

Example 1. Analysis of 2D frame

In this lesson you will learn how to:

- generate design model of 2D frame;
- deal with design options;
- fill in the DCF table and Load Editor table;
- determine reinforcement for elements of frame;
- design continuous beam;
- design column.

Description:

Model of the frame and its boundary conditions are presented in Fig.1.1.

Sections for elements of the frame are presented in Fig.1.2.

Material for frame – reinforced concrete B30.

Loads:

- dead uniformly distributed load $g_1 = 2.0 \text{ t/m}$;
- dead uniformly distributed load $g_2 = 1.5 \text{ t/m}$;
- dead uniformly distributed load $g_3 = 3.0 \text{ t/m}$;
- live uniformly distributed load $g_4 = 4.67 \text{ t/m}$;
- live uniformly distributed load $g_5 = 2.0 \text{ t/m}$;
- wind load (from the left) $P_1 = -1.0 \text{ t}$;
- wind load (from the left) $P_2 = -1.5 \text{ t}$;
- wind load (from the left) $P_3 = -0.75 \text{ t}$;
- wind load (from the left) $P_4 = -1.125 \text{ t}$;
- wind load (from the right) $P_1 = 1.0 \text{ t}$;
- wind load (from the right) $P_2 = 1.5 \text{ t}$;
- wind load (from the right) $P_3 = 0.75 \text{ t}$;
- wind load (from the right) $P_4 = 1.125 \text{ t}$.

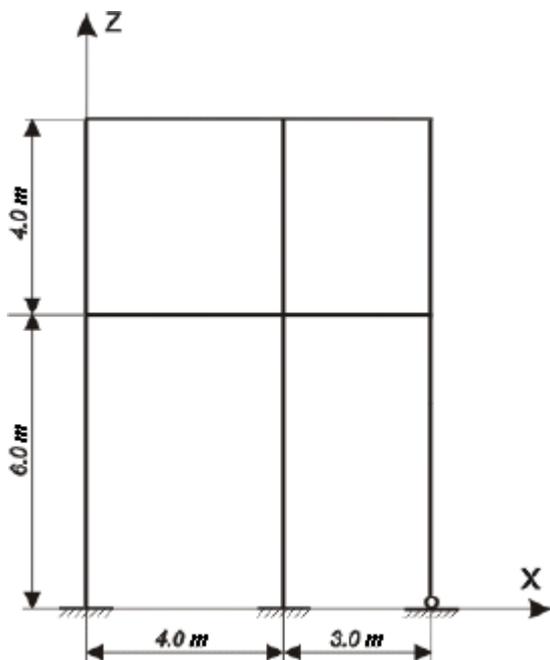


Figure 1.1 Model of the frame

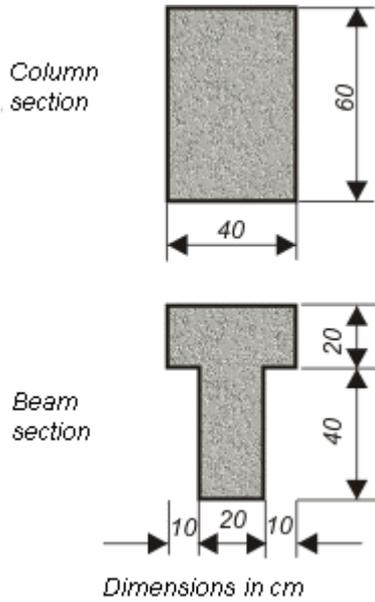
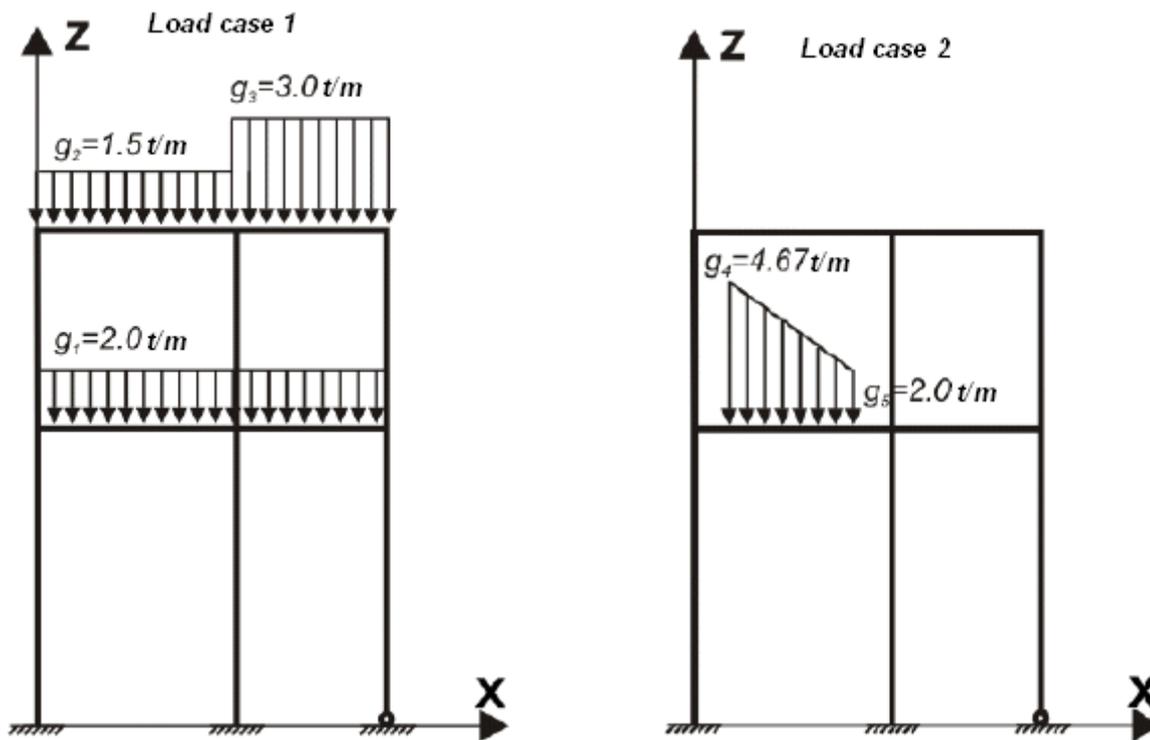


Figure 1.2 Sections for elements of the frame

Perform analysis for four load cases presented in the Fig.1.3.



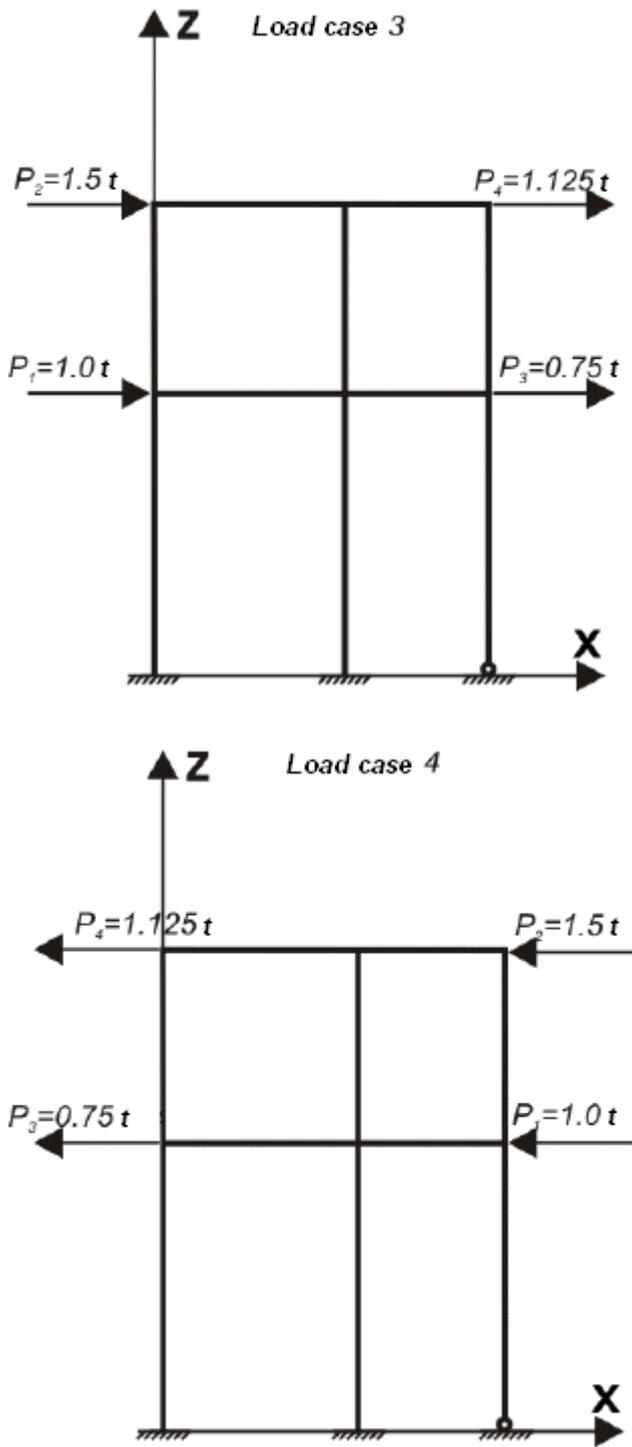


Figure 1.3 Load cases for the frame

- ⇒ 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.1.4) specify the following data:
 - problem name – **Example1** (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** .

