Example 18. Analysis of 3D framework of the structure and import of selected reinforcement for further nonlinear analysis

In this lesson you will learn how to:
– carry out static analysis of 3D framework of the structure and calculate DCF;
– unify bar and plate elements;
– analyse reinforcement;
– import reinforcement selected in linear analysis for further nonlinear analysis of structure.

Description:
Two-span three-storey building.
Span lengths – 6 m, column spacing – 6 m, storey height – 3 m.
Columns are fixed at placed where they are supported by foundation slab.
Material – reinforced concrete B25, reinforcement A-III.

Sections of elements:
– columns of the first and the last frames – rectangular section 500 x 500 mm;
– extreme columns of the middle frame – T-section with height 1200 mm (flange width – 1200 mm, flange thickness – 300 mm, web thickness – 300 mm);
– central columns of the middle frame – I-section with height 600 mm (flange width – 600 mm, flange thickness – 200 mm, web thickness – 200 mm);
– floor beam and roof beam – rectangular section 400 x 500 mm;
– floor slab and roof slab – thickness 200 mm.

Loads:
– load case 1 – dead load from elements of the model and building envelope;
– load case 2 – uniformly-distributed p = 0.5 t/m2 applied at floor slabs and roof slab.

Create new problem

⇨   On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2017** and then click **LIRA-SAPR 2017**.

Step 1. Creating new problem

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

⇨   In the **Model type** dialog box (see Fig.18.1) specify the following data:

  ▪ problem name – **Example18**;

  ▪ model type – **5 – Six degrees of freedom per node** (translations X, Y, Z and rotations Ux, Uy, Uz).

⇨   Click **OK** ✅ .

Example 18   Tutorial on LIRA-SAPR version 2017
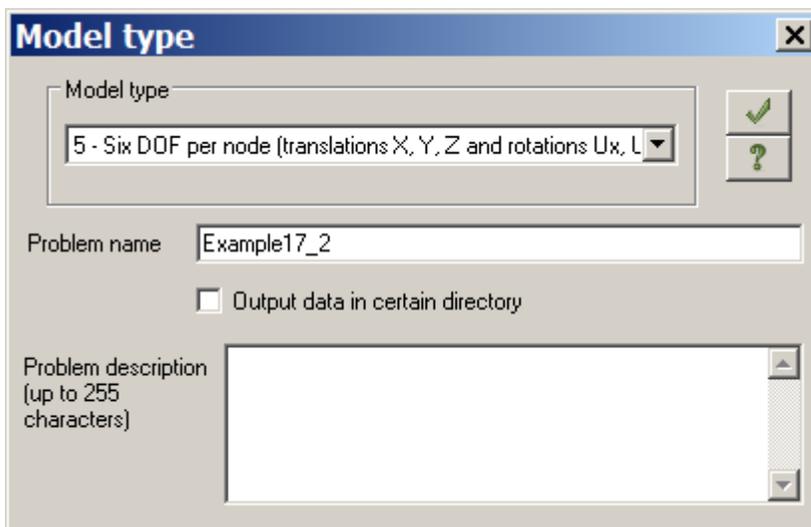
Figure 18.1   **Model type** dialog box

ⓘ   *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.*

Generate model geometry

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.18.2), clear the **Generate foundation slab** check box.

⇨ Then specify the following data for 3D frame:

- spacing along X: spacing along Y: spacing along Z:

L(m)	N	M		L(m)	N	M		L(m)	N	M
6	2	12		6	2	12		3	3	1

- other parameters remain by default.

⇨ Click **Apply** .

3D frame

☑ Specify with pointer

   X |0   m

   Y |0   m

   Z |0   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

|0| Angle of rotation about the Z-axis

X   Value   Number   Mesh

L(m)	#	M
6.00	2	12

Y   Value   Number   Mesh

L(m)	#	M
6.00	2	12

Z   Value   Number   Mesh

L(m)	#	M
3.00	3	1

☑ Intersect with other FE
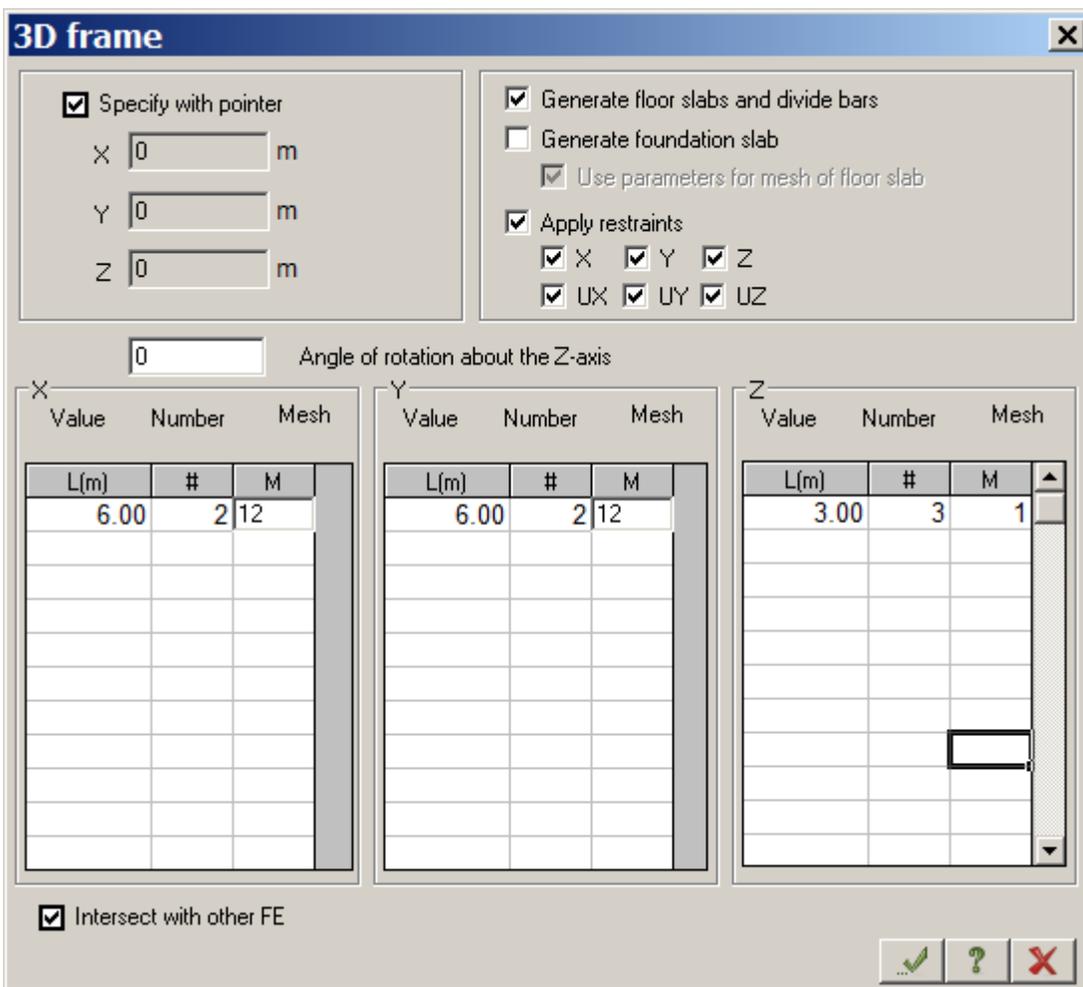
Figure 18.2   **3D frame** dialog box

<u>To pack the model:</u>

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

⇨   In the **Pack model** dialog box (see Fig.18.3), click **Apply** ✓ . It is necessary to pack the model in order to 'throw together' coincident nodes and elements and to eliminate (that is, to remove completely) deleted nodes and elements from design model.

