

Example 2. Analysis of slab

In this lesson you will learn how to:

- generate design model of slab;
- apply loads and generate DCF table;
- deal with design options to determine reinforcement by Karpenko method and Wood method.

Description:

Reinforced concrete slab 3 x 6 m, thickness 150 mm. Rear side of the slab is freely supported along the whole length, front side is freely supported with its ends on columns. Left and right sides of the slab (see Fig.2.1) are free.

Analysis is performed for FE mesh 6 x 12.

Loads:

- Load case 1 – dead weight of slab;
- Load case 2 – concentrated loads $P = 1\text{t}$ applied as shown in Fig.2.1, load case 2;
- Load case 3 – concentrated loads $P = 1\text{t}$ applied as shown in Fig.2.1, load case 3.

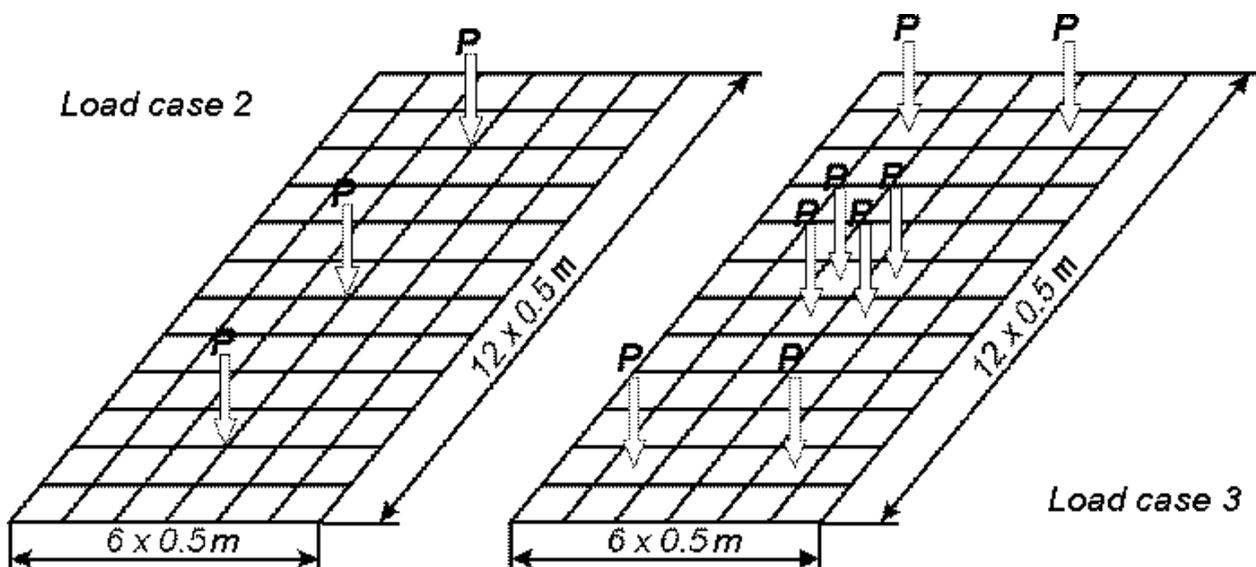


Figure 2.1 Design model of slab

- ⇒ 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.2.2) specify the following data:
 - problem name – **Example2** (problem code by default coincides with the problem name)
 - model type – **3 – Three degrees of freedom per node** (translation Z and rotations Ux, Uy) X0Y.
- ⇒ Click **OK** .

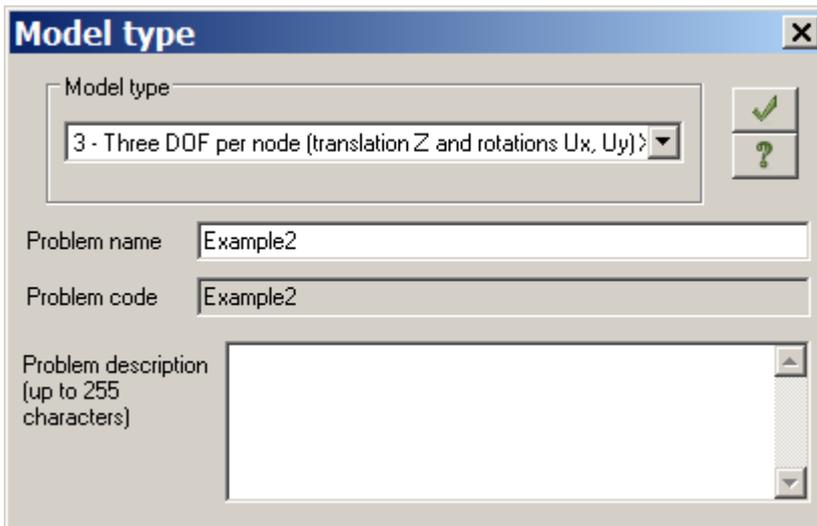


Figure 2.2 **Model type** dialog box



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 3 (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 3 (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 slab**  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
0.5 6	0.5 12
 - other parameters remain by default (see Fig.2.3).
- ⇒ Click **Apply** .