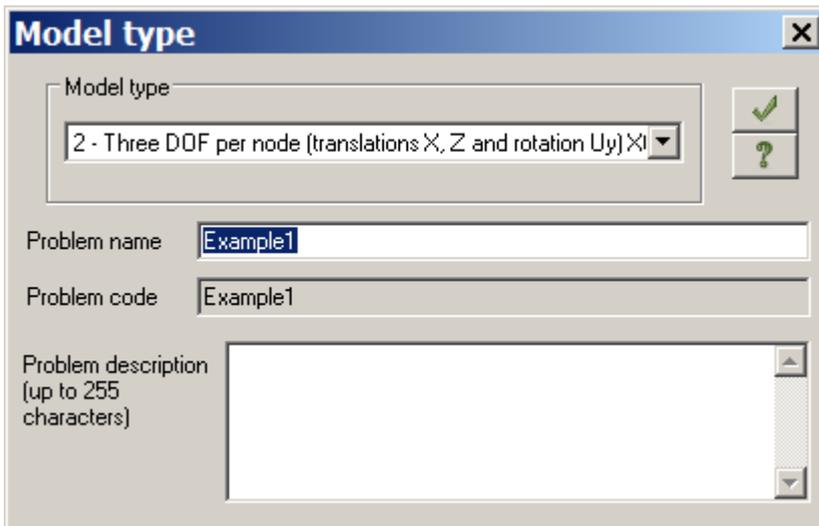


Figure 1.4 **Model type** dialog box



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** 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(i) N	L(i) N
4 1	6 1
3 1	4 1.
 - other parameters remain by default (see Fig.1.5).
- ⇒ Click **Apply** .

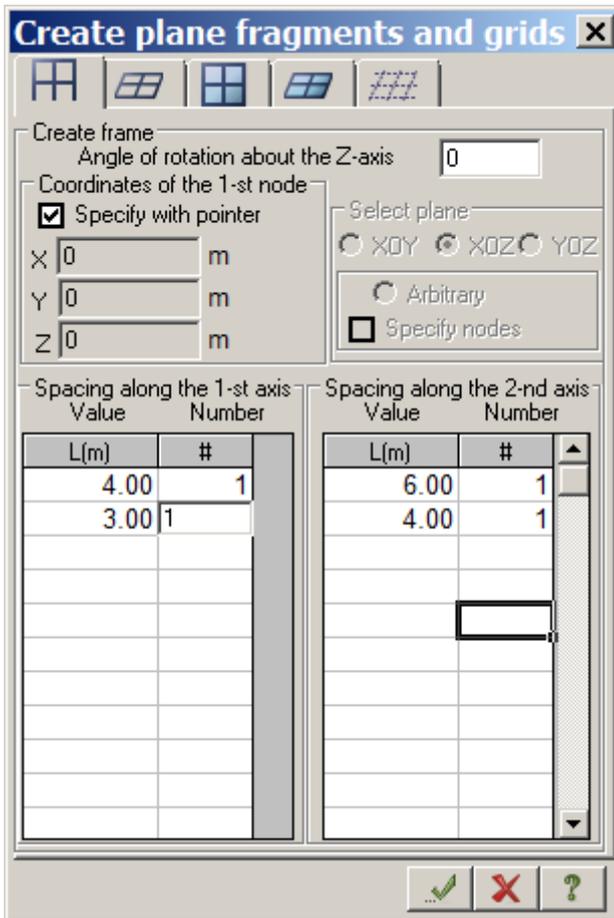


Figure 1.5 Create plane fragments and grids dialog box

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 – **Example1**;
 - location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

Step 3. Defining boundary conditions

To present numbers of nodes and elements on the screen:

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

The model with numbers of nodes and elements is presented in Fig.1.6.

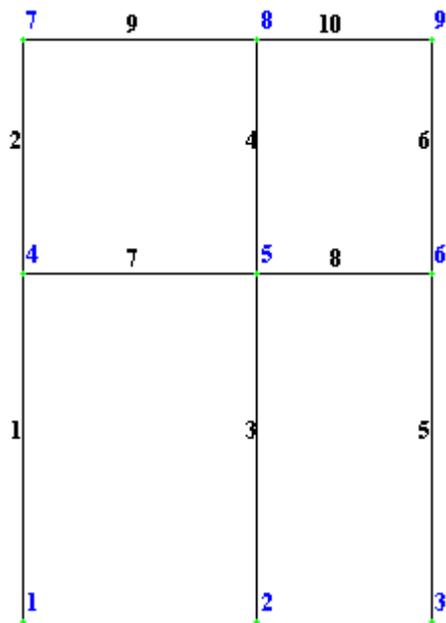


Figure 1.6 Design model with numbers of nodes and elements

To select nodes No.1 and 2:

- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Select nodes No.1 and 2 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 define boundary conditions for nodes No.1 and 2:

- ⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Restraints** button .
- ⇒ In the **Restraints on nodes** dialog box (see Fig.1.7) 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).

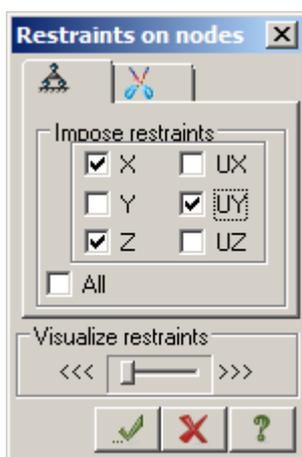


Figure 1.7 **Restraints on nodes** dialog box

To define boundary conditions for node No.3:

- ⇒ Select node No.3 with the pointer.

- ⇒ In the **Restraints on nodes** dialog box specify directions along which displacements of nodes are not allowed (X, Z). To do this, simply clear the UY check box.
- ⇒ Click **Apply**  .
- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button  in order to make this command not active.

Step 4. 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.1.8), define parameters for the first design option:
 - in the **Analysis of sections by** list, select **DCF**;
 - other parameters remain by default.
- ⇒ Click **Apply**  .

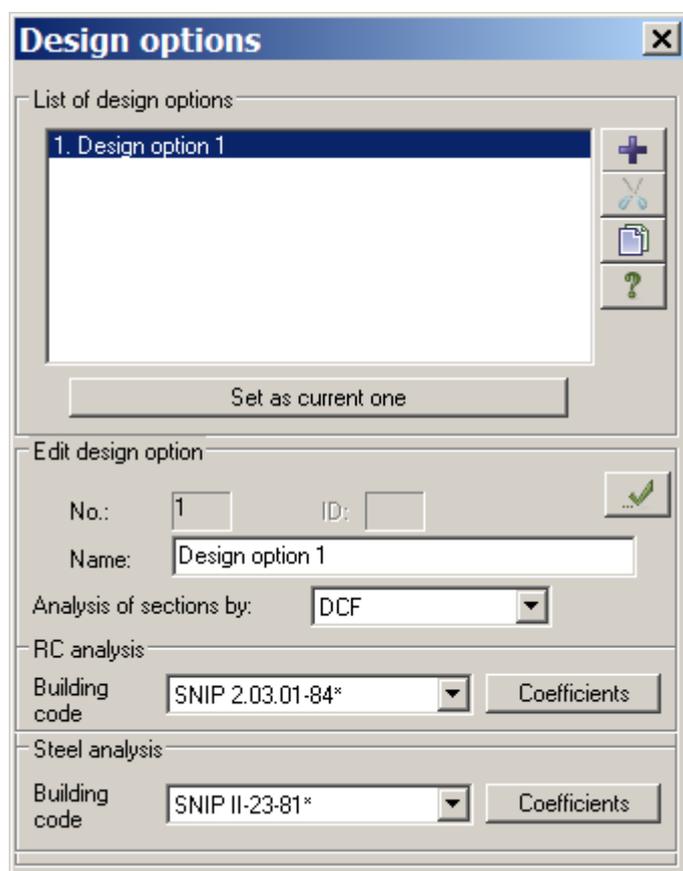


Figure 1.8 **Design options** dialog box

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

Step 5. Defining material properties to elements of the frame



To analyse the model you should specify material properties for elements. Number of specified material properties depends on FE type. These properties include areas of cross-sections, moments of inertia for sections, thickness for slab and shell elements, modulus of elasticity and shear modulus, moduli of subgrade reaction for elastic foundation.