Example 18   Tutorial on LIRA-SAPR version 2017
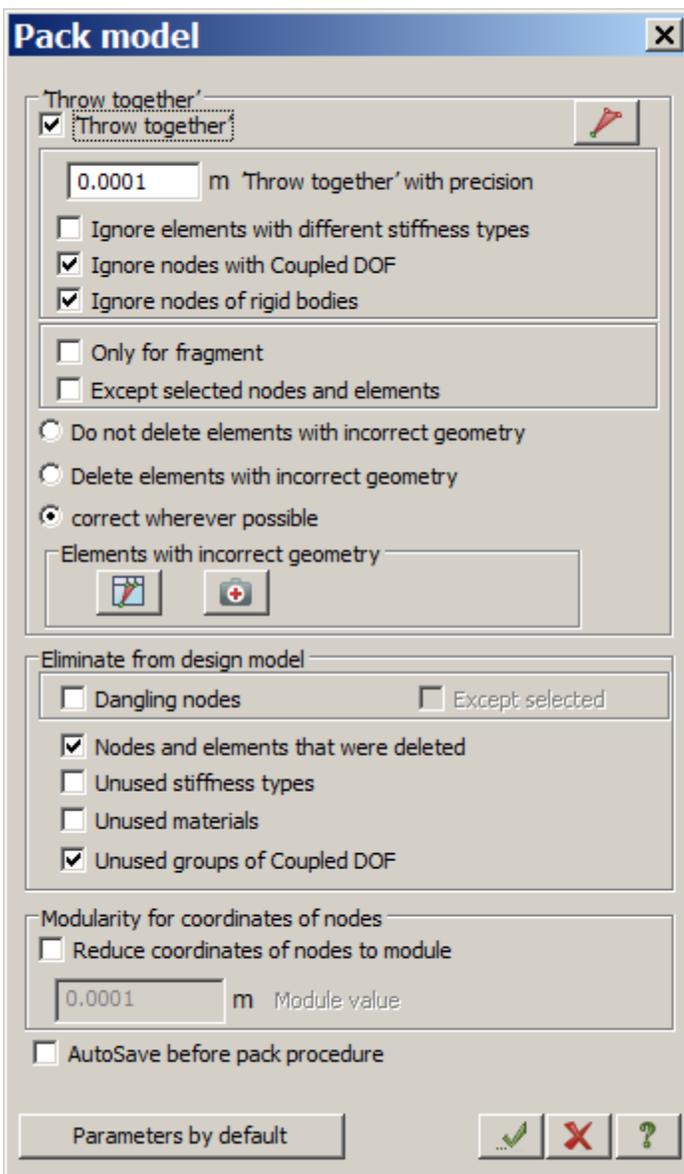
Figure 18.3   **Pack model** dialog box

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 – **Example18**;

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

⇨   Click **Save**.

Define design option

Step 3. Defining design options

⇨ On the **Create and edit** ribbon tab, on the **Design** panel, click **Design options for main model** command .

⇨ In the **Design options** dialog box (see Fig.18.4), 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** .
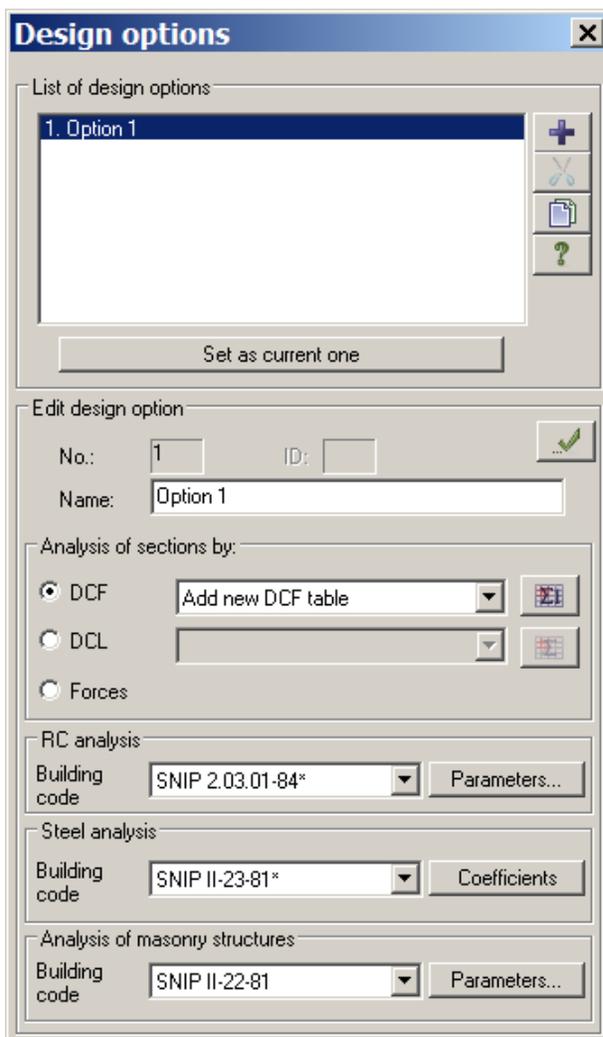
Figure 18.4   **Design options** dialog box

⇨ Close the **Design options** dialog box.

Example 18   Tutorial on LIRA-SAPR version 2017

Define material properties to elements of the model

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 ![button] .

➪   In the **Stiffness and materials** dialog box (see Fig.18.5a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.18.5b).


a	b
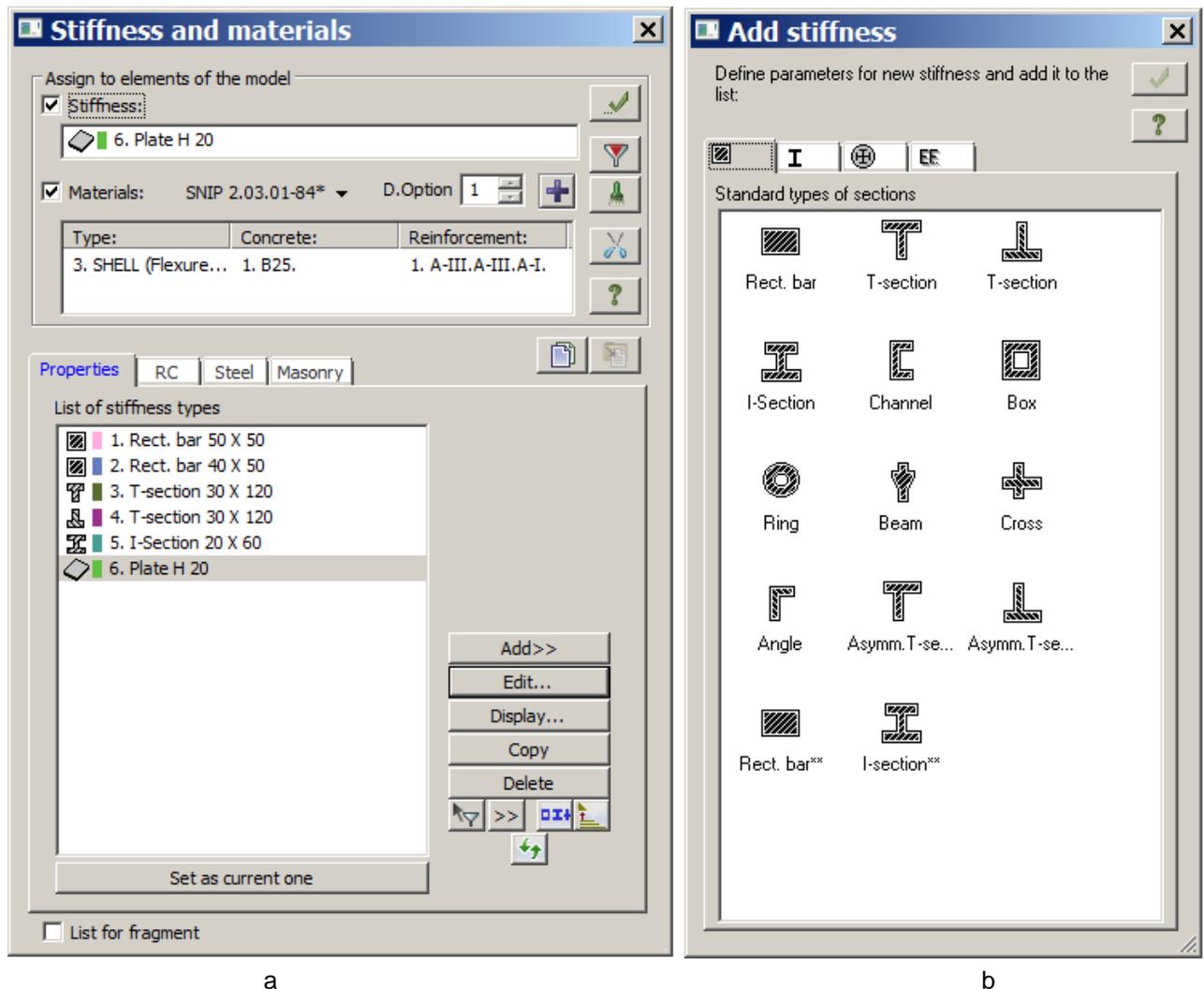
Figure 18.5   Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