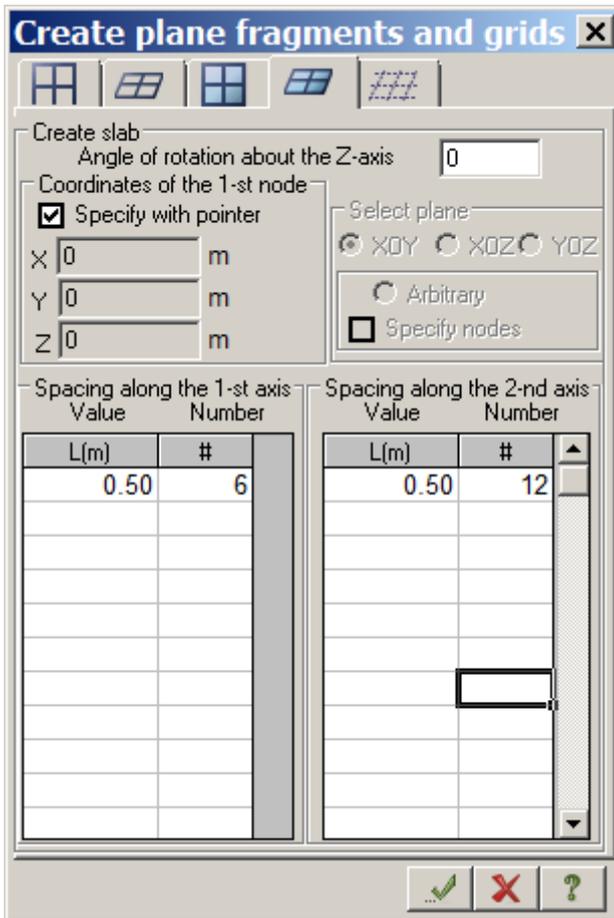


Figure 2.3 Create plane fragments and grids dialog box

To save data about design model:

- ⇒ On the **LIRA-SAPR menu** (Application menu), click **Save** command .
- ⇒ In the **Save as** dialog box specify the following data:
 - file name – **Example2**;
 - 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 on the screen:

- ⇒ On the **Select** toolbar, click **Flags of drawing** button .
- ⇒ In the **Display** dialog box (see Fig.2.4), select the **Node numbers** check box on the **Nodes** tab.
- ⇒ Click **Redraw** .

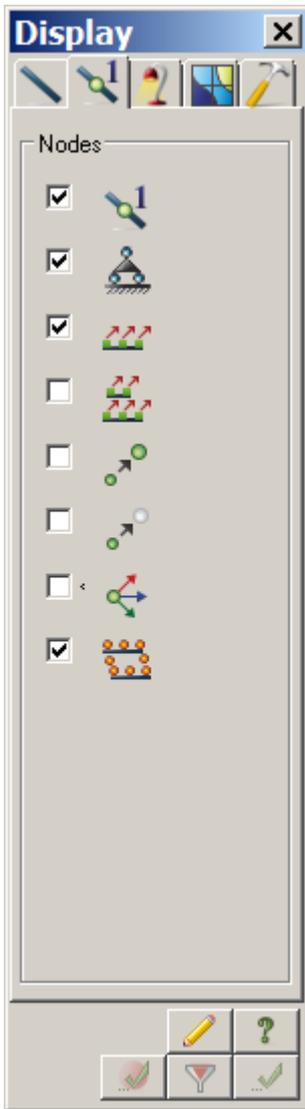


Figure 2.4 Create plane fragments and grids dialog box

The model with numbers of nodes is presented in Fig.2.5.

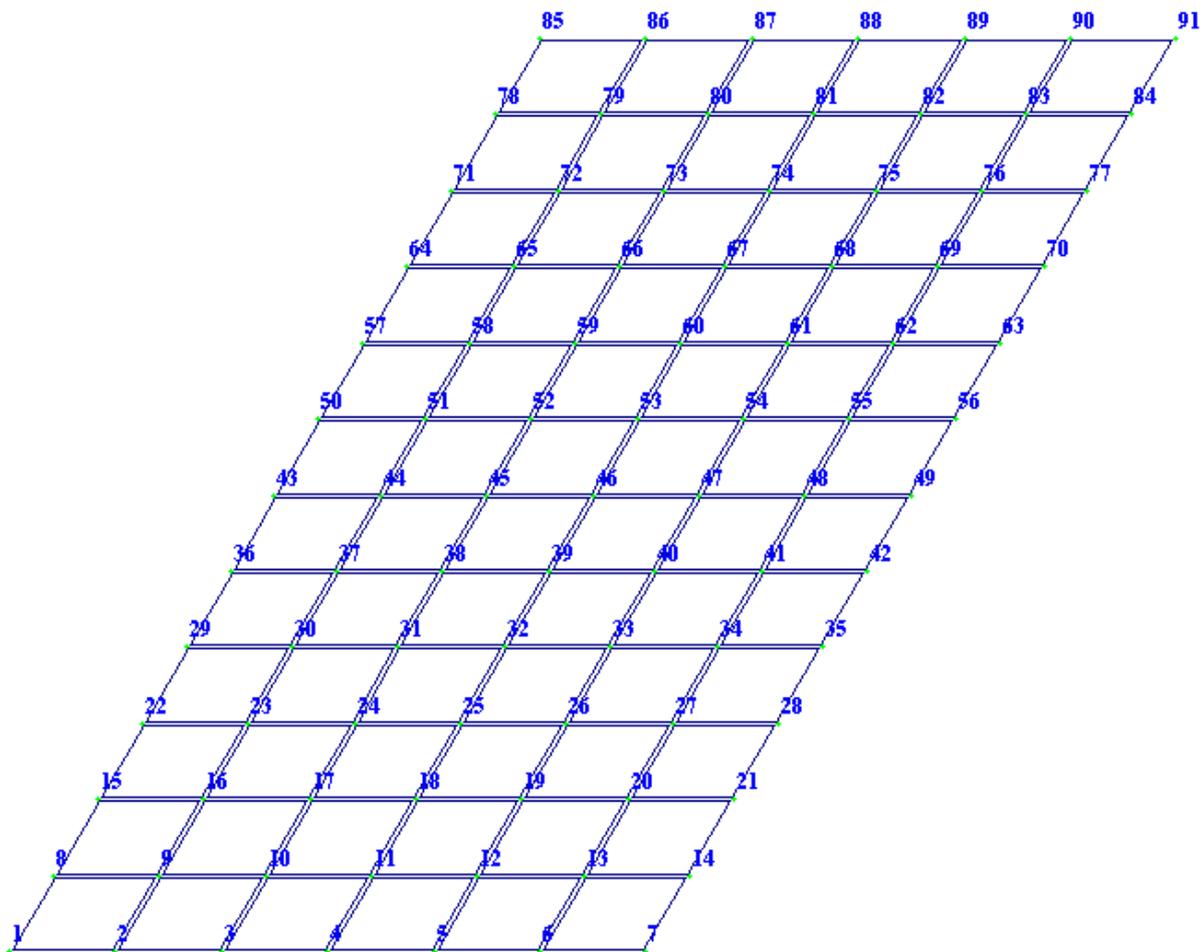


Figure 2.5 Design model of slab with node numbers

To select nodes of support:

