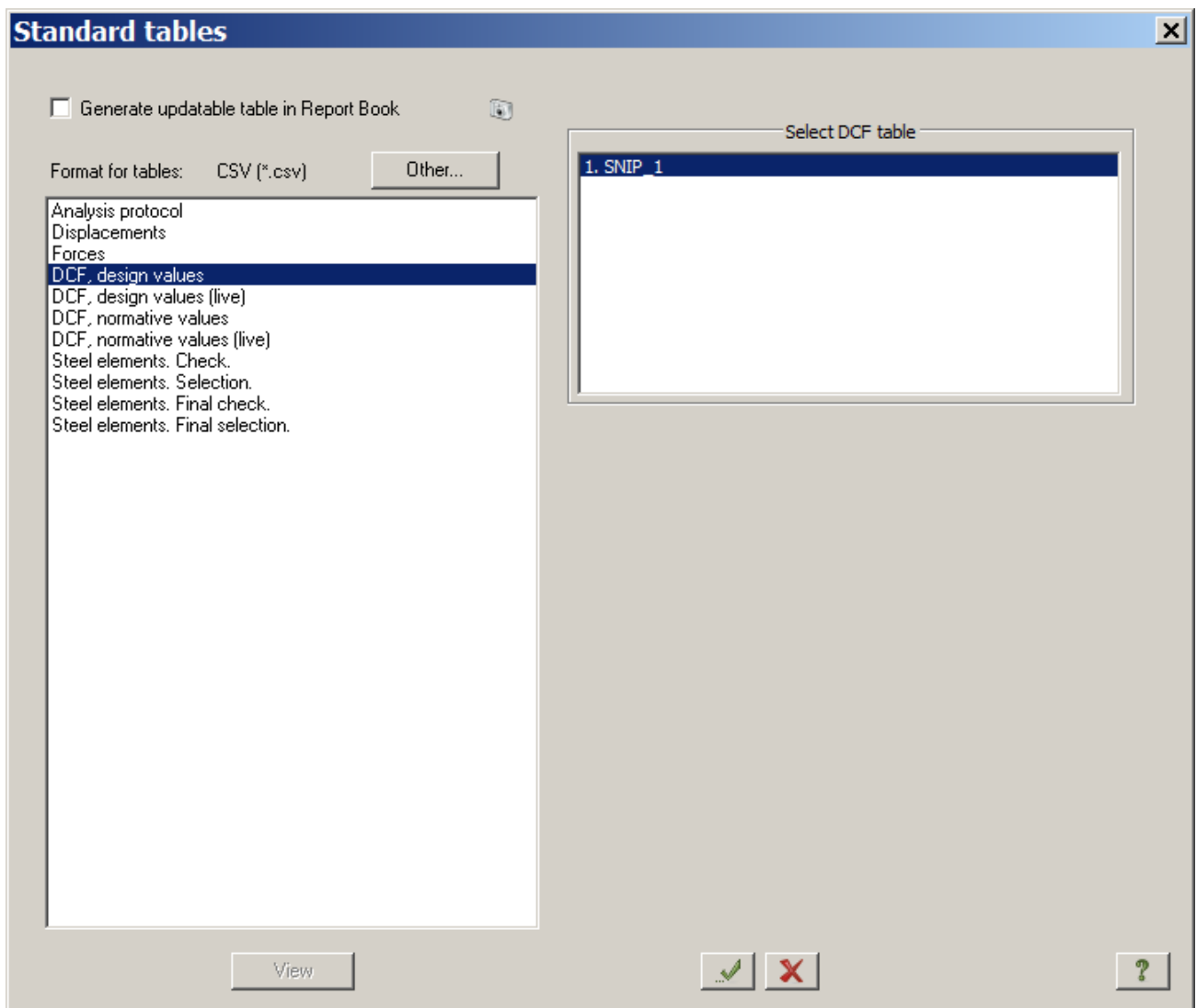


*By default, standard tables are generated in the \*.csv format. Information presented in these tables is divided into different tabs: input data (optional), e.g. DCF coefficients; output data for bars; output data for plates; etc.*