➪   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.18.6):

  ▪ modulus of elasticity – E = 3e6 t/m^2 (for the U.S. keyboard layout);

  ▪ geometric properties – B = 50 cm; H = 50 cm;

- unit weight of material – Ro = 2.5 t/m^3.

⇨ To confirm the specified data, click **OK** ![checkmark].
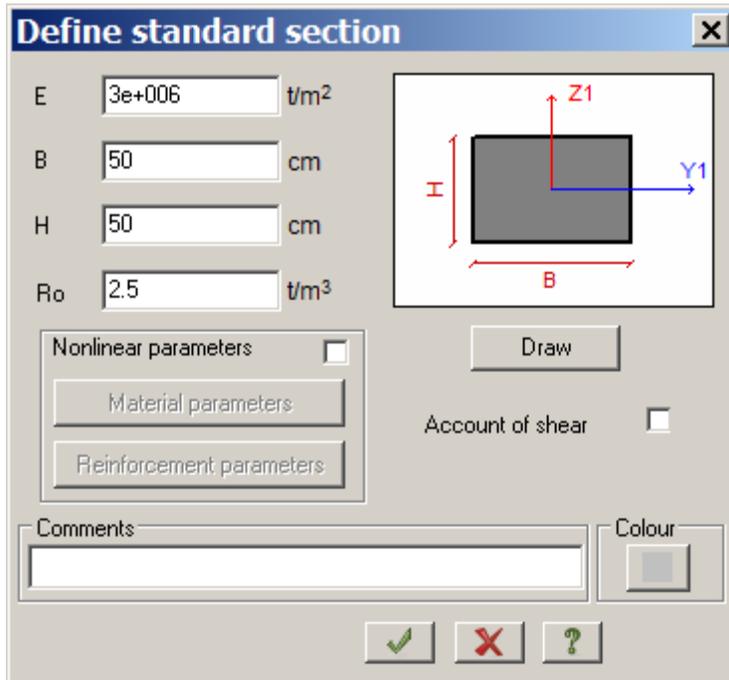
Figure 18.6   **Define standard section** dialog box

⇨ Then in the **Add stiffness** dialog box, double-click the **Rectangular section** icon in the list once again.

⇨ In another **Define standard section** dialog box, specify the following parameters for **Rectangular section**:

- geometric properties – B = 40 cm, H = 50 cm;
- other parameters by default are equal to the latest defined values (as for the previous stiffness).

⇨ To confirm the specified data, click **OK** ![checkmark].

⇨ 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)**:

- geometric properties – B = 30 cm; H = 120 cm; B1 = 120 cm; H1 = 30 cm;
- other parameters by default are equal to the latest defined values.

⇨ To confirm the specified data, click **OK** ![checkmark].

⇨ In the **Add stiffness** dialog box, double-click the **T-section (table at the bottom)** icon in the list.

⇨ In another **Define standard section** dialog box, specify the following parameters for **T-section (table at the bottom)**:

- geometric properties – B = 30 cm; H = 120 cm; B1 = 120 cm; H1 = 30 cm;
- other parameters by default are equal to the latest defined values.

⇨ To confirm the specified data, click **OK** ![checkmark].

Example 18   Tutorial on LIRA-SAPR version 2017

⇨   In the **Add stiffness** dialog box, double-click the **I-section** icon in the list.

⇨   In another **Define standard section** dialog box, specify the following parameters for **I-section**:

  ▪   geometric properties – B = 20 cm; H = 60 cm; B1 = 60 cm; H1 = 20 cm; B2 = 60 cm; H2 = 20 cm;

  ▪   other parameters by default are equal to the latest defined values.

⇨   To confirm the specified data, click **OK** ✓ .

⇨   In the **Add stiffness** dialog box (see Fig.18.5b), select the **Plates, solids, numerical** tab (the fourth tab) and double-click the **Plates** icon in the list.

⇨   In the **Stiffness for plates** dialog box (see Fig.18.7), specify the following parameters for **Plate** (floor slab):

  ▪   Poisson's ratio – v = 0.2;

  ▪   thickness – H = 20 cm.

⇨   To confirm the specified data, click **OK** ✓ .
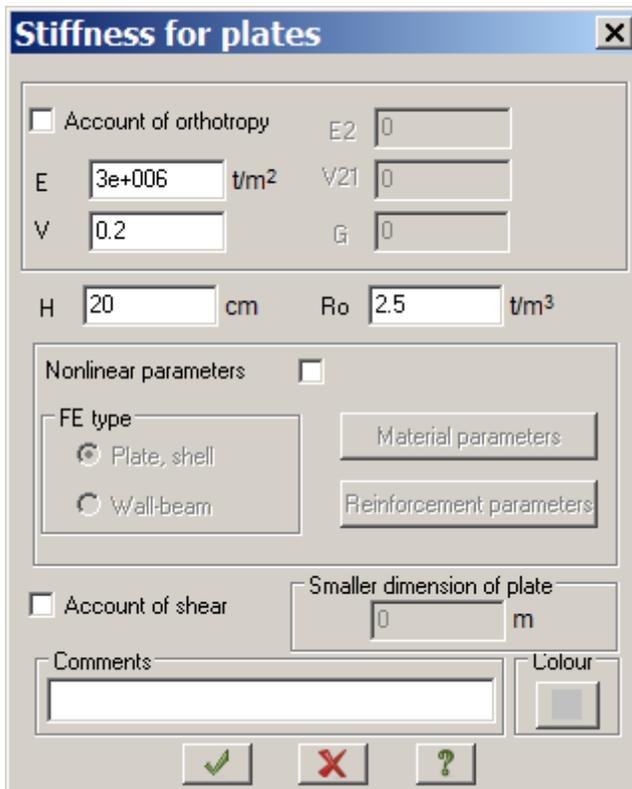
Figure 18.7   **Stiffness for plates** dialog box

⇨   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 **Edit**.