Material properties are specified in the following way:

*- first of all, input numerical data for material properties. Every set of data we will name as **type of***

stiffness or simply **stiffness**. Certain number will be assigned to every type of stiffness;
 - one of the stiffness types is set as a **current** stiffness type;
 - then it is necessary to select elements to which the **current** stiffness will be assigned;

- to assign material properties of the current stiffness type to all selected elements, click **Apply**  .
 To activate the **Add stiffness** dialog box, in the **Stiffness and materials** dialog box, click **Add** button (the **Properties** tab should be active). The **Add stiffness** dialog box contains three tabs where you can find the **library of stiffness parameters**. By default the tab with **Standard types of sections** is displayed. On the other two tabs you can define parameters for standard steel sections and numerical description of stiffness that corresponds to certain types of FE. You can also define **non-standard** and **thin-walled** sections there.

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.1.9a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.1.9b).

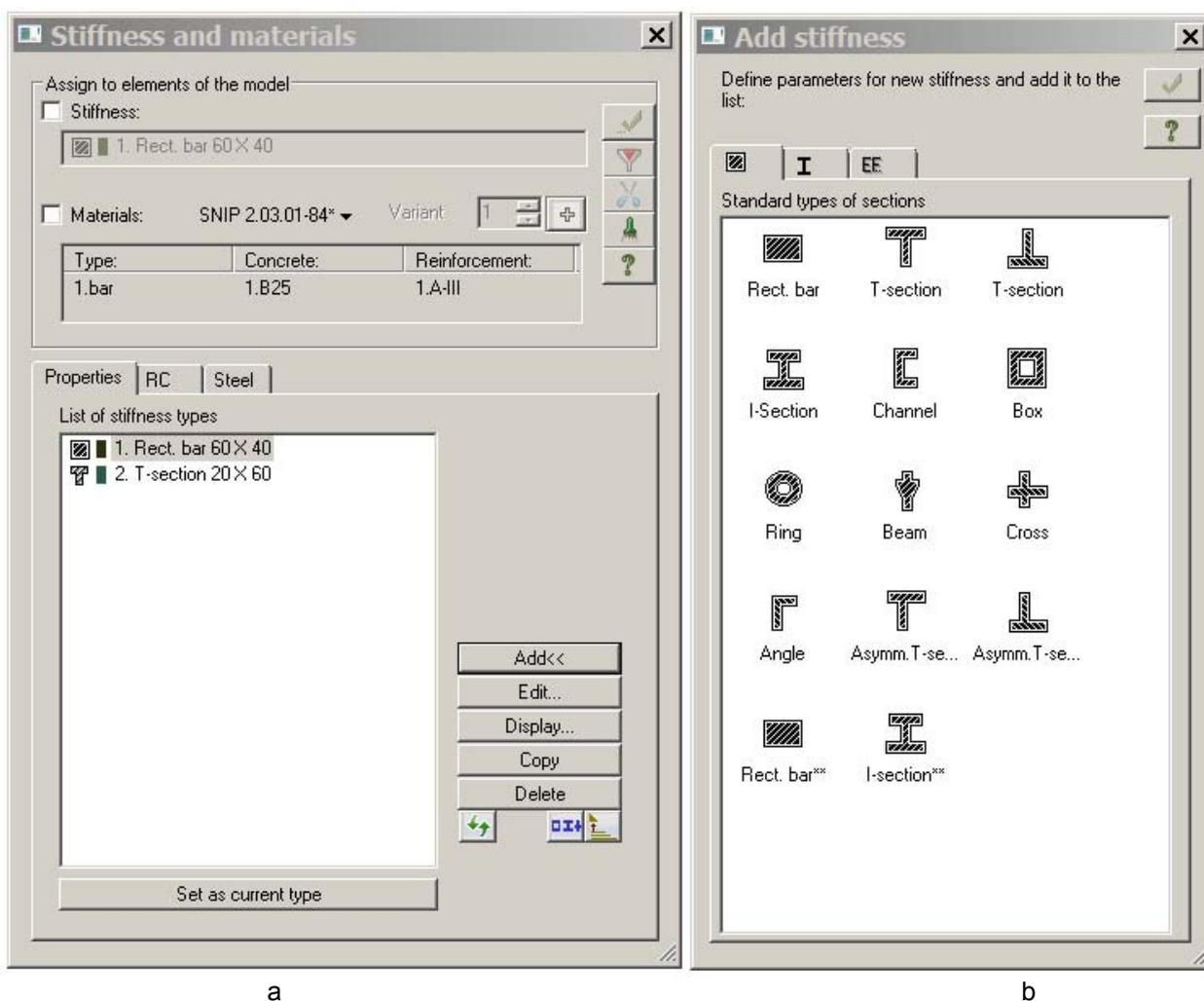


Figure 1.9 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ Double-click the **Rectangular bar** icon in the list. The **Define standard section** dialog box opens. In this dialog box you can define material properties for selected type of the section.
- ⇒ In the **Define standard section** dialog box specify the following parameters for **Rectangular bar** (see Fig.1.10):
 - modulus of elasticity – $E = 3e6 \text{ t/m}^2$ (for the U.S. keyboard layout);

- geometric properties – B = 60 cm; H = 40 cm.
- ⇒ To preview schematic presentation, click **Draw**.
- ⇒ To confirm the specified data, click **OK** .

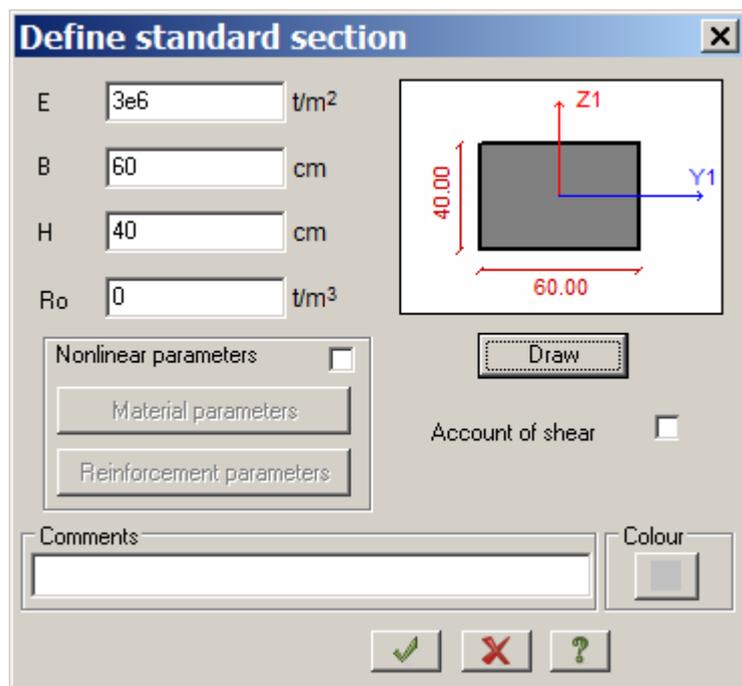


Figure 1.10 **Define standard section** dialog box

- ⇒ Then in the **Add stiffness** dialog box, double-click the **T-section (table at the top)** icon in the list.
- ⇒ In another **Define standard section** dialog box, specify the following parameters for **T-section (table at the top)**:
- modulus of elasticity – E = 3e6 t/m²;
 - geometric properties – B = 20 cm; H = 60 cm; B1 = 40 cm; H1 = 20 cm.
- ⇒ To confirm the specified data, click **OK** .
- ⇒ To hide library of stiffness properties, in the **Stiffness and materials** dialog box click **Add** unfold button.

To define materials for reinforced concrete (RC) structures:



The **Reinforced concrete structures** mode is mentioned to analyse reinforcement and design RC bar and plate elements. Analysis and design procedures are carried out according to the following building codes SNIP 2.03.01-84, TSN102-00, DSTU 3760-98, SP 63.13330.2012, DBN V.2.6-98:2009, etc.