*To generate table in \*.csv format and add it to the Report Book, select the **Generate updatable table in Report Book** check box. If the table is located in the Report Book, it is possible to update it later (if required) and add it to the report file with the Report Book options.*

*To modify format of the table, in the **Standard tables** dialog box, click **Select format**. Then in the **Table format** dialog box, select appropriate option and click **OK**. To generate table in [Document Maker \(DOC-SAPR module\)](#), select RPT format.*

*Selected format is saved and will be applied by default in further work with standard tables.*

Figure 12.29 **Standard tables** dialog box




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

### Step 13. 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 serviceability limit state), on the **Design** ribbon tab, on the **Steel: check and select** panel, click **Check, SLS** 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 .




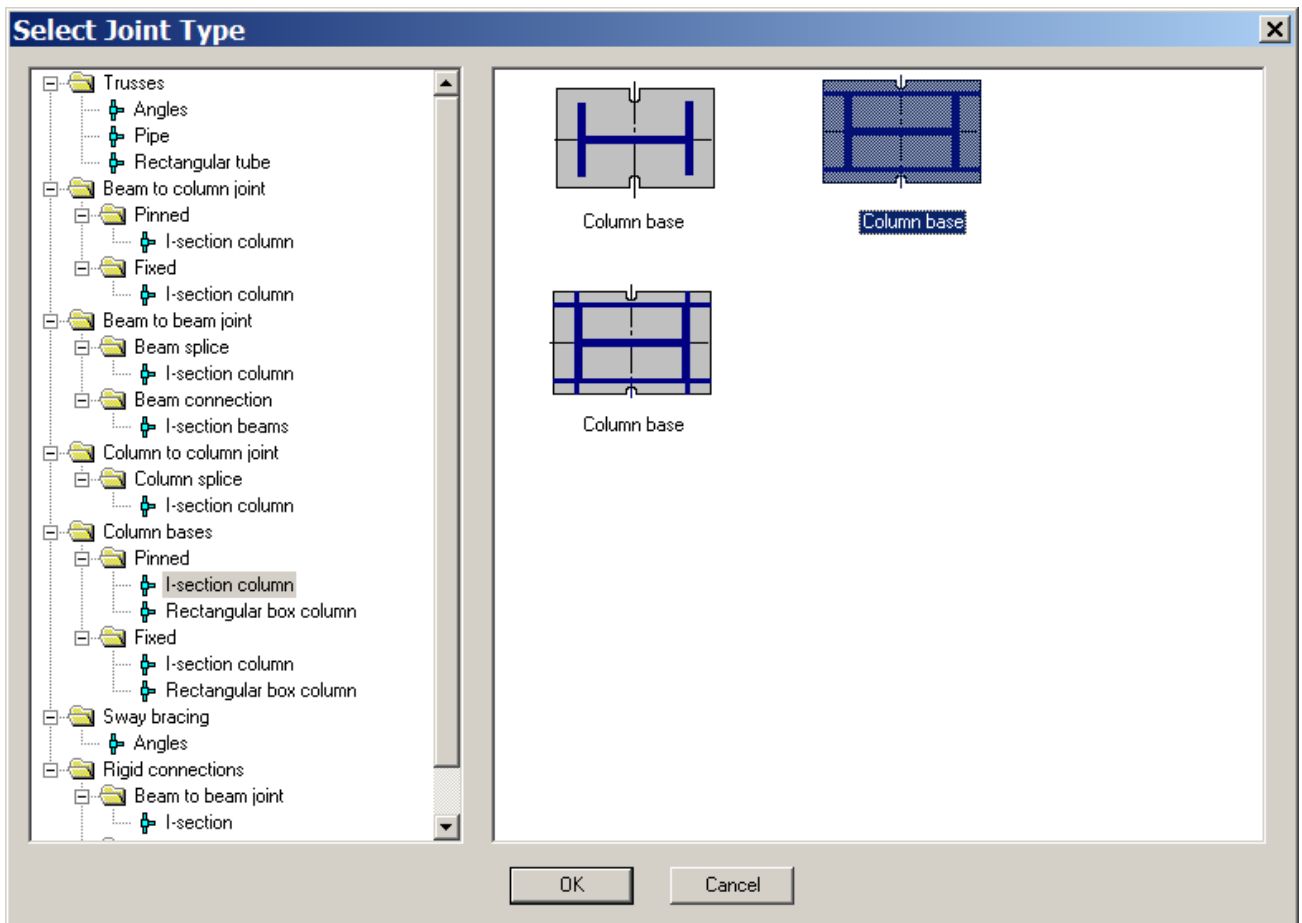
To generate the table with results (assigned sections):