⇨   In the **Material properties for analysis of RC structures** dialog box (see Fig.18.8), select the **Type: Bar** in the first line and then in the right part of the dialog box define the following parameters:

- in the **Name** box, type **Columns**;

- in the **Analysis type** list, select **Column regular**;

- in the **Reinforcement** list, select **Symmetric**;

- in the **Analysis** area, select the **Design requirements** check box;

- in the **Analysis by serviceability limit states (SLS)** area, select the **Diameter of rebars** option and select the value 40mm in the appropriate list;

- in the **Length of element, Effective lengths** area, select the **Factor** option;

- define parameters LY=0.7, LZ=0.7;

- other parameters remain by default.

Figure 18.8   **Material properties for analysis of RC structures** dialog box

⇨ To add new row for parameters of bar elements, click the **Add** button and then in the right part of the dialog box define the following data for beams:

- in the **Name** box, type **Beams**;

- in the **Analysis type** list, select **Beam**;

- in the **Reinforcement** list, select **Asymmetric**;

- in the **Analysis** area, select the **Design requirements** check box;

- in the **Analysis by serviceability limit states (SLS)** area, select the **Diameter of rebars** option and select the value 40mm in the appropriate list;

- in the **Length of element, Effective lengths** area, define parameters LY=0, LZ=0;

- other parameters remain by default.

⇨ To add the row for parameters of plate elements, click the first row in the **Type: Plate** list. Then in the right part of the dialog box define the following parameters for floor slabs and roof slab:

- in the **Name** box, type **Slabs**;

- in the **Analysis type** list, select **Shell**;

www.liraland.com

Example 18   Tutorial on LIRA-SAPR version 2017

- in the **Distance to gravity centre of reinforcement** area, define A1Y = 4 cm, A2Y = 4 cm;

- other parameters remain by default.

⇨ Then click the first row in the **Concrete** list and in the right part of the dialog box define the following parameters:

  - in the **Class of concrete** list, select the row B25;

  - other parameters remain by default.

⇨ Then click the first row in the **Reinforcement** list and in the right part of the dialog box define the following parameters:

  - in the **Transverse reinforcement** list, select the row A-I;

  - in the **Max diameter of longitudinal reinforcement** list, select 40;

  - other parameters remain by default.

  ⇨ To confirm the specified data, click **OK** ✅ .

To assign material properties to elements of the model:

⇨ In the **Stiffness and materials** dialog box, in the **Assign to elements of the model** area, select the **Materials** check box.

⇨ Make sure that stiffness '**6.Plate H20**' is defined as current one and type of materials '**3.Shell**', class of concrete **1.B25** and class of concrete **1.A-III** are also defined as current ones.

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

⇨ 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** ✅ . In the **Warning** box (see Fig.18.9), click **OK**.
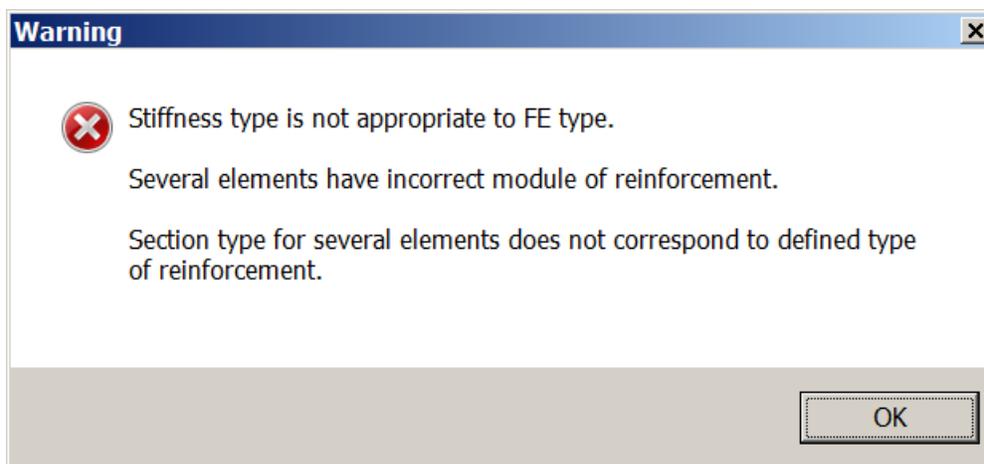

Figure 18.9   **Warning** box

⇨ In the **Stiffness and materials** dialog box, on the **RC** tab, in the list of material parameters for RC structures, click the row '**2. Beam. Beams**'.

⇨ Click **Set as current type**. In this case selected type will be displayed in the **Materials** 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.

⇨ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the row '**2.Rectangular bar 40x50**'.

⇨ Click **Set as current type**. Selected type will be displayed in the **Stiffness** box in the **Assign to elements of the model** area.

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

⇨ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the stiffness type '**2.Rect. bar 50x50**'.

⇨ Click **Set as current type**.

⇨ In the **Stiffness and materials** dialog box, on the **RC** tab, in the list of material parameters for RC structures, click the stiffness type '**1. Column regular. Columns**'.

⇨ Click **Set as current type**.

⇨ 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, click **Apply** .

⇨ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the stiffness type '**3.T-section (table at the top) 30x120**'.

⇨ With the pointer, select only columns (from the 1st up to the 3rd storey) located along the right edge of middle frame in the framework.

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

⇨ In the **Warning** box, click **Yes**.

⇨ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the stiffness type '**4.T-section (table at the bottom) 30x120**'.

⇨ With the pointer, select only columns (from the 1st up to the 3rd storey) located along the left edge of middle frame in the framework.

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

⇨ In the **Warning** box, click **Yes**.

⇨ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the stiffness type '**5.I-section 20x60**'.

⇨ With the pointer, select only columns (from the 1st up to the 3rd storey) located at the centre of middle frame in the framework.

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

⇨ In the **Warning** box, click **Yes**.

⇨ On the **Select** menu, click **Select vertical bars** in order to make this command not active.

Example 18   Tutorial on LIRA-SAPR version 2017

Apply loads

Step 5. Applying loads

To 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.18.10), click **All elements** and specify **Load factor** as equal to **1.1**.

⇨ Then click **Apply** (dead weight is automatically applied to elements).
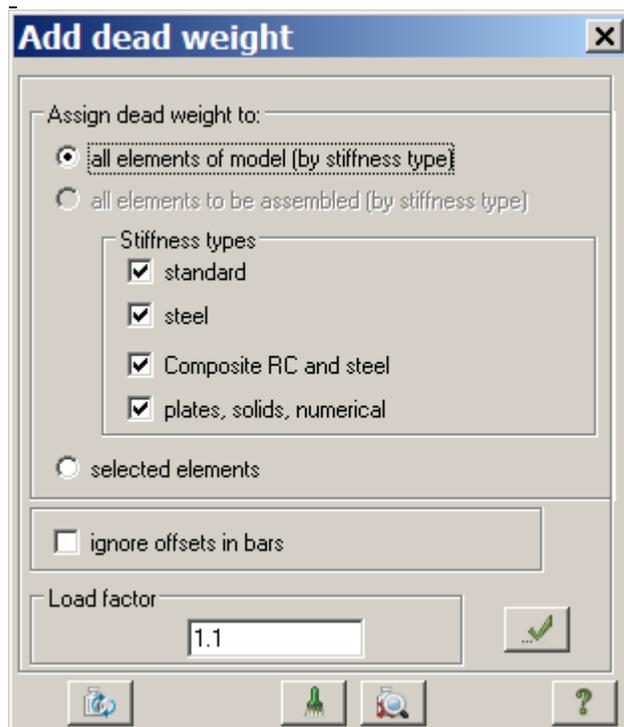
Figure 18.10   **Add dead weight** dialog box

⇨ On the **Select** toolbar, click **Select horizontal bars** button .