- ⇒ On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Select nodes No.1, 7, 85 - 91 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 of support:

- ⇒ 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.2.6), specify directions along which displacements of nodes are not allowed (Z). To do this, select appropriate check boxes.
- ⇒ Click **Apply**  (the nodes will be coloured blue).

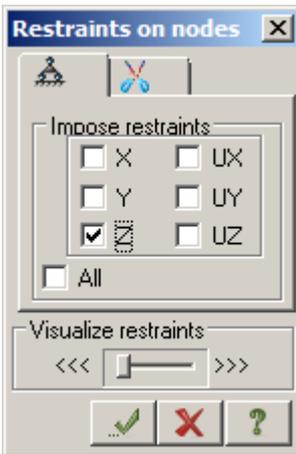


Figure 2.6 Restraints on nodes dialog box

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

Step 4. Defining design options

- ⇒ On the **More edit options** ribbon tab, on the **Design** panel, click **Design options for main model** command .
- ⇒ In the **Design options** dialog box (see Fig.2.7), define parameters for the first design option:
 - in the **Analysis of sections by** list, select **DCF**;
 - other parameters remain by default.
- ⇒ Click **Apply** .

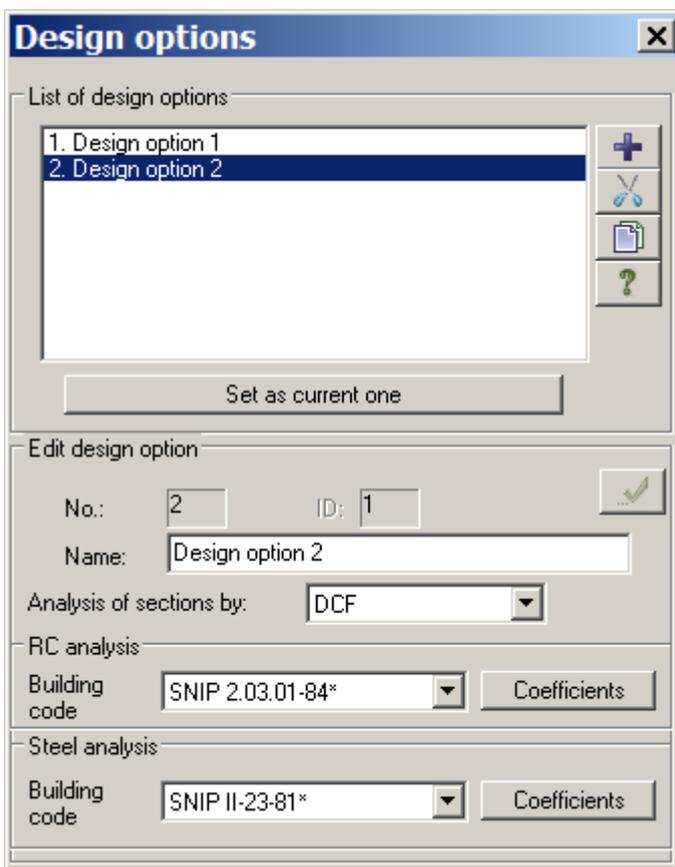


Figure 2.7 Design options dialog box



To define new design option, in the **Design options** dialog box, click **Create new design option**



. By default, all parameters of new design option will obtain values defined previously in the **Analysis parameters** dialog box on the appropriate tabs.

Then define the following parameters:

- name for design option;
- building codes for RC and steel analyses;
- type for analysis of sections (by DCF, DCL or Forces).

To input data for design option, click **Apply**

To assign certain design option as current one (i.e. an active in graphic environment), select appropriate design option in the **List of design options** and click **Set as current one**. To do the same you could also double-click appropriate row in the list.

To define materials for design option, use the **Stiffness and materials** dialog box (see Fig.2.8a).