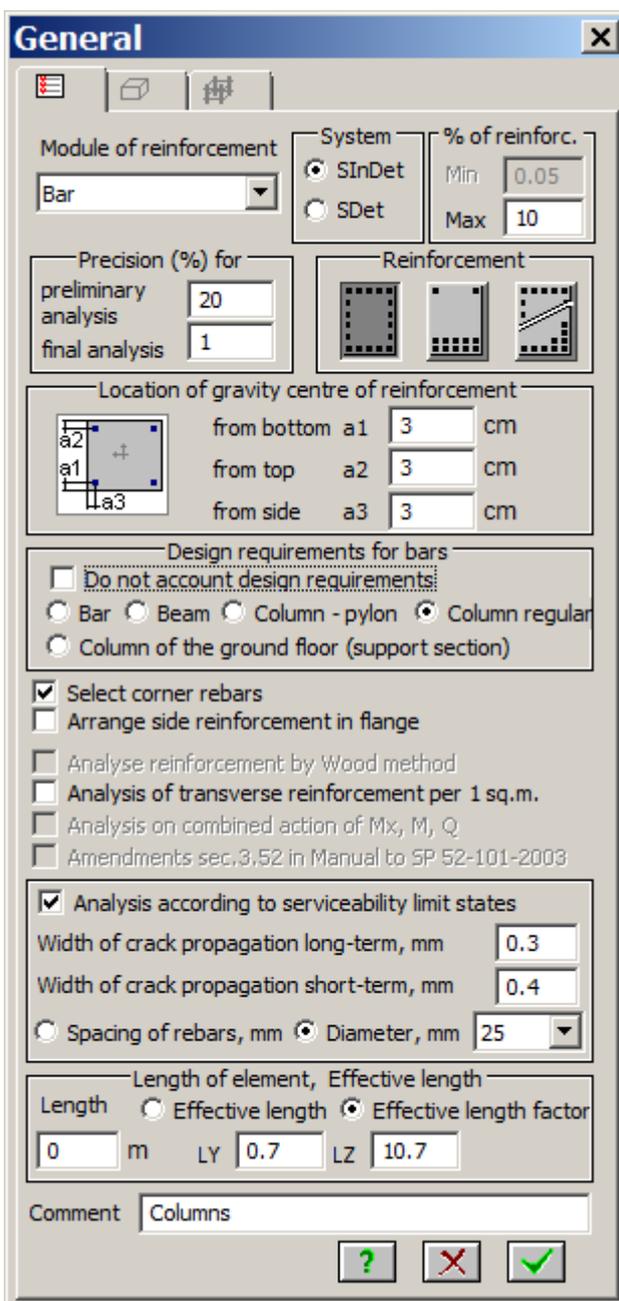
Material properties and parameters of sections should be defined when you generate design model. Materials for reinforced concrete structures (classes of concrete and reinforcement) may be defined in the mode of **Model generation** and in the **Reinforced concrete structures** mode.

There are four modules of reinforcement where it is possible to analyse reinforcement according to ultimate limit states (ULS) and serviceability limit states (SLS).

- module **bar**;
- module **wall-beam**;
- module **slab**;
- module **shell**.

- ⇒ To define parameters for reinforced concrete structures, in the **Stiffness and materials** dialog box, click the second tab **Reinforced concrete (RC)**.
- ⇒ Select **Type** option and click **Add**.

- ⇒ In the **General parameters** dialog box (see Fig.1.11), define the following parameters for columns:
- in the **Module of reinforcement** list, select **Bar**;
 - in the **Reinforcement** area, select **Symmetric** type of reinforcement;
 - in the **Design requirements for bars** area, select **Column regular** option and clear the **Do not account design requirements** check box;
 - in the **Analysis according to serviceability limit states** area, select **Diameter** option;
 - in the drop-down list, select diameter of reinforcement 25mm;
 - in the **Length of element, Effective length**, select **Effective length factor** option;
 - define parameters LY = 0.7, LZ = 0.7;
 - in the **Comment** box, type comment - **Columns**;
 - other parameters remain by default.
- ⇒ Click **OK** .



General

Module of reinforcement: Bar

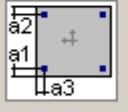
System: SInDet SDet

% of reforc. Min: 0.05 Max: 10

Precision (%) for preliminary analysis: 20 final analysis: 1

Reinforcement: 

Location of gravity centre of reinforcement:

 from bottom a1: 3 cm
from top a2: 3 cm
from side a3: 3 cm

Design requirements for bars:

Do not account design requirements

Bar Beam Column - pylon Column regular Column of the ground floor (support section)

Select corner rebars

Arrange side reinforcement in flange

Analyse reinforcement by Wood method

Analysis of transverse reinforcement per 1 sq.m.

Analysis on combined action of Mx, My, Q

Amendments sec.3.52 in Manual to SP 52-101-2003

Analysis according to serviceability limit states

Width of crack propagation long-term, mm: 0.3

Width of crack propagation short-term, mm: 0.4

Spacing of rebars, mm Diameter, mm: 25

Length of element, Effective length:

Length: 0 m Effective length Effective length factor

LY: 0.7 LZ: 10.7

Comment: Columns



Figure 1.11 **General parameters** dialog box

- ⇒ In the **Stiffness and materials** dialog box, click **Add** once again.
- ⇒ In another **General parameters** dialog box, define parameters for beams:
 - in the **Reinforcement** area, select **Asymmetric** type of reinforcement;
 - in the **Design requirements for bars** area, select **Beam** option and clear the **Do not account design requirements** check box;
 - in the **Analysis according to serviceability limit states** area, select **Diameter** option;
 - in the drop-down list, select diameter of reinforcement 25mm;
 - in the **Comment** box, type comment - **Beams**;
 - other parameters remain by default.
- ⇒ Click **OK**  .
- ⇒ In the **Stiffness and materials** dialog box, select the **Concrete** option.
- ⇒ Click **Default** (in this case, concrete B25 is accepted by default).
- ⇒ In the same dialog box, select the **Reinforcement** option.
- ⇒ Click **Default** (in this case, reinforcement A-III is accepted by default).

To assign stiffness and material properties to elements of the frame:

- ⇒ On the **Select** toolbar, click **Select horizontal bars** button  . In this case, make sure that stiffness **2.T-section 20x60** 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: type – **2.bar**, concrete class – **1.B25** and class of reinforcement – **1.A-III**.
- ⇒ Select all horizontal elements of the model with the pointer (selected elements will be coloured red).



*You can select elements either with a single click or by dragging selection window around appropriate elements. When the **Select horizontal bars** command is active, you can drag selection window around the whole model and only horizontal bars will be selected.*

- ⇒ In the **Stiffness and materials** dialog box, click **Apply**  . The elements become unselected. It indicates that the current combination of stiffness type and material is assigned to selected elements.
- ⇒ On the **Select** menu, click **Select horizontal bars**  in order to make this command not active.
- ⇒ In the **Stiffness and materials** dialog box, on the **RC** tab, select **Type** option and in the list of types of material properties for RC structures, select the line **1.bar Columns**.
- ⇒ Click **Set as current type** (In this case selected type of material properties will be displayed in the **Materials** box in the **Assign to elements of the model** area. You can also specify the current type by double-clicking the necessary type in the **List of stiffness types**.)
- ⇒ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the stiffness type **1. Rect. bar 60x40**.
- ⇒ 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).
- ⇒ On the **Select** toolbar, click **Select vertical bars** button  .
- ⇒ Select all vertical elements of the model with the pointer. The elements will be coloured red. (When the **Select vertical bars** command is active, you can drag selection window around the whole model and only vertical bars will be selected.)
- ⇒ 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 6. Applying loads



How to assign load case

It is allowed to define up to 300 load cases. Certain number and arbitrary name are assigned to every load case. Load case may include any number of loads. To assign number and name to a load case, on the **Create and edit** ribbon tab, select the **Loads** panel and click **Edit load cases** . In the **Edit load cases** dialog box (see Fig.1.12) specify appropriate data. When you start a program, name **Load case 1** is displayed by default.

How to define loads

To define loads on nodes and elements, on the **Create and edit** ribbon tab, select the **Loads** panel and click appropriate command from the **Loads on nodes and elements** drop-down list. In the **Define loads** dialog box (see Fig.1.13) specify appropriate data.

Define loads dialog box contains tabs where you can specify loads on nodes, bars, plates, solids and super-elements, and loads for time history analysis. It is accepted by default that **loads belong to the same active load case**; the number of this load case was specified beforehand. There is also a tabbed page where it is possible to edit or delete loads of the active load case.

In the dialog box you can also specify coordinate system (global or local), direction of load (X, Y, Z), type of static or dynamic load (static load is coloured brown, initial displacement – yellow, dynamic load – pink, load for special FE turnbuckle – green). Menu of these buttons is changed automatically and depends on the type of FE. When you click these buttons, you will see another dialog box where it is necessary to specify parameters of load. Applied loads and effects will appear in the **Current load** field of the dialog box.