⇨ To switch to projection on the XOY-plane, on the **Projection** toolbar, click **Projection on XOY-plane** .

⇨ With selection window, select all elements of beams only along perimeter of the model.

⇨ 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.18.11), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).
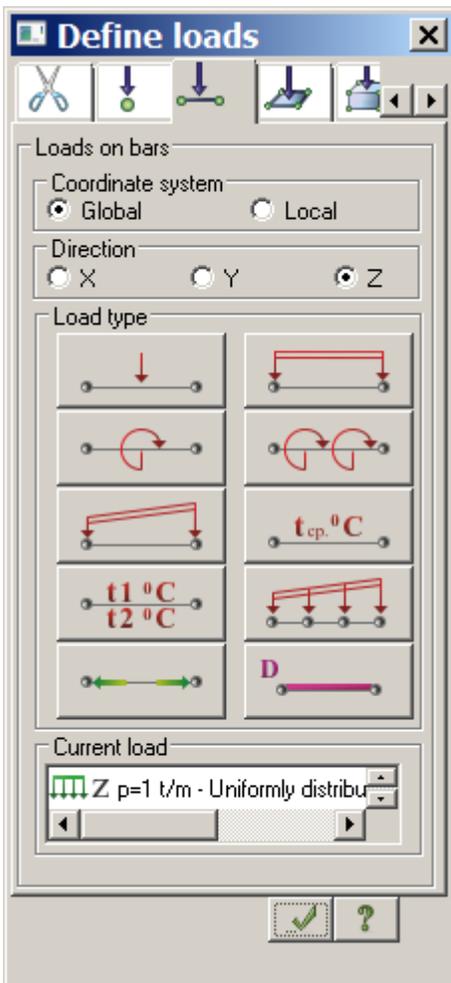
Figure 18.11   **Define loads** dialog box

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

⇨   In the **Load parameters** dialog box specify P = 1.0 t/m (see Fig.18.12).

⇨   Click **OK** [image] .
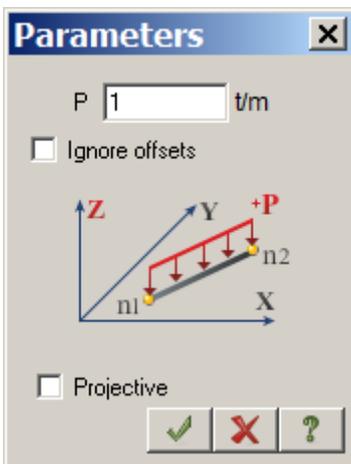
Figure 18.12   **Load parameters** dialog box

Example 18   Tutorial on LIRA-SAPR version 2017

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

To create load case No.2:

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

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

⇨   Select all elements of the model with the pointer.

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

⇨   In the **Load type** area, click **Uniformly distributed load** button .
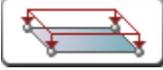
⇨   In the **Load parameters** dialog box specify P = 0.5 t/m^2 .

⇨   In the **Warning** box, click **OK** .
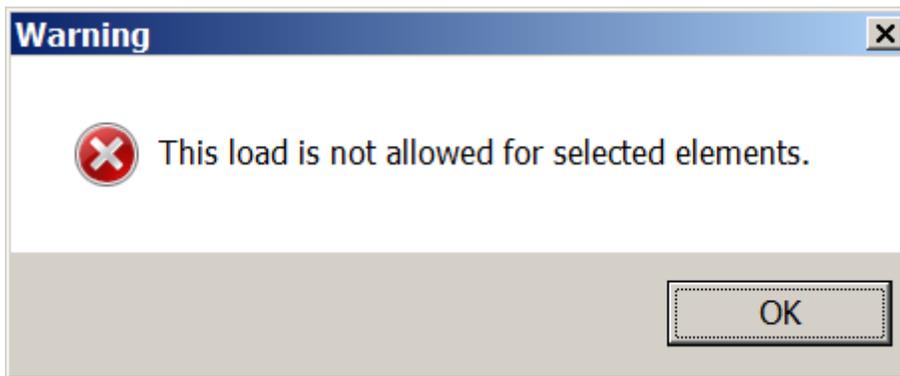
Figure 18.13   **Warning** box

ⓘ
         The warning message appears because when you select all elements of the model, bars and plates are selected at the same time. Load applied to plates is not allowed for bar elements.

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

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.21.20).

⇨   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** .
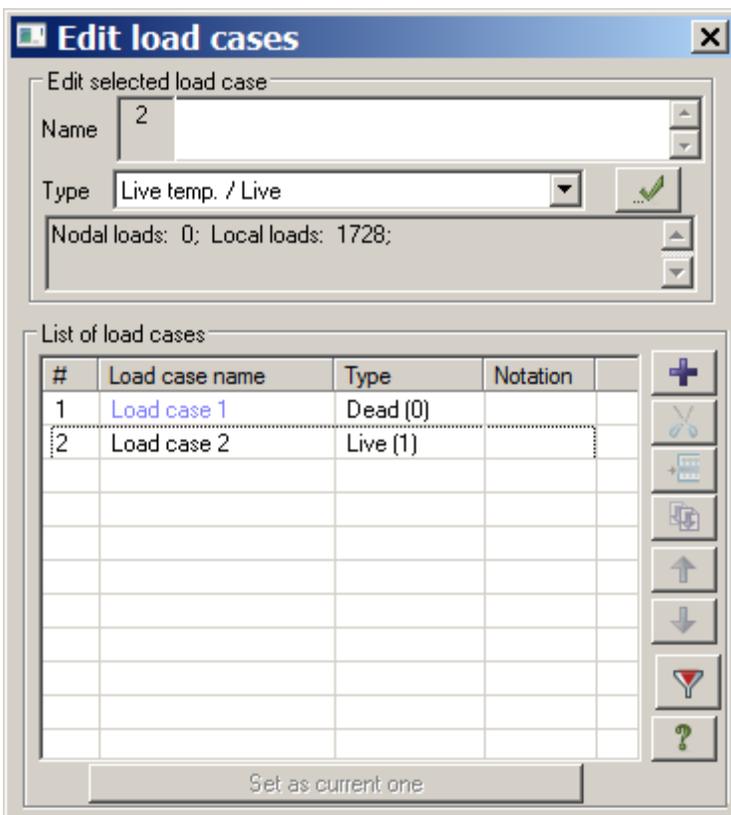
Figure 18.14   **Edit load cases** dialog box

⇨   Close the **Edit load cases** dialog box.

Generate DCF table

Step 6. Generating DCF table

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

⇨   To fill in the DCF table with values accepted by default for every load case, click the **Fill in DCF table with default values** button .

⇨   To confirm default values, click **OK** .