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

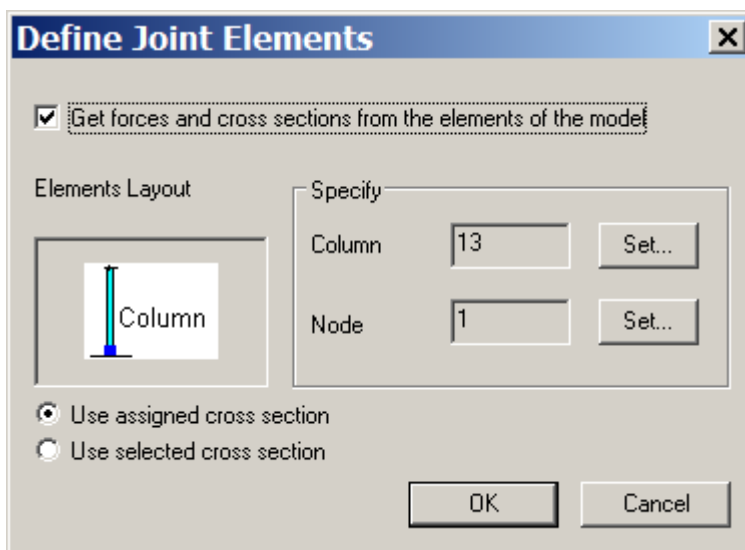
## Step 14. Analysis of joints


To carry out analysis of column base:

- ⇒ On the **Design** tab, select **Steel: analysis** panel and click **Analyse joint** button .
- ⇒ In the **Select joint type** dialog box, select **Column bases / Pinned / I-section column** in the list.
- ⇒ In the right part of the dialog box, select the type of joint as presented in the figure (see Fig.12.30).
- ⇒ Click **OK**.

Figure 12.30 **Select joint type** dialog box

- ⇒ In the **Define joint elements** dialog box (see Fig.12.31), click the **Use assigned cross section** option.
- ⇒ To define column number, click **Set** in the **Column** line. The **Select element** dialog box appears on the screen.

Figure 12.31 **Define joint elements** dialog box

- ⇒ In the **Select element** dialog box (see Fig.12.32), select the **define on model** check box, select with the pointer column No. 13 (its number will be displayed in the dialog box).
- ⇒ Click **OK**  (the program returns to the **Define joint elements** dialog box and column number will be displayed in the appropriate box).