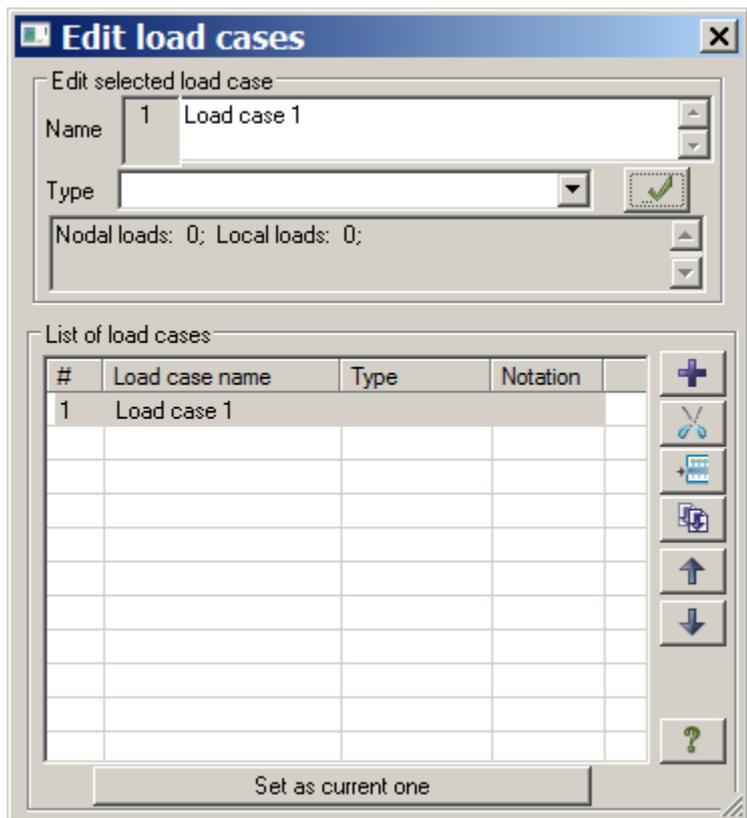


Figure 1.12 **Edit load cases** dialog box

To define detailed information about load cases:

- ⇒ On the **Create and edit** ribbon tab, on the **Loads** panel, click **Edit load cases** . The **Edit load cases** dialog box is displayed on the screen (see Fig.1.12).
- ⇒ For load case 1 – in the **Edit selected load case** area, in the **Type** box, select **Dead** and click **Apply** .

- ⇒ To add the second load case, in the **List of load cases** area, click **Add load case (to the end)** .
- ⇒ For load case 2 – in the **Edit selected load case** area, in the **Type** box, select **Live** and click **Apply** .
- ⇒ To add the third load case, in the **List of load cases** area, click **Add load case (to the end)** .
- ⇒ For load case 3 – in the **Edit selected load case** area, in the **Type** box, select **Instant** and click **Apply** .
- ⇒ To add the fourth load case, in the **List of load cases** area, click **Add load case (to the end)** .
- ⇒ For load case 4 – in the **Edit selected load case** area, in the **Type** box, select **Instant** and click **Apply** .
- ⇒ To continue generating the first load case, in the **List of load cases** area, select the row **1. Load case 1** and click **Set as current one**. You could also define the current load case by double-clicking appropriate row in the list.



Detailed information about load cases may be also defined when load cases are generated. In this case, you have to define only the type of load case.

To create load case No.1:

- ⇒ On the **Select** toolbar, click **Select horizontal bars** button .
- ⇒ Select horizontal elements No. 7 and 8.
- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel, then select **Load on bars** command .
- ⇒ In the **Define loads** dialog box (see Fig.1.13), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

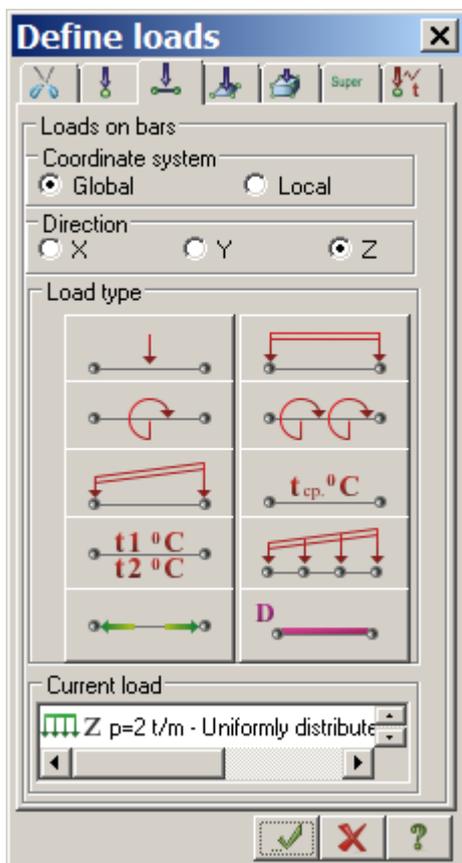


Figure 1.13 Define loads dialog box

- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 2.0$ t/m (see Fig.1.14).
- ⇒ Click **OK** .

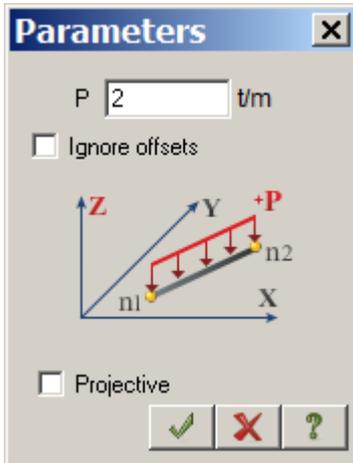


Figure 1.14 Load parameters dialog box

- ⇒ In the **Define loads** dialog box, click **Apply** .
- ⇒ Select element No. 9.
- ⇒ In the **Define loads** dialog box, click **Uniformly distributed load** button.
- ⇒ In the **Load parameters** dialog box specify $P = 1.5$ t/m.
- ⇒ Click **OK** .
- ⇒ In the **Define loads** dialog box, click **Apply** .
- ⇒ Select element No. 10.
- ⇒ In the **Define loads** dialog box, click **Uniformly distributed load** button.
- ⇒ In the **Load parameters** dialog box specify $P = 3.0$ t/m.
- ⇒ Click **OK** .
- ⇒ In the **Define loads** dialog box, click **Apply** .

To create load case No.2:

- ⇒ To change the number of the current load case, click the **Next load case** button  located on the Status bar or on the toolbar.
- ⇒ Select element No. 7.
- ⇒ In the **Define loads** dialog box, click **Trapezoidal load** button .
- ⇒ In the **Load parameters** dialog box, specify the following data: $P1 = 4.67$ t/m, $A1 = 0.5$ m, $P2 = 2.0$ t/m, $A2 = 3.5$ m (see Fig.1.15).
- ⇒ Click **OK**.

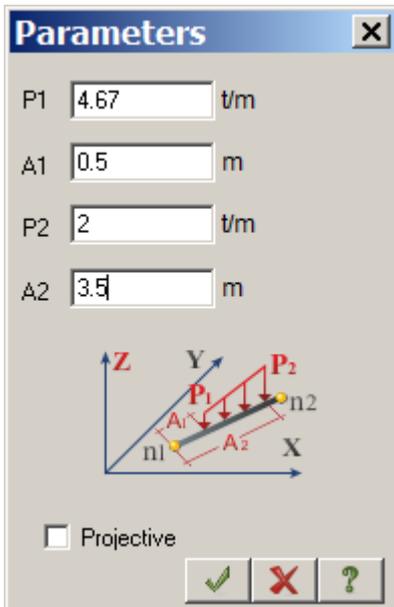


Figure 1.15 Load parameters dialog box (trapezoidal load)

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

To create load case No.3:

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

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

⇒ Select node No. 4 with the pointer.

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

⇒ Specify **Global** coordinate system and direction along the X-axis.

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

⇒ In the **Load parameters** dialog box specify $P = -1$.

⇒ Click **OK**.

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

⇒ Select node No. 7.

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

⇒ In the **Load parameters** dialog box, specify $P = -1.5$ t.

⇒ Click **OK**.

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

⇒ In a similar way, define the following loads:

- on node No. 6 – $P_3 = -0.75$ t;
- on node No. 9 – $P_4 = -1.125$ t.

To create load case No.4:

⇒ Change the number of the current load case and specify number **4**.

⇒ Select node No. 4.

- ⇒ In the **Load type** area, click the **Concentrated load** button.
- ⇒ In the **Load parameters** dialog box, specify $P = 0.75$ t.
- ⇒ Click **OK**.
- ⇒ In the **Define loads** dialog box, click **Apply**.

- ⇒ In a similar way, define the following loads:
 - on node No. 6 – $P1 = 1$ t;
 - on node No. 9 – $P2 = 1.5$ t;
 - on node No. 7 – $P4 = 1.125$ t.

- ⇒ 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.

Step 7. Generating DCF table



According to building codes, it is necessary to analyse reinforcement, determine and check steel sections by the most dangerous combinations of forces. That's why, to work further in **Reinforced concrete and steel structures** mode, it is necessary to calculate DCF or DCL. Calculation of design combinations of forces (DCF) is made according to criterion of extreme stresses at specific points of element sections according to building codes (unlike calculation of DCL where calculation is made by direct adding appropriate values of nodal displacements and forces in elements).

See also detailed description of DCF table (at the end of this document)

- ⇒ On the **Analysis** ribbon tab, select the **DCF** panel and click **DCF table** button  .