Example 18   Tutorial on LIRA-SAPR version 2017
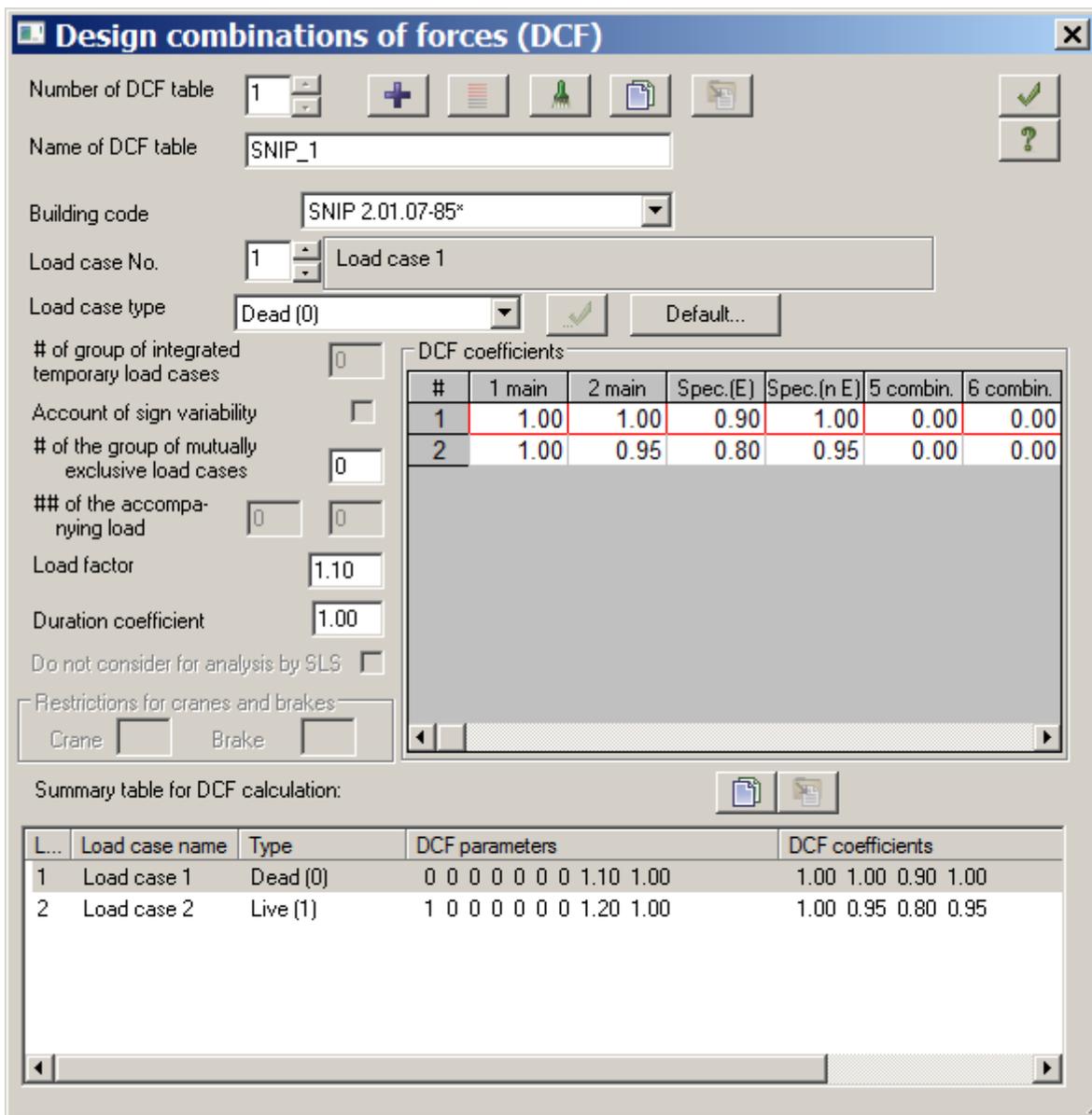
Figure 18.15   **Design combinations of forces** dialog box

Unify elements

Step 7. Unifying elements

To hide presentation of loads on design model:

⇨ On the **Select** toolbar, click **Flags of drawing** button ⬚ . In the **Display** dialog box (see Fig.18.16), clear the **Loads** check box on the **General** tab.

⇨ Click **Redraw** ⬚ .
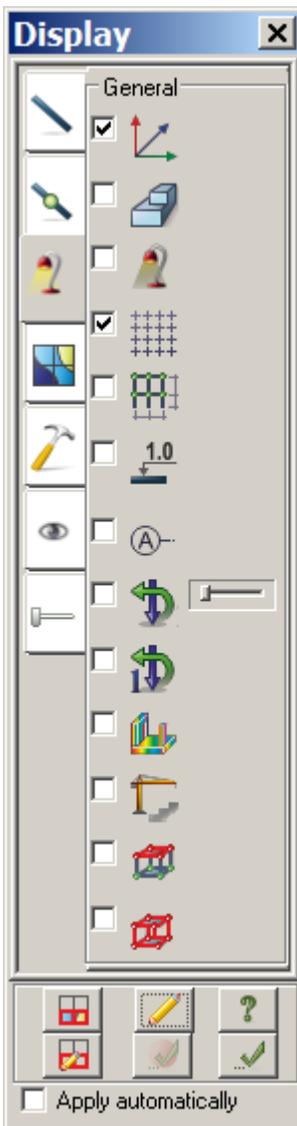
Figure 18.16   **Display** dialog box

To unify elements:

ⓘ   *In this example, for all elements of the model the first type of unification will be applied (**Single section for the whole group**). To speed up unification process, every 'cell' of the model will be divided into three parts along all directions.*

⇨   To switch to projection on the XOY-plane, on the **Projection** toolbar, click **Projection on XOY-plane**.

⇨   On the **Create and edit** ribbon tab, select the **Design** panel, click **Unify elements** command. Appropriate dialog box (see Fig.18.17) appears on the screen.

Example 18   Tutorial on LIRA-SAPR version 2017
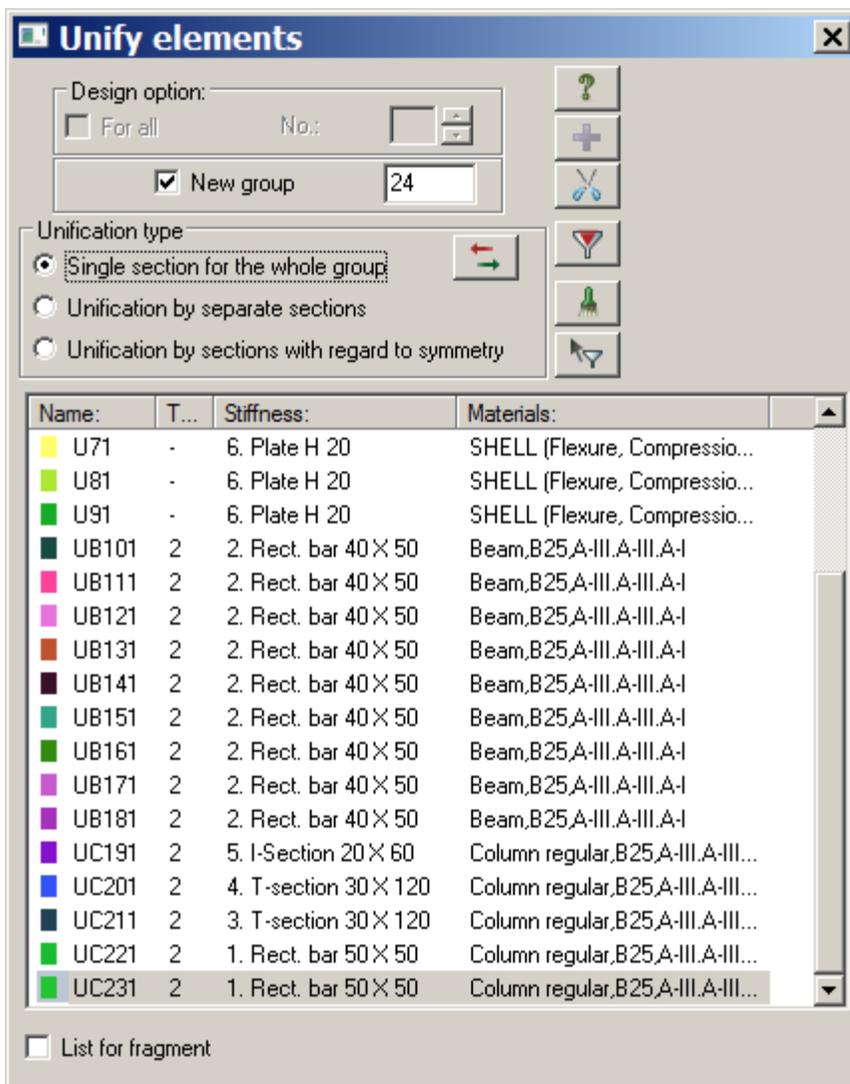
Figure 18.17   **Unify elements** dialog box

⇨   To select certain elements on the model, on the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **PolyFilter** ▽ .