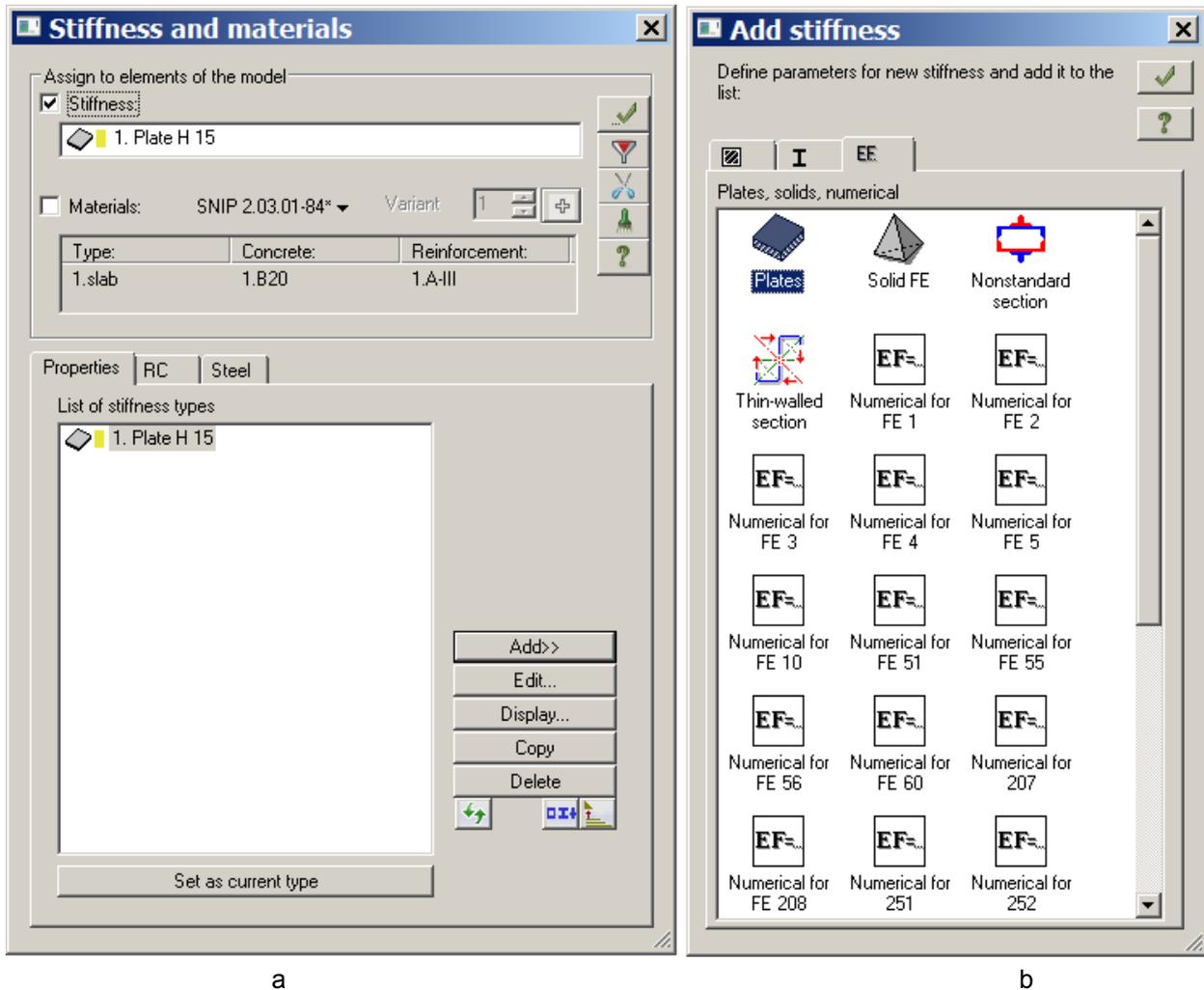
To define the second design option:

- ⇒ To define the second design option, click **Create new design option** .
- ⇒ Then define data for the second design option (analysis of reinforcement by Wood method):
 - in the **Analysis of sections by** list, select **DCF**;
 - other parameters remain by default.
- ⇒ Click **Apply** .
- ⇒ To assign the first design option as the current one, in the **List of design options** either double-click appropriate row or select the row and click **Set as current one**.
- ⇒ To close the **Design options** dialog box, click the **Close** button.

Step 5. Defining material properties to elements of the slab

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.2.8a), click **Add** button. The dialog box will expand and you will see the library of stiffness parameters.
- ⇒ Select the third tab **Plates, solids, numerical** in the window (see Fig.2.8b) and double-click the **Plates** icon in the list.



a

b

Figure 2.8 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ In the **Stiffness for plates** dialog box, specify the following parameters for **Plate** (see Fig.2.9):
 - modulus of elasticity – $E = 3e6 \text{ t/m}^2$ (for the U.S. keyboard layout);
 - Poisson's ratio – $\nu = 0.2$;
 - thickness – $H = 15 \text{ cm}$;
 - unit weight of material – $R_o = 2.75 \text{ t/m}^3$.
- ⇒ To confirm the specified data, click **OK** .

Stiffness for plates

Account of orthotropy

E t/m² E2

V V21

G

H cm Ro t/m³

Nonlinear parameters

FE type

Plate, shell

Wall-beam

Account of shear

Smaller dimension of plate m

Comments

Colour

Figure 2.9 **Stiffness for plates** dialog box

⇒ 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:

- ⇒ 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** (see Fig.2.10).