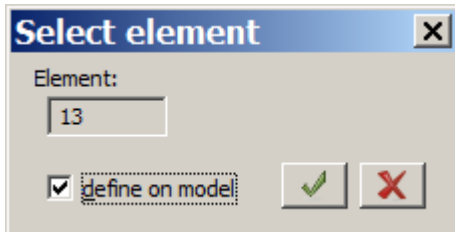






Figure 12.32 **Select element** dialog box

- ⇒ To define node number, click **Set** in the **Node** line. The **Select node** dialog box appears on the screen.
- ⇒ In the **Select node** dialog box, select the **define on model** check box, select with the pointer node No. 1 (its number will be displayed in the dialog box).
- ⇒ Click **OK**  (the program returns to the **Define joint elements** dialog box and node number will be displayed in the appropriate box).
- ⇒ To analyse selected joint, in the **Define joint elements** dialog box, click **OK**.
- ⇒ Then the program will be opened in the mode of analysis of joint. In this mode, when you evaluate analysis results for column base joint, save the problem file and close this mode window.

#### To design and analyse compound joints:

- ⇒ To create and analyse compound joints, on the **Design** tab, select **Steel: analysis** panel and click **Compound joint** button  in the drop-down list.
- ⇒ In the **Compound joints** dialog box, click **Add joint** button (the first row **Compound joint I** will appear in the list of the dialog box).
- ⇒ To add the first connection of compound joint, click **Add**.
- ⇒ In the **Select joint type** dialog box, in the tree-like list, for pinned connection beam-to-column, select column section as **I-section**.
- ⇒ In the right part of the dialog box, double-click the joint with seat angle.
- ⇒ In the new **Define joint elements** dialog box, click the **Use assigned cross section** option.
- ⇒ To define beam number, click **Set** in the **Beam** line. The **Select element** dialog box appears on the screen.
- ⇒ In the **Select element** dialog box, select the **define on model** check box, select with the pointer beam No. 25 (its number will be displayed in the dialog box).
- ⇒ Click **OK**  (the program returns to the **Define joint elements** dialog box and beam number will be displayed in the appropriate box).
- ⇒ To define column number, click **Set** in the **Column** line. The **Select element** dialog box appears on the screen.
- ⇒ In the **Select element** dialog box, select the **define on model** check box, select with the pointer column No. 40 (its number will be displayed in the dialog box).
- ⇒ Click **OK**  (the program returns to the **Define joint elements** dialog box and column number will be displayed in the appropriate box).

- ⇒ To analyse selected joint, in the **Define joint elements** dialog box, click **OK**. The program returns to the **Compound joints** dialog box where the second row **Beam-to-column connection** appears in the list.