⇨   In the **PolyFilter** dialog box (see Fig.18.18), select the **Filter for elements** tab (the second tab).

⇨   Select **By Fe shape** check box and specify the line '4-node FE (plates)'.

⇨   Click **Apply** ✓ .
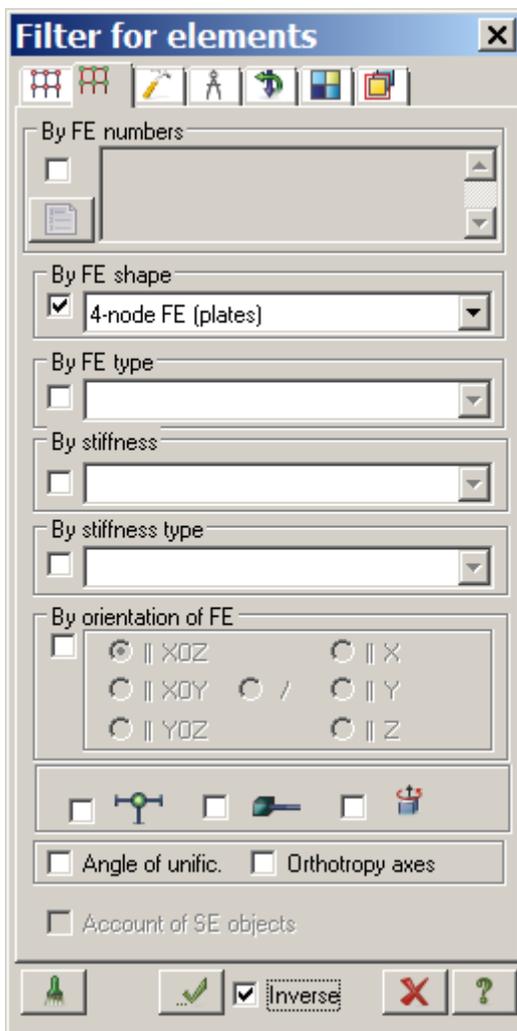
Figure 18.18   **Filter for elements** dialog box

⇨   On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button [icon] .
     With 'selection window' select elements of bearing of the floor slabs above the central column in the
     middle frame: the square that includes (from location of the column) 4 finite elements to one side and
     another side along the X-axis and Y-axis.

⇨   In the **Unify elements** dialog box (see Fig.18.17), in the **Unification type** area, select the **Single
     section for the whole group** option. To define the first group of unification, click the **Crete new UG or ...**
     button [icon] .

⇨   Then with 'selection window' select elements of the middle part of floor slabs between the central column
     of middle frame and central columns of the 1st and the 3rd frames: 2 rectangles that include 4 finite
     elements to one side and another side along the X-axis from beams that connect these columns and 4
     finite elements along the Y-axis.

⇨   In the **Unify elements** dialog box, make sure that the **Single section for the whole group** option is
     selected (in the **Unification type** area) and **New group** check box is selected. To define the second
     group of unification, click the **Crete new UG or ...** button [icon] .

⇨   With 'selection window' select elements of bearing of the floor slabs near the central columns of the 1st
     and the 3rd frames: 2 rectangles that include 4 finite elements to one side and another side along the X-
     axis from columns and 4 finite elements along the Y-axis from these columns to the middle of the
     building.

Example 18   Tutorial on LIRA-SAPR version 2017

⇨ In the **Unify elements** dialog box, make sure that the **Single section for the whole group** option is selected (in the **Unification type** area) and **New group** check box is selected. To define the third group of unification, click the **Crete new UG or ...** button .

⇨ With 'selection window' select elements of bearing of the floor slabs in the middle between central and extreme columns of the 1st and the 3rd frames: 4 squares that include 4 finite elements along beams located in midspans of the 1st and the 3rd frames along the X-axis and 4 finite elements along the Y-axis from these beams to the middle of the building.

⇨ In the **Unify elements** dialog box, click the **Crete new UG or ...** button .

⇨ In a similar way, define the following unification groups:

▪ unification group 5: elements of bearing of floor slabs near extreme columns of the 1st and the 3rd frames (4 squares that include 4 finite elements from these columns along the X-axis and the Y-axis);

▪ unification group 6: elements of bearing of floor slabs in midspans of the 1st and the 2nd spans of the middle frame (2 rectangles that include 4 finite elements to one side and another side along the Y-axis from beams of these spans and 4 finite elements along the X-axis in midspans);

▪ unification group 7: elements of bearing of floor slabs near extreme columns of the middle frame (2 rectangles that include 4 finite elements to one side and another side along the Y-axis from these columns and 4 finite elements along the X-axis towards the middle of the building);

▪ unification group 8: elements of central parts of floor slabs of every from four parts enveloped by beams (4 squares that include 4 finite elements along the X-axis and the Y-axis; elements located at the distance 2m from each beam);

▪ unification group 9: elements of bearing of floor slabs in the middle of enveloping beams between frames (4 squares that include 4 finite elements along the X-axis from these beams towards the middle of the building and 4 finite elements along the Y-axis in the middle of beams).

⇨ On the **Select** toolbar, click **Select horizontal bars** button .

⇨ With 'selection window' select two extreme elements in each of four beams that are supported by the central column of the middle frame (bearing onto the central column of the middle frame).

⇨ In the **Unify elements** dialog box, click the **Crete new UG or ...** button .

⇨ With 'selection window' select two extreme elements in each of four beams that are supported by the central column of the middle frame (bearing onto the central columns of the 1st and the 3rd frames and bearing onto extreme left and extreme right columns of the middle frame).

⇨ In the **Unify elements** dialog box, click the **Crete new UG or ...** button .

⇨ In a similar way, define the following unification groups:

▪ unification group 12: the 3rd and the 4th elements from each of four beams that are supported by the central column of the middle frame (elements near the bearing onto the central column);

▪ unification group 13: the 5th, 6th and 7th elements from each of four beams that are supported by the central column of the middle frame (measured from the central column);

▪ unification group 14: the 3rd, 4th and 5th elements from each of four beams that are supported by the central column of the middle frame (measured from the central columns of the 1st and 3rd frames and from extreme left and extreme right columns of the middle frame);

▪ unification group 15: three extreme elements from each of enveloping beams located along perimeter (bearing of these beams onto the central columns of the 1st and the 3rd frames and onto extreme left and extreme right columns of the middle frame);

- unification group 16: the 4th, 5th and 6th elements from each of enveloping beams located along perimeter (measured from the central columns of the 1st and the 3rd frames and from extreme left and extreme right columns of the middle frame);

- unification group 17: the 4th, 5th and 6th elements from each of enveloping beams located along perimeter (measured from four corner columns);

- unification group 18: three extreme elements from each of enveloping beams located along perimeter (bearing of these beams onto corner columns).

⇨ On the **Select** toolbar, click **Select horizontal 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** 📐 .