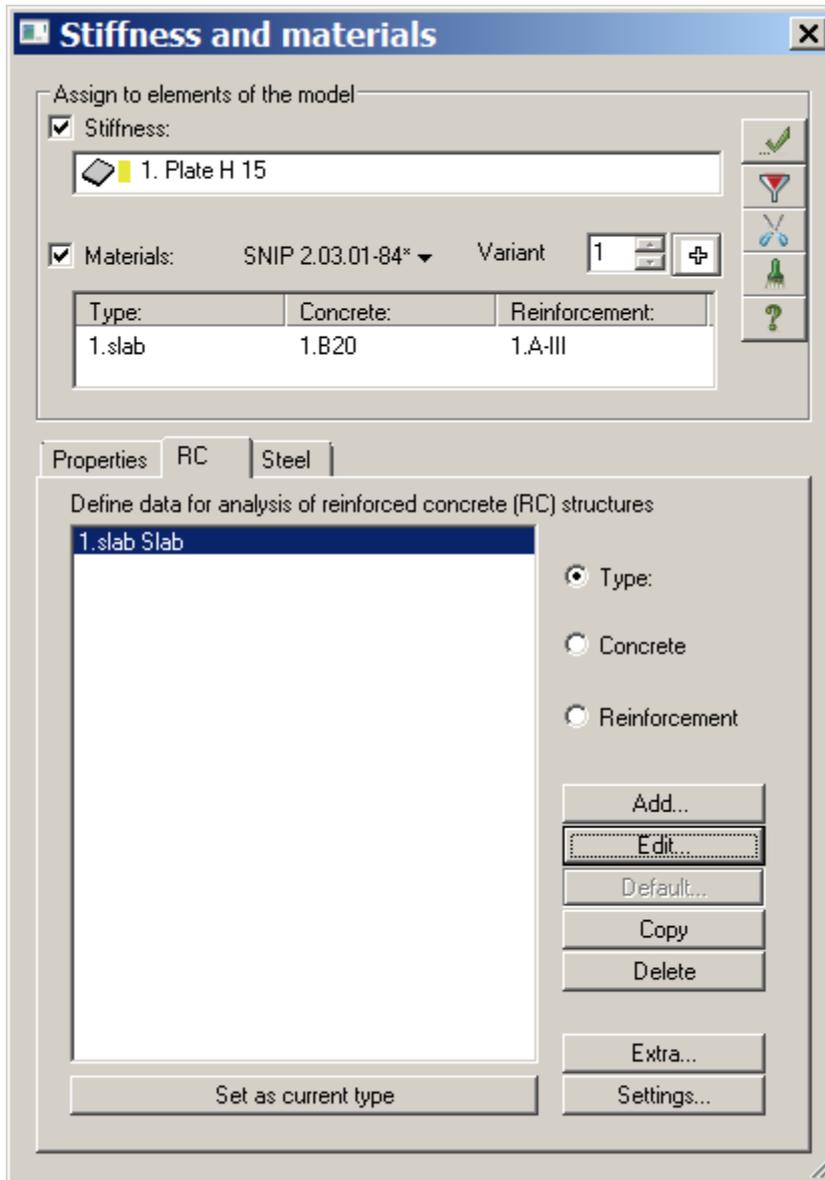


Figure 2.10 **Stiffness and materials** dialog box

- ⇒ In the **General parameters** dialog box (see Fig.2.11), define the following parameters for plate elements:
 - in the **Module of reinforcement** list, select **Slab**;
 - in the **Comment** box, type comment - **Slab**;
 - other parameters remain by default.
- ⇒ Click **OK**  .

General

Module of reinforcement: Slab

System: SInDet SDet

% of reforc.: Min 0.05, Max 10

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

Reinforcement: [Grid icons]

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 rebar, mm Diameter, mm: 100

Length of element, Effective length:

Length: 0 m, LY: 1, LZ: 1

Effective length: Effective length Effective length factor

Comment: Slab

[?] [X] [✓]

Figure 2.11 General parameters dialog box

- ⇒ In the **Stiffness and materials** dialog box, select the **Concrete** option and click **Add**.
- ⇒ In the **Concrete** dialog box (see Fig.2.12), define the following parameters:
 - in the **Class of concrete** list, select **B20** type of reinforcement;
 - other parameters remain by default.
- ⇒ Click **OK** .

Concrete [X]

Class of concrete: B20

Type of concrete: heavyweight

Grade of lightweight concrete by average density D: 800

Accidental eccentricities

along section height EY: 0 cm

along section width EZ: 0 cm

Process of hardening

natural hardening thermal treatment autoclave treatment

Service conditions for the structure

standard favourable for increase of concrete strength

Coefficients

Product of coefficients from table 15 SNIP 2.03.01-84* (except Yb2 and): 1

Values

	Value
Class	B20
Rb	1170.00 t/m**2
Rbt	91.80 t/m**2
Rbn	1530.00 t/m**2
Rbtn	143.00 t/m**2
Eb	2750000.00 t/m**2

Comment: Parameters of concrete

Accept by default ? X ✓

Figure 2.12 Concrete dialog box

- ⇒ In the **Stiffness and materials** dialog box, select the **Reinforcement** option and click **Add**.
- ⇒ In the **Reinforcement** dialog box (see Fig.2.13), to define parameters, click **OK** .

Reinforcement

Longitudinal reinforcement
 along X: A-III d=10...40
 along Y: A-III d=10...40
 Transverse reinforcement: A-I d=6...40

Max diameter, mm: 40
 Max number of rebars in corners of section: 1

Account of earthquake load
 Coefficient from table 7 SNIP II-7-81: 1
 Partial safety factor for inclined sections (table 7 SNIP II-7-81): 1

Partial safety factor for reinforcement (product of coefficients from table 24 SNIP 2.03.01-84*): 1

Value	X Longit...	Y Longit...	Transve...
Class	A-III	A-III	A-I
Diameters	10...40	10...40	6...40
Rs t/m**2	37500.0	37500.0	23000.0
Rsw t/m**2	30000.0	30000.0	18000.0
Rsc t/m**2	37500.0	37500.0	23000.0
Rs,ser t/...	40000.0	40000.0	24000.0
Es t/m**2	200000...	200000...	210000...

Comment: Parameters of reinforcement

Buttons: Accept by default, ?, X, ✓

Figure 2.13 Reinforcement dialog box

To define material properties for the 2nd design option for steel structures:

- ⇒ To switch to the 2nd design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box, specify number 2.
- ⇒ In the **Define parameters for reinforced concrete (RC) structures** area, select the **Type** option and then click **Add**.
- ⇒ In the **General parameters** dialog box (see Fig.2.11), define the following parameters for plate elements:
 - in the **Module of reinforcement** list, select **Slab**;
 - select the **Analyse reinforcement by Wood method** check box;
 - in the **Comment** box, type comment - **Slab - Wood method**;
 - other parameters remain by default.
- ⇒ Click **OK** .

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

- ⇒ On the **Select** toolbar, click **Select elements** button . In this case, make sure that stiffness **1.Plate H 15** 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 – **1.slab**, concrete class – **1.B20** and class of reinforcement – **1.A-III**.
- ⇒ Select all 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.