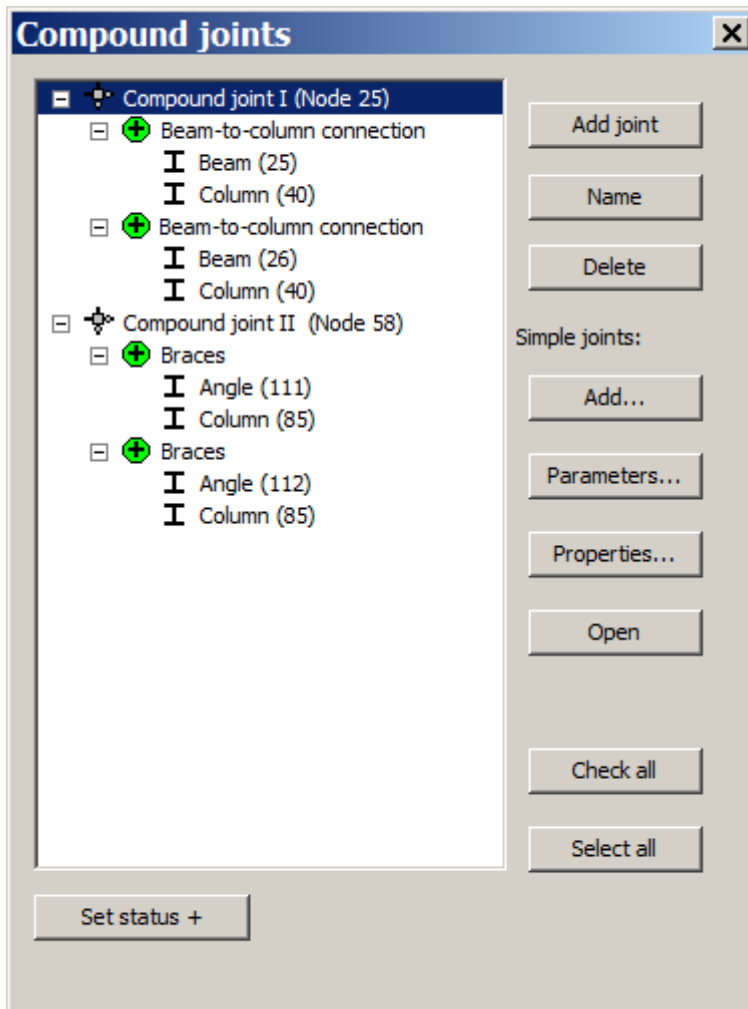









Figure 12.33 **Compound joints** dialog box

- ⇒ To add the second connection of compound joint, in the **Compound joints** dialog box, click **Add**.
- ⇒ In the **Select joint type** dialog box, in the tree-like list, for pinned connection beam-to-column, select column section as **I-section**.
- ⇒ In the right part of the dialog box, double-click the joint with seat angle.
- ⇒ In the new **Define joint elements** dialog box, click the **Use assigned cross section** option.
- ⇒ To define beam number, click **Set** in the **Beam** line.
- ⇒ In the **Select element** dialog box, select the **define on model** check box, select with the pointer beam No. 26 (its number will be displayed in the dialog box).
- ⇒ Click **OK** .
- ⇒ To define column number, click **Set** in the **Column** line.
- ⇒ In the **Select element** dialog box, select the **define on model** check box, select with the pointer column No. 40.
- ⇒ Click **OK** .

- ⇒ To analyse selected joint, in the **Define joint elements** dialog box, click **OK**.
  
- ⇒ In the **Compound joints** dialog box, click **Add joint** button (the new row **Compound joint II** will appear in the list of the dialog box).
- ⇒ To add the first connection of compound joint, click **Add**.
- ⇒ In the **Select joint type** dialog box, in the tree-like list, for sway bracing, select angular section.
- ⇒ In the right part of the dialog box, double-click **Mixed joint** icon.
- ⇒ In the new **Define joint elements** dialog box, click the **Use assigned cross section** option.
- ⇒ To define number for the first element, click **Set** in the **Element 1** line. The **Select element** dialog box appears on the screen.
- ⇒ In the **Select element** dialog box, select the **define on model** check box, select with the pointer element of bracing No. 111 (its number will be displayed in the dialog box).
- ⇒ Click **OK** .
- ⇒ To define number for the second element, click **Set** in the **Element 2** line.
- ⇒ In the **Select element** dialog box, select the **define on model** check box, select with the pointer beam No. 85.
- ⇒ Click **OK** .
  
- ⇒ To add the second connection of compound joint, in the **Compound joints** dialog box, click **Add**.
- ⇒ In the **Select joint type** dialog box, in the tree-like list, for sway bracing, select angular section.
- ⇒ In the right part of the dialog box, double-click **Mixed joint** icon.
- ⇒ In the new **Define joint elements** dialog box, click the **Use assigned cross section** option.
- ⇒ To define number for the first element, click **Set** in the **Element 1** line.
- ⇒ In the **Select element** dialog box, select the **define on model** check box, select with the pointer element of bracing No. 112.
- ⇒ Click **OK** .
- ⇒ To define number for the second element, click **Set** in the **Element 2** line.
- ⇒ In the **Select element** dialog box, select the **define on model** check box, select with the pointer beam No. 85.
- ⇒ Click **OK** .
  
- ⇒ To analyse selected joint, in the **Define joint elements** dialog box, click **OK**.
  
- ⇒ If every connection is denoted with plus sign  , it means that the joint is designed properly.
- ⇒ Close the **Compound joints** dialog box.