As the type of load cases was defined in the **Edit load cases** dialog box (see Fig.1.12), the DCF table is generated automatically with parameters accepted by default for every load case. Now you have to modify parameters for the third and the fourth load cases.

- ⇒ In the **Design combinations of forces** dialog box (see Fig.1.16), select building code **SNIP 2.01.07-85*** and specify the following data:
 - for Load case 3 – in the **No. of group of mutually exclusive load cases** box specify **1**, then click **Apply**  ;
 - for Load case 4 (the same data as for load case 3) – in the **No. of group of mutually exclusive load cases** box specify **1**, then click **Apply**  ;
- ⇒ Click **OK**  .

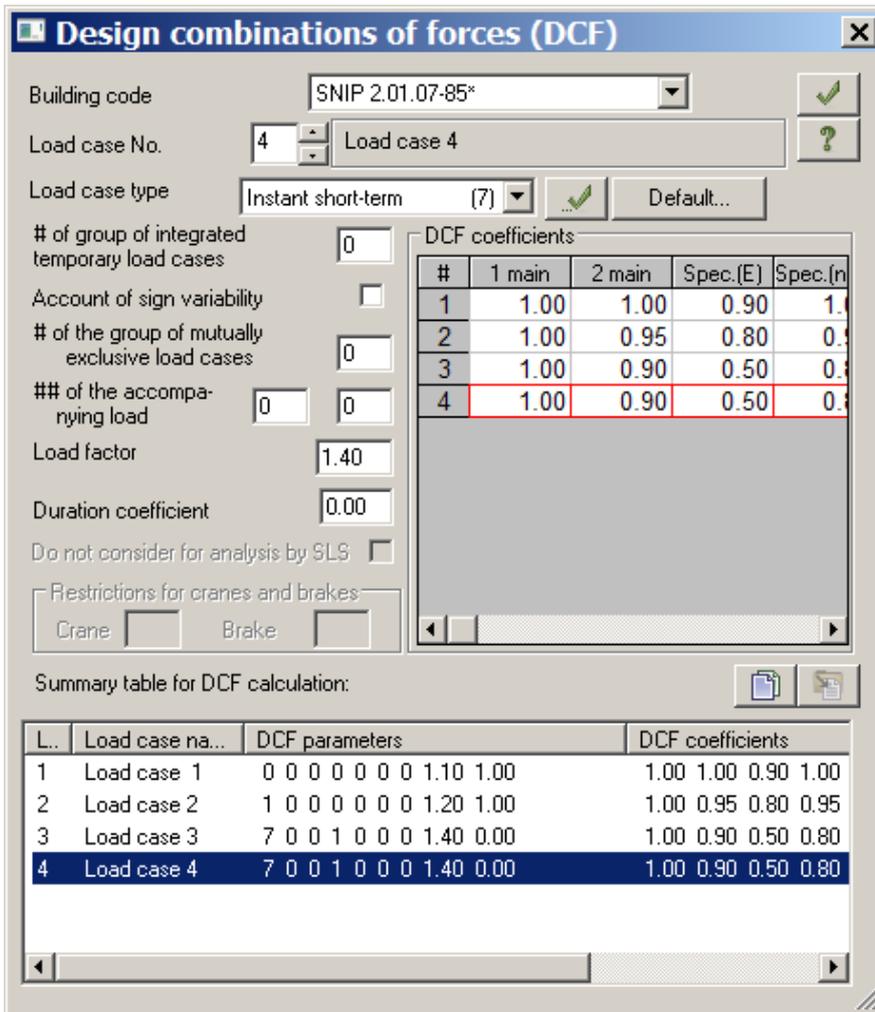


Figure 1.16 Design combinations of forces dialog box

Step 8. Defining design sections for beams

- ⇒ Select all horizontal elements on the model.



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.1.17), specify number of sections N = 5.
- ⇒ Click **Apply**  (to design an element subject to bending, it is necessary to calculate forces in three or more sections) and close the dialog box.



Figure 1.17 Design sections dialog box

Step 9. Defining structural elements

To define structural element BEAM:

- ⇒ Select horizontal elements No. 7 and 8.
- ⇒ 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.1.18), under **Edit structural elements**, click **Create** (structural element BEAM is assigned in order to consider that it is precisely continuous beam).

To define structural element COLUMN:

- ⇒ On the **Select** toolbar, click **Select vertical bars** button .
- ⇒ Select elements No. 1 and 2.
- ⇒ In the **Structural elements** dialog box, under **Edit structural elements**, click **Create** (structural element COLUMN is assigned in order to consider that it is precisely solid column).

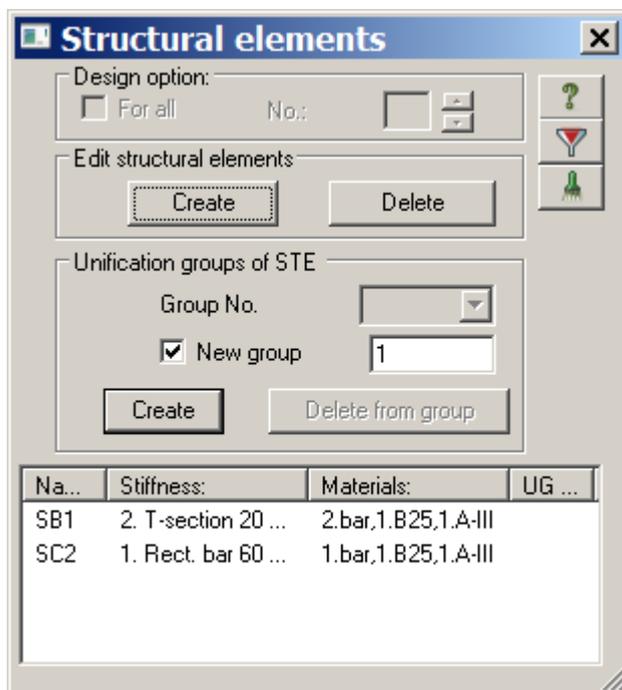


Figure 1.18. **Structural elements** dialog box

Step 10. 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 11. 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 (see Fig.1.19). To display the model without nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button .

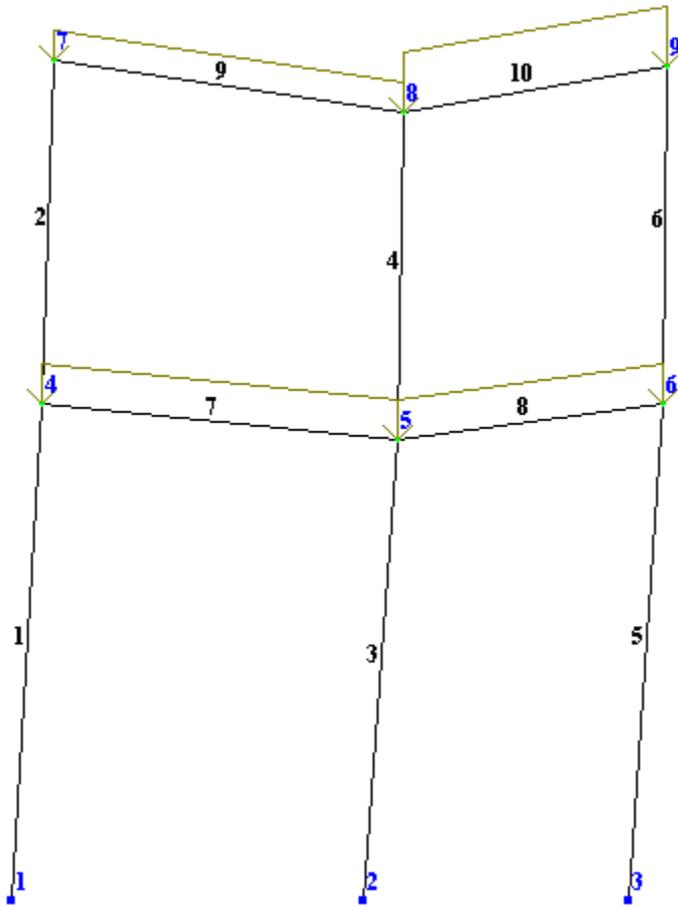


Figure 1.19 Design model with account of nodal displacements

To present diagrams of internal forces:

- ⇒ To display diagram M_y (see Fig.1.20), on the **Results** tab, select **Forces in bars** panel and click **Moment diagrams (M_y)** button .