- ⇒ 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.
- ⇒ To switch to the first design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box, specify number 1.
- ⇒ To define materials to the first design option, clear the **Stiffness** check box under **Assign to elements of the model**.
- ⇒ In the **Stiffness and materials** dialog box, in the list of materials for RC elements, select the row **1.slab Slab**.
- ⇒ Click **Set as current one**.
- ⇒ Select all elements of the model with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .

Step 6. Applying loadsTo create load case No.1:

- ⇒ To define load from dead weight of the slab, on the **Create and edit** ribbon tab, select the **Loads** panel and click **Add dead weight** .
- ⇒ In the **Add dead weight** dialog box (see Fig.2.14), click **All elements** and specify **Load factor** as equal to 1. Then click **Apply**  (dead weight of elements is added according to the specified unit weight R_0).

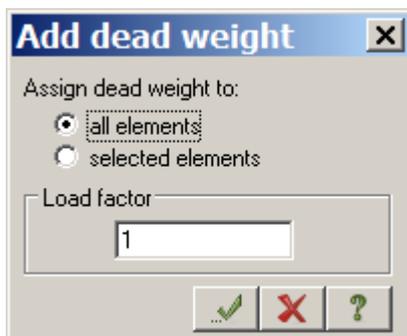
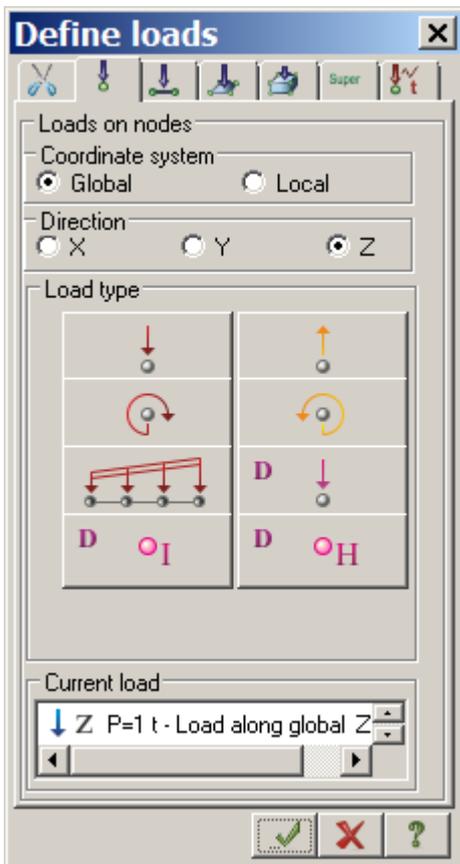


Figure 2.14 **Add dead weight** dialog box

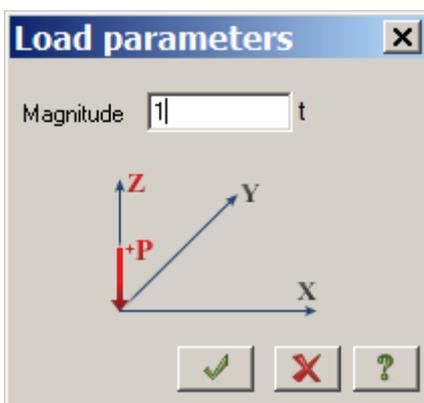
To create load case No.2:

- ⇒ 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 **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- ⇒ Select nodes No.18, 46 and 74 with the pointer.

- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel, then select **Load at nodes** command  from the **Loads on nodes and elements** drop-down list.
- ⇒ In the **Define loads** dialog box (see Fig.2.15), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

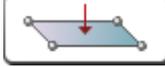
Figure 2.15 **Define loads** dialog box

- ⇒ In the **Load type** area, click **Concentrated load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 1$ t (see Fig.2.16).
- ⇒ Click **OK** .

Figure 2.16 **Load parameters** dialog box

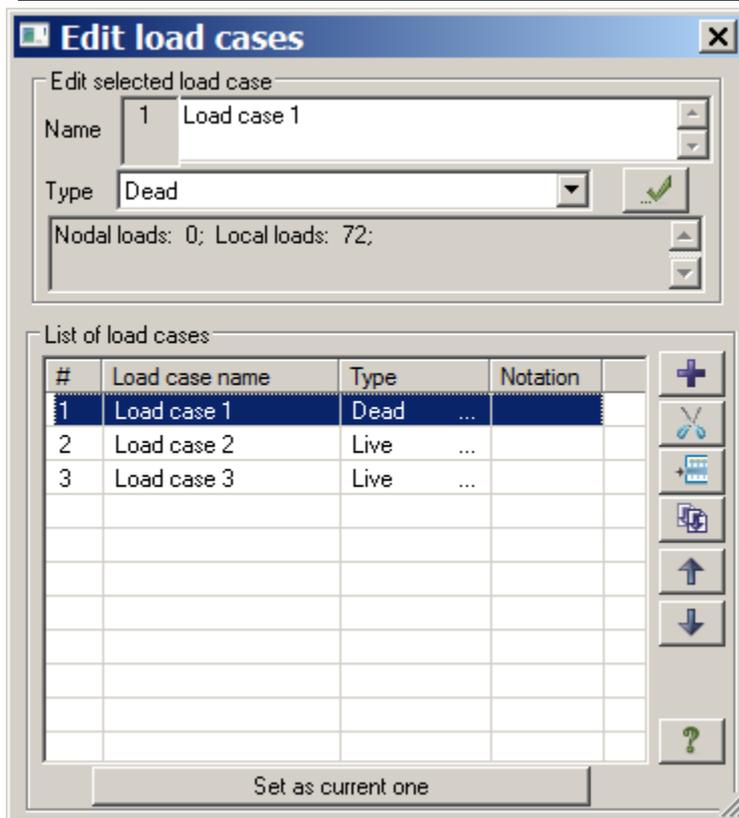
- ⇒ In the **Define loads** dialog box, click **Apply** .