⇨ On the **Select** toolbar, click **Select vertical bars** button 🔍 .

⇨ With 'selection window' select only elements of the central columns in the middle frame.

⇨ In the **Unify elements** dialog box, click the **Crete new UG or ...** button ➕ .

⇨ In a similar way, define the following unification groups:

- unification group 20: only elements of extreme left columns of the middle frame;

- unification group 21: only elements of extreme right columns of the middle frame;

- unification group 22: only elements of the central columns of the 1st and the 3rd frames;

- unification group 23: only elements of all corner columns.

Complete analysis of the model

Step 8. Complete analysis of the model

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

Static analysis: review and evaluation of analysis results

Step 9. Review and evaluation of static analysis results

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

⇨ In the mode of analysis results visualization, by default design model is presented with account of nodal displacements.

To present diagrams of internal forces:

⇨ On the **Select** toolbar, click **PolyFilter** 🔽 .

⇨ In the **PolyFilter** dialog box (see Fig.18.18), select the **Filter for elements** tab (the second tab).

Example 18   Tutorial on LIRA-SAPR version 2017

⇨   Click **By FE shape** option and select **Bars** in the list.

⇨   Click **Apply** .

⇨   To present on the screen only selected bars, on the **Select** toolbar, click **Fragmentation** .

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

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 Mx, 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 Mx** button on the same panel.

⇨   To present stress mosaic plot for Nx, click **Stress Nx** button on the same panel.

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.18.19), select **Design combinations of forces, design values** in the list.

⇨   Click **Apply**.

(i)
   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.*
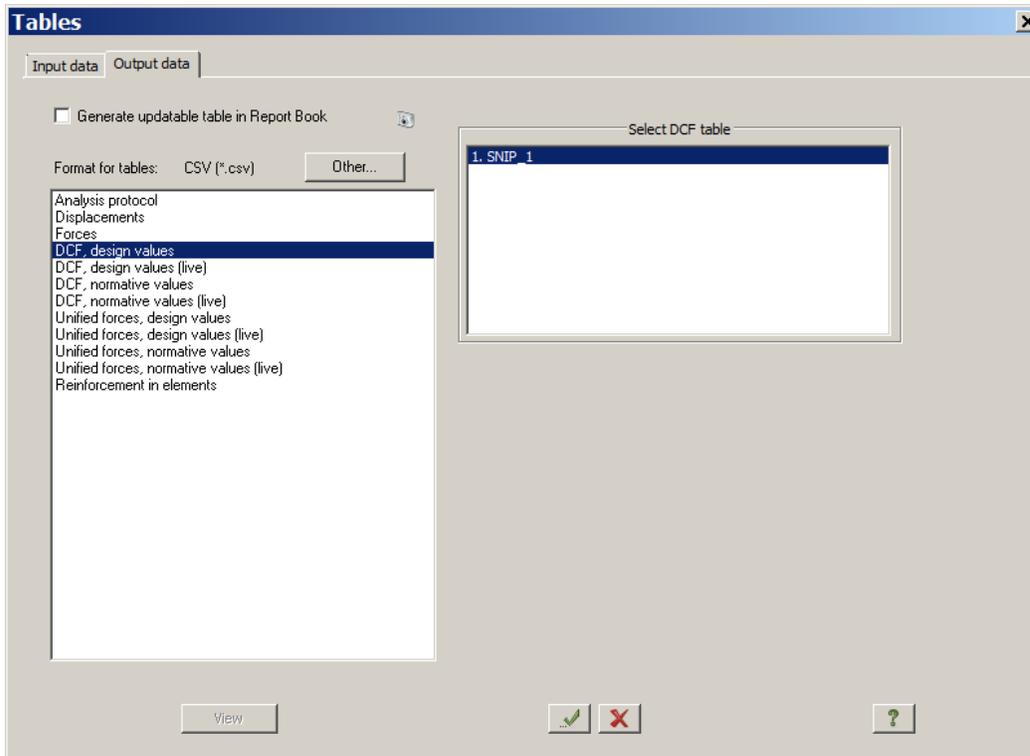
Figure 18.19   **Standard tables** dialog box

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

Analysis of reinforcement: review and evaluation of analysis results

Step 11. Review and evaluate results from analysis of reinforcement

(i)
   When analysis procedure is complete, to review and evaluate analysis results for reinforcement, select the **RC** *ribbon tab (in the* **Ribbon plus** *style).*

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 bar or plate element on the model.

Example 18   Tutorial on LIRA-SAPR version 2017

⇨ 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 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** dialog box, the following data is mentioned by default: **Reinforcement in elements** option in selected in the list, the **in bars** option is selected under the **Reinforcement**.

⇨ To generate the table with analysis results for reinforcement in bars, click **Apply** .

Import of analysis results of reinforcement for generation of nonlinear problem

Step 12. Import of analysis results of reinforcement for generation of nonlinear problem

⇨ On the **RC** ribbon tab, on the **Design** panel, click **Material properties** button .

⇨ In the **Stiffness and materials** dialog box, click the **Replace stiffness with data from ARM-SAPR and STC-SAPR modules** button .

⇨ In the Replace stiffness dialog box (see Fig.18.20), to define nonlinear problem, click **OK** .
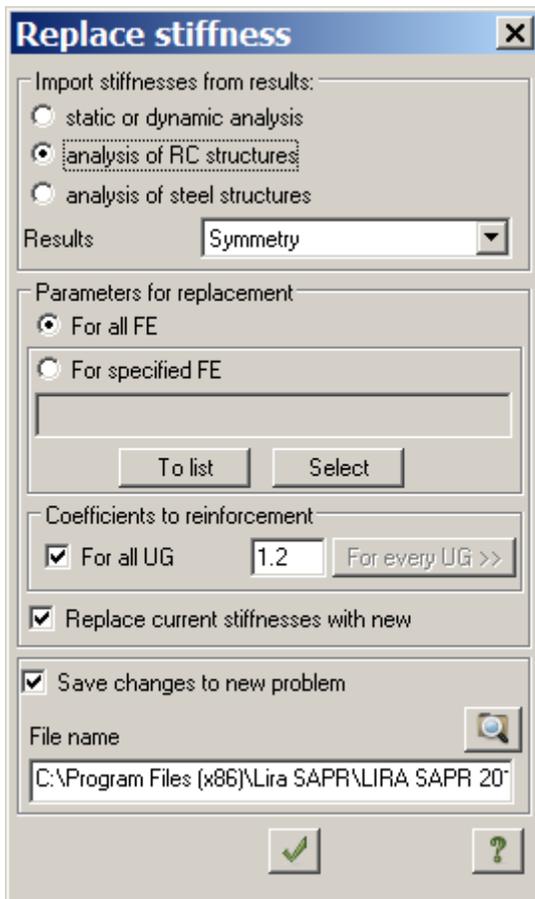
Figure 18.20   **Stiffness and material** dialog box

To complete generation of nonlinear problem:

(i)   *When analysis results of reinforcement are imported to a new file, types of finite elements are automatically changed to physically nonlinear ones. Stiffness parameters are also changed automatically with account of nonlinearity (material properties and parameters of reinforcement are defined automatically).*

⇨   When the nonlinear problem is defined, to complete generation of design model, specify the following data:

  ▪ delete the dead weight and define it ones again with load factor equal to 1 (normative value);

  ▪ edit loads and replace design values of loads from the enveloping structures and the load in the second load case with the normative values (divide by appropriate load factors);

  ▪ delete the DCF table;

  ▪ according to description in Example 7, generate the table for modelling nonlinear loads.

⇨   When design model is complete, you could carry out the physically nonlinear analysis of the model and evaluate analysis results.