Load case 1
 Diagram M_y
 Units of measurement - t*m

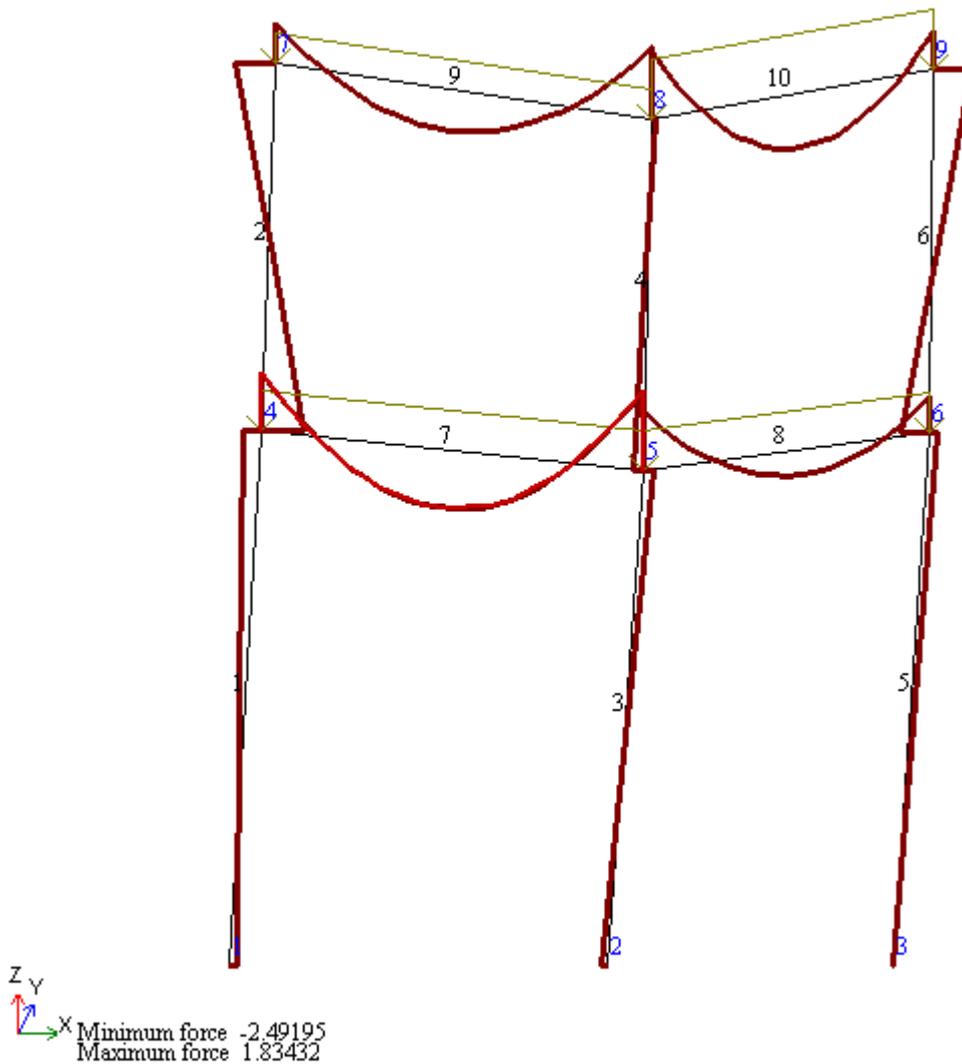


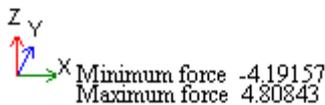
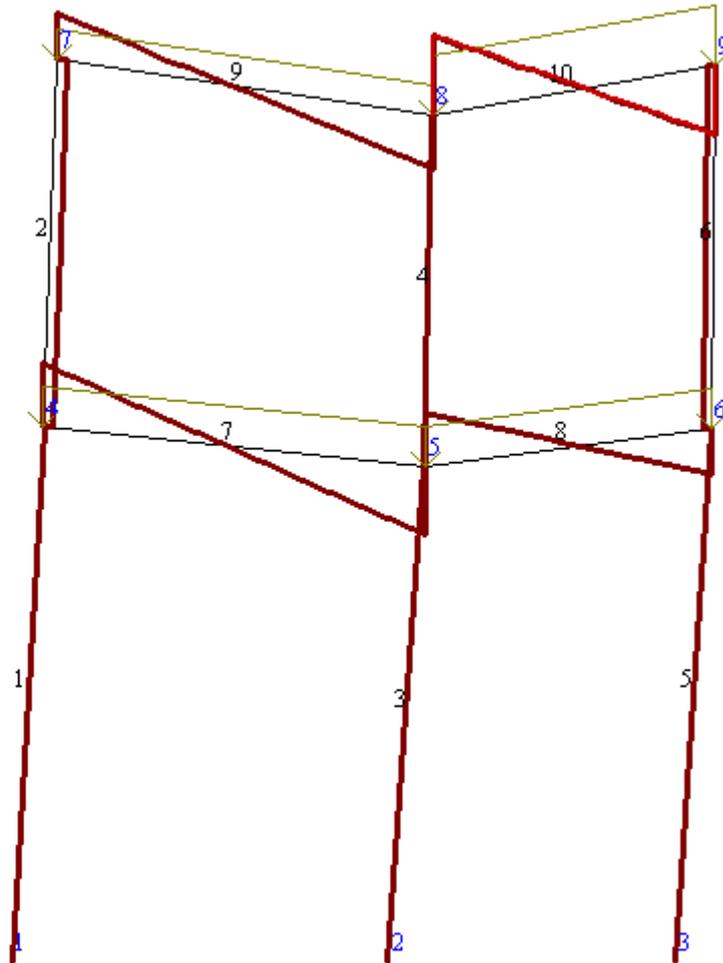
Figure 1.20 Diagram of bending moments M_y

- ⇒ To display diagram Q_z (see Fig.1.21), on the **Results** tab, select **Forces in bars** panel and click **Shear force diagrams (Q_z)** button .
- ⇒ To display mosaic plots Q_z , 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.

Load case 1

Diagram Qz

Units of measurement - t

Figure 1.21 Shear force diagram Q_z

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.1.22), select **Design combinations of forces, design values** 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.

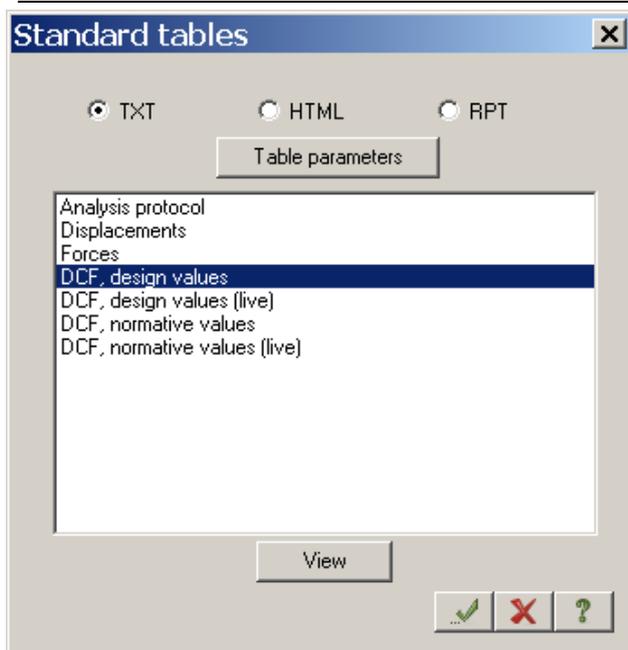


Figure 1.22 **Standard tables** dialog box

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

Step 12. Review and evaluate results from analysis of reinforcement



*When analysis procedure is complete, to review and evaluate analysis results for reinforcement, select the **Design** ribbon tab.*

To present results from analysis of reinforcement:

- ⇒ To present information about determined reinforcement in a certain element, on the **Select** toolbar, click **Information about nodes and elements** button  and specify with a pointer any element on the model.
- ⇒ In the dialog box that appears on the screen, select the **Longitudinal reinforcement** tab. This dialog box contains complete information about selected element, including results for reinforcement.
- ⇒ To close the dialog box, click **Close** button.
- ⇒ To switch to the mode for presentation of symmetric reinforcement in rebars, on the **Design** ribbon tab, select **RC: Bars** panel and click **Symmetric reinforcement** command  in the **Reinforcement** drop-down list.
- ⇒ To display mosaic plot for area of longitudinal reinforcement in the lower left corner of the section AU1, click the **Corner reinforcement AU1** button  (on the **Design** ribbon tab, the **RC: Bars** panel).
- ⇒ To display mosaic plot for area of longitudinal reinforcement in the lower right corner of the section AU2, click the **Corner reinforcement AU2** button  (on the **Design** ribbon tab, the **RC: Bars** panel).
- ⇒ To switch to the mode for presentation of asymmetric reinforcement in rebars, on the **Design** ribbon tab, select **RC: Bars** panel and click **Asymmetric reinforcement** command  in the **Reinforcement** drop-down list.

To generate and review table with analysis results for reinforcement:

- ⇒ On the **Design** ribbon tab, select the **Tables** panel and click **Analysis results tables for RC** command  in the **Documents** drop-down list.