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.
- ⇒ To display element numbers on the screen, on the **Select** toolbar, click **Flags of drawing** button .
- ⇒ In the **Display** dialog box (see Fig.2.4), select the **Element numbers** check box on the **Elements** tab.
- ⇒ Click **Redraw** .
- ⇒ Select elements No.14, 23, 30, 31, 42, 43, 50, 59.
- ⇒ In the **Define loads** dialog box (see Fig.2.15), select **Loads on plates** tab  and specify **Global** coordinate system and direction along the **Z**-axis (default parameters).
- ⇒ In the **Load type** area, click **Concentrated load** button .
- ⇒ In the **Load parameters** dialog box specify the following parameters:
 - $P = 1 \text{ t}$;
 - $A = 0.25 \text{ m}$;
 - $B = 0.25 \text{ m}$.
- ⇒ Click **OK** .
- ⇒ In the **Define loads** dialog box, click **Apply** .

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.2.17).
- ⇒ 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 **Live** and click **Apply** .

Figure 2.17 **Edit load cases** dialog box

Step 7. Generating DCF table

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



As the type of load cases was defined in the **Edit load cases** dialog box (see Fig.2.17), the DCF table is generated automatically with parameters accepted by default for every load case. Now you have to confirm defined parameters.

⇒ To confirm parameters accepted by default for every load case, in the **Design combinations of forces** dialog box (see Fig.2.18), Click **OK** .

Design combinations of forces (DCF)

Building code: SNIP 2.01.07-85* ✓

Load case No.: 3 Load case 3 ?

Load case type: Live (1) ✓ Default...

of group of integrated temporary load cases: 0

Account of sign variability:

of the group of mutually exclusive load cases: 0

of the accompanying load: 0 0

Load factor: 1.20

Duration coefficient: 1.00

Do not consider for analysis by SLS:

Restrictions for cranes and brakes:

Crane Brake

DCF coefficients

#	1 main	2 main	Spec.(E)	Spec.(n)
1	1.00	1.00	0.90	1.00
2	1.00	0.95	0.80	0.95
3	1.00	0.95	0.80	0.95

Summary table for DCF calculation:

L..	Load case na...	DCF parameters	DCF coefficients
1	Load case 1	0 0 0 0 0 0 0 1.10 1.00	1.00 1.00 0.90 1.00
2	Load case 2	1 0 0 0 0 0 0 1.20 1.00	1.00 0.95 0.80 0.95
3	Load case 3	1 0 0 0 0 0 0 1.20 1.00	1.00 0.95 0.80 0.95

Figure 2.18 Design combinations of forces dialog box

Step 8. Complete analysis of slab

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

Step 9. 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.2.19). To display the model without nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button .

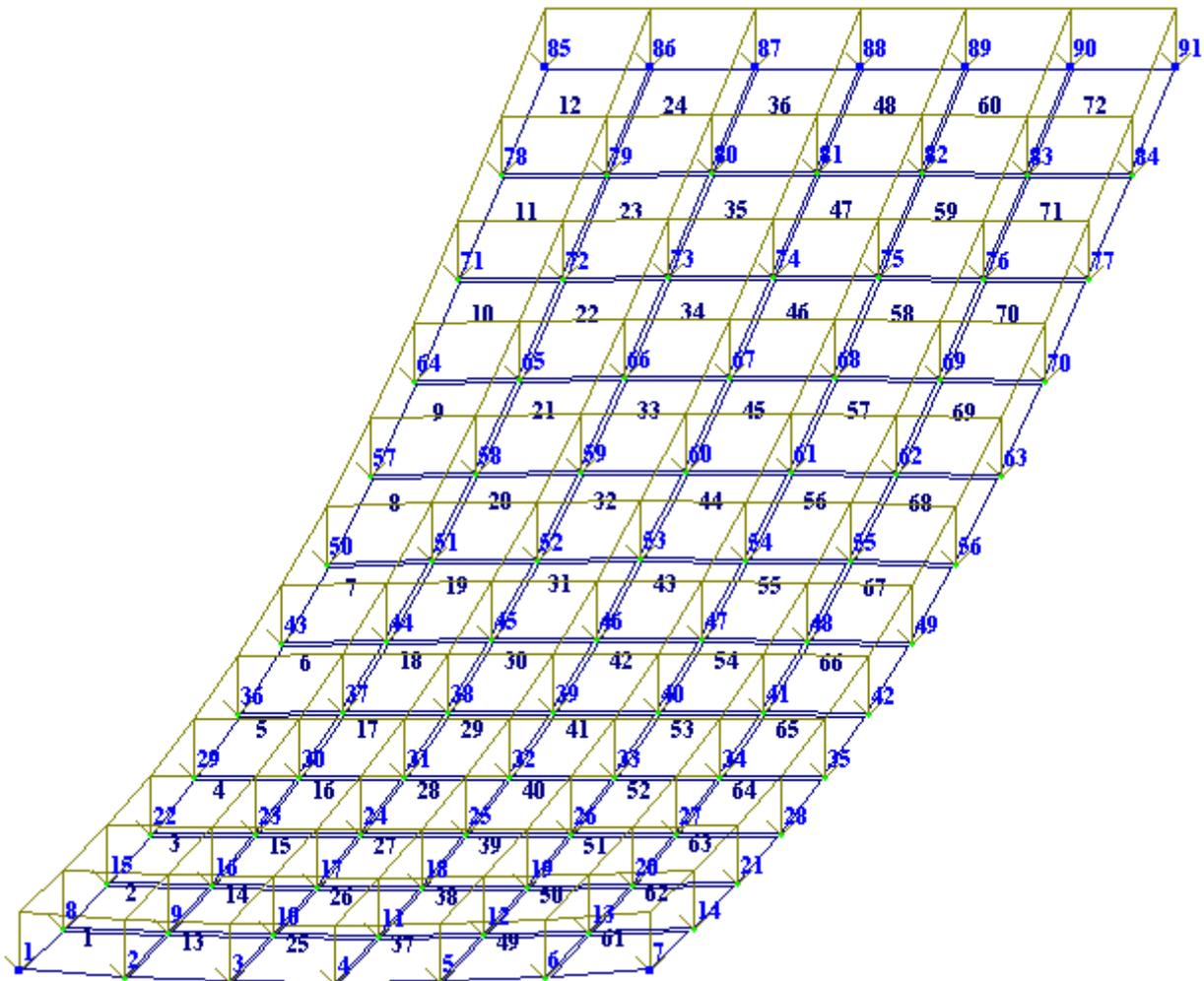


Figure 2.19 Design model with account of nodal displacements

- ⇒ To display the model without element numbers, node numbers and applied loads on the **Select** toolbar, click **Flags of drawing** button .
- ⇒ In the **Display** dialog box (see Fig.2.4), clear the **Element numbers** check box on the **Elements** tab.
- ⇒ In the same dialog box, clear the **Node numbers** check box on the **Nodes** tab.
- ⇒ Then clear the **Loads** check box on the **General** tab.
- ⇒ Click **Redraw** .

To present displacement contour plots:

- ⇒ To present contour plot of displacements along the Z-axis, on the **Results** ribbon tab, on the **Deformations** panel, select the **Displacement mosaic/contour plot in global coordinate system** command  in the **Displacement mosaic/contour plot** drop-down list.
- ⇒ Then click **Displacements along Z** button  on the same panel.

To present stress mosaic plots:

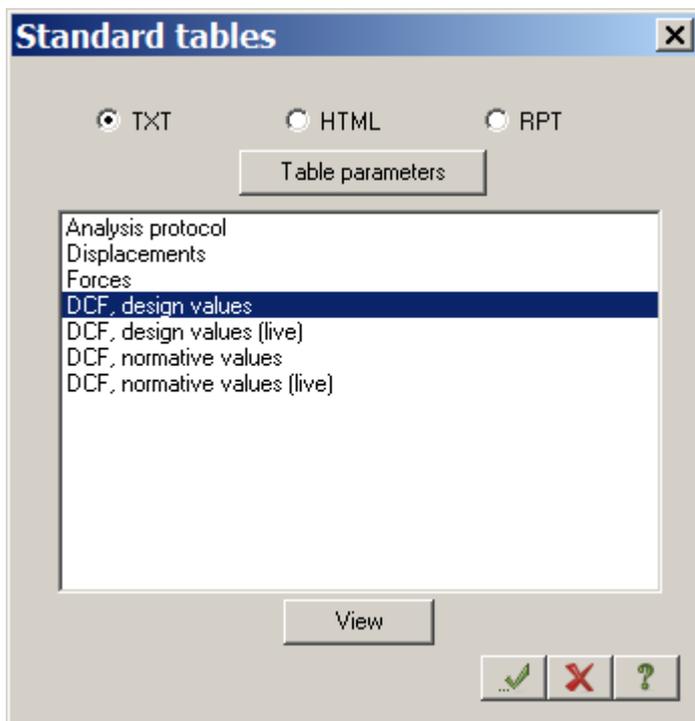
- ⇒ To present stress mosaic plot for M_x , on the **Results** ribbon tab, on the **Stress in plates and solids** panel, select the **Stress mosaic plot** command  in the **Stress mosaic/contour plots** drop-down list.
- ⇒ Then click **Stress M_x** button  on the same panel.
- ⇒ To present stress mosaic plot for M_y , click **Stress M_y** button  on the same panel.

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.2.20), 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 2.20 **Standard tables** dialog box

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

Step 10. 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 (for standard ribbon interface).*

To present results from analysis of reinforcement:

- ⇒ To present information about determined reinforcement in a certain plate element, on the **Select** toolbar, click **Information about nodes and elements** button  and specify with a pointer any plate element on the model.
- ⇒ In the dialog box that appears on the screen, select the **Information about reinforcement** tab. This dialog box contains complete information about selected element, including results for reinforcement.
- ⇒ To close the dialog box, click **Close** button.

- ⇒ To display mosaic plot for area of lower reinforcement in plates along the X1-axis, click the **Lower reinforcement in plates along X1** button  (on the **Design** ribbon tab, the **RC: Plates** panel).
- ⇒ To display mosaic plot for area of lower reinforcement in plates along the Y1-axis, click the **Lower reinforcement in plates along Y1** button  (on the **Design** ribbon tab, the **RC: Plates** panel).

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.2.21), click the **Plates** option in the **Elements** area. The following data is mentioned by default: 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).

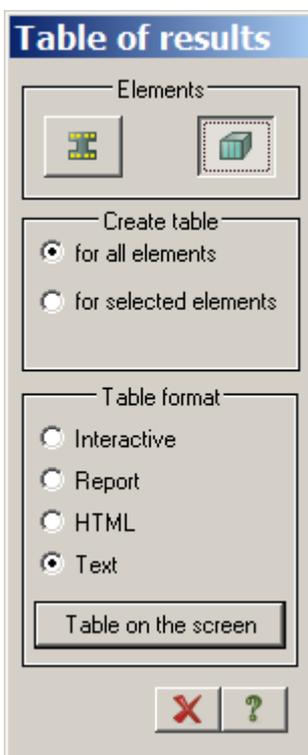


Figure 2.21 **Tables of analysis results** dialog box