

Example 21. Analysis of 3D framework with different design options for RC structures

In this lesson you will learn how to:

- generate design model;
- deal with design options, that is, vary material properties, building codes, section dimensions;
- analyse reinforcement for elements of the framework;
- generate DCF table.

Description:

One storey building with two spans. Span dimensions – 7,5 m, column step – 7 m, storey height – 4 m. Columns are fixed at places where they connect the base slab.

Element sections:

beams – T-section of height 500 mm (flange width – 500 mm, flange thickness – 200 mm, web thickness – 300 mm);
columns – rectangular section with dimension 400 x 400 mm;
floor slabs – thickness 200 mm.

Material:

for design option 1 – reinforced concrete B25, reinforcement A-III (SNIP 2.03.01-84*), symmetric reinforcement in columns;
for design option 2 – reinforced concrete B25, reinforcement A-III (SNIP 2.03.01-84*), asymmetric reinforcement in columns;
for design option 3 – reinforced concrete B25, reinforcement A-III (SNIP 2.03.01-84*), symmetric reinforcement in columns, corner rebars are not selected;
for design option 4 – reinforced concrete B25, reinforcement A400 (SP 63.13330.2012), symmetric reinforcement in columns;
for design option 5 – reinforced concrete B30 (beams and columns), B25 (roof slab), reinforcement A-III (SNIP 2.03.01-84*), symmetric reinforcement in columns.

Loads:

load case 1 – dead weight;
load case 2 – dead uniformly distributed load $P = 1.5 \text{ t/m}^2$ applied to roof;
load case 3 – snow load $p = 0.35 \text{ t/m}^2$;
load case 4 – wind load along X;
load case 5 – wind load along Y.

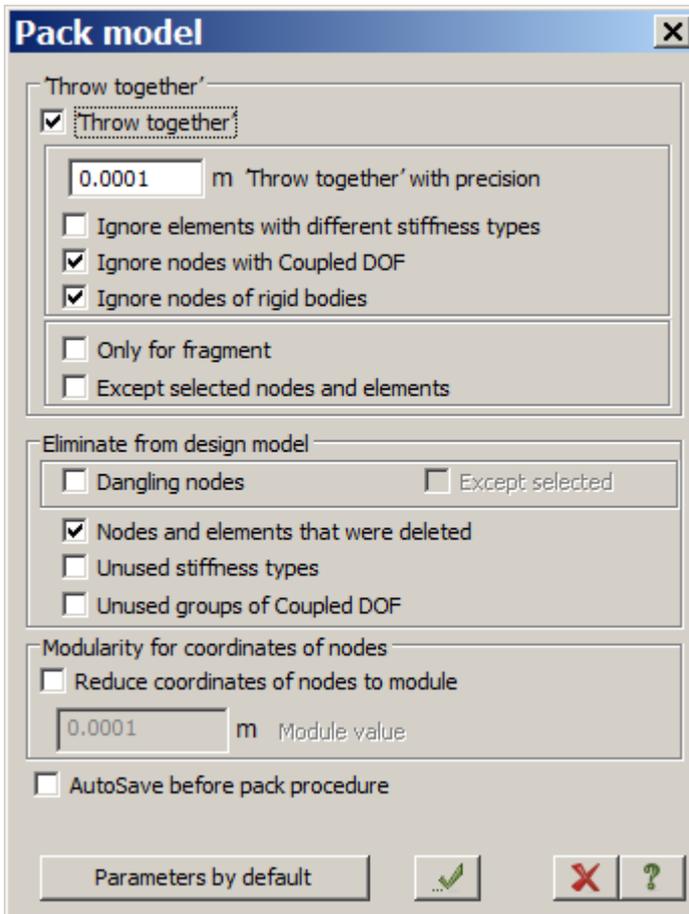


Figure 21.3 Pack model dialog box