- ⇒ In the **Tables of analysis results** dialog box (see Fig.1.23), the following data is mentioned by default: **Reinforcement in bars** option in the **Elements** area, option **For all elements** in the **Create table** area and **Text** option in the **Table format** area.
- ⇒ Click **Table - on the screen** (It is also possible to generate tables with analysis results in other formats in the same way, just select appropriate format option).

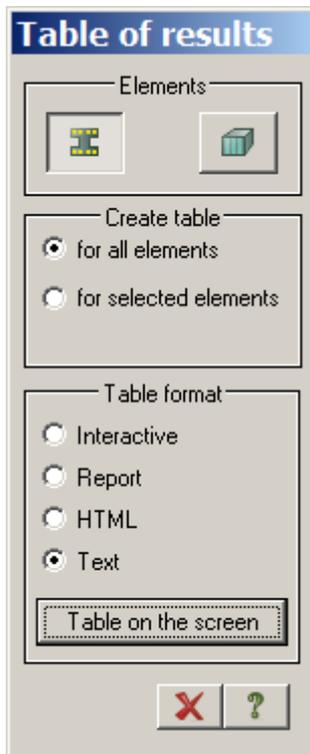


Figure 1.23 **Tables of analysis results** dialog box

Design of beam for reinforced concrete (RC) frame

Step 13. Presenting drawing of the beam

- ⇒ On the **Design** ribbon tab, select **RC: Rebars** panel and click **Design beam** button .
- ⇒ Specify element No.7 with the pointer (**BEAM** module will be presented in a separate window).
- ⇒ In the **BEAM** module, click **Analysis** (button  on the toolbar).
- ⇒ To display diagram of design reinforcement, on the **Results** menu, click **Material diagram** (button  on the toolbar).
- ⇒ To display drawing of the beam, on the **Results** menu, click **Drawing** (button  on the toolbar).

Design of column for reinforced concrete (RC) frame

Step 14. Presenting drawing of the column

- ⇒ On the **Design** ribbon tab, select **RC: Rebars** panel and click **Design column** button .
- ⇒ Specify element No.1 with the pointer (**COLUMN** module will be presented in a separate window).
- ⇒ In the **COLUMN** module, click **Analysis** (button  on the toolbar).
- ⇒ To display diagram of design reinforcement, on the **Results** menu, click **Material diagram** (button  on the toolbar).
- ⇒ To display drawing of the beam, on the **Results** menu, click **Drawing** (button  on the toolbar).

Design combinations of forces

Input data for DCF (design combinations of forces) calculation is defined automatically in [LIRA-SAPR](#) program according to appropriate building codes.

To calculate DCF, the program determines extreme values for those components of stress-strain-state that will be taken as the most dangerous criteria (DCF criteria) for this stress-strain-state. In this case, distinctive features of stress-strain-state of different FE types are considered while number of DCF under consideration is significantly reduced.

The extreme values of normal and shear stresses calculated in specific points of reduced* rectangular section and extreme values of forces in section are taken as the most dangerous DCF criteria for bars. (* - *reduced section* is taken to mean the section of arbitrary shape, transformed into rectangular section.) Stresses calculated by Wood-Armer method are taken as criteria for elements of plane stress state, slabs and shells.

The extreme values of stresses are taken as criteria for solids.

There are several rules for generation of DCF table:

- DCF parameters are specified for every load case of the problem;
- There are 9 types of load cases with which appropriate logical bindings are provided automatically. In this case it is possible to take into account sign variability, mutual exclusion and accompaniment of load cases.

Every type of load case is assigned its code:

- (0) – dead;
- (1) – live;
- (2) – short-term;
- (3) – crane short-term;
- (4) – surge short-term;
- (5) – earthquake (specific);
- (6) – specific (except earthquake);
- (7) – instant short-term;
- (9) – inactive (wind static with pulsation).

This classification slightly differs from normative one. For instance, snow load or ice load is not selected into separate group. But you yourself can assign type of load case to them – either live or short-term as stipulated in appropriate standards.

- The software automatically (by default) generates parameters for current type of load case. However, you can edit these parameters later;
- DCF table is generated in the **Design combinations of forces (DCF)** dialog box (see Figure 1.16);
- Data for DCF selection may be defined in the mode of creating design model (before analysis) or in the mode of analysis results visualization (when analysis is performed).



Important. Term **load case** is used in the following cases:

Load case number – unique number specified by the user for certain group of loads acting on the model simultaneously;

Type of load case – name of the load case type specified in [LIRA-SAPR](#) software.

Parameters of DCF

DCF table should be compiled for all load cases defined in the problem. That's why the **Load case No.** box is the first DCF parameter in the dialog. Sequence order of load case numbers may be arbitrary.

Every load case may have its name.

Number of the load case will be displayed in the first column of the summary table. You will see this table either in complete version (in the lower part of the dialog box) or partially (in the **DCF coefficients** area). To review DCF coefficients, use the vertical and horizontal scroll bars.

All DCF parameters are divided into two groups: **DCF parameters** and **DCF coefficients**.

DCF parameters include the following data:

- **Load factors γ_f** . By default, the following values are accepted:
 dead load case $\gamma_f = 1.1$;
 live $\gamma_f = 1.2$;
 short-term $\gamma_f = 1.2$;
 crane and surge $\gamma_f = 1.1$;

instant short-term $\gamma_f = 1.4$;
specific $\gamma_f = 1.0$.

– **Duration coefficient ψ_g** . This coefficient allows you to single out the continuous part of load case. The following duration coefficients are accepted by default:

dead and live load cases $\psi_g = 1.0$;

short-term $\psi_g = 0.35$;

crane short-term $\psi_g = 0.6$;

other load cases $\psi_g = 0.0$;

– **Accompanying load cases**. There are load cases (not more than two) that may be considered together with the main load case. For instance, if vertical crane loads are the main load case, then horizontal surge load is accompanying load case.

This DCF parameter as well as the following two parameters is introduced to take account of logical bindings between load cases.

– **No. of group of mutually exclusive load cases**. This parameter is used to impose limitations on load cases that cannot act simultaneously. For instance, **Wind from the right** and **Wind from the left** are considered as mutually exclusive loads.

– **Account of sign variability**. If this check box is selected, then the load case is included in DCF two times - with its own sign and the opposite one. For instance, earthquake load.

The several limitations are imposed on the logical bindings between load cases:

a) dead (0) and crane (3) load cases cannot be variable in sign;

b) it is allowed to integrate only live (1), short-term (2) and instant (7) load cases;

c) surge (4) load case may accompany only crane (3) load case;

d) live (1), short-term (2), instant (7), earthquake (5) and specific (6) load cases can be treated as accompanying;

e) double accompaniment (when the same load case accompanies two or more load cases) is allowed;

f) it is not allowed to include any accompanying load case into the groups of integration and mutual exclusion;

g) user can create up to 9 groups of integration and mutual exclusion;

h) dynamic load case cannot accompany any other load case.

DCF coefficients

For every DCF, four combinations are considered: two main combinations, one special with earthquake combination and one special except earthquake combination (see Figure 1.16). Coefficients of forces in combinations ψ_i , $i = 1,2,3$ are displayed in every row according to appropriate DCF.

Coefficient values are generated by default depending on the type of load case (see Table 1.1).

Type of load case	Main combinations		Special combination with earthquake	Special combination except earthquake
	1st	2nd		
Dead	1.0	1.0	0.9	1
Live	1.0	0.95	0.8	0.95
Short-term	1.0	0.90	0.5	0.8
Crane	1.0	0.90	0.0	0.0
Surge	1.0	0.90	0.0	0.0
Earthquake	0	0	1.0	0.0
Specific (except earthquake)	0	0	0	1.0
Instant	1.0	0.9	0.5	0.8
Wind static	0	0	0	0

Table 1.1. Values of DCF coefficients accepted by default

Summary table for DCF calculation is presented in the lower part of the dialog box.

Note that by default all coefficients for wind static load case are equal to zero. It is stipulated by specific requirements for generation of wind load case with pulsation.

Summary table is composed automatically as you define data in the dialog box. There are 13 columns in the table. Names for every column are presented in Figure 1.24. The sample first row from the summary table is also presented there.

No. and name of load case	DCF parameters								DCF coefficients				
	Type of load case	No. of group of integrated temporary load cases	Sign variable	No. of group of mutually exclusive load cases		No. of accompanying load cases	No. of accompanying load cases	Load factor	Duration coefficient	1st main	2nd main	Special with earthquake	Special except earthquake
1	0	0	0	0	0	0	0	1.10	1.00	1.00	1.00	0.90	1.00

Figure 1.24 Columns of summary table for DCF calculation

To edit any parameter of summary table, modify its value it in the appropriate field in the upper part of the dialog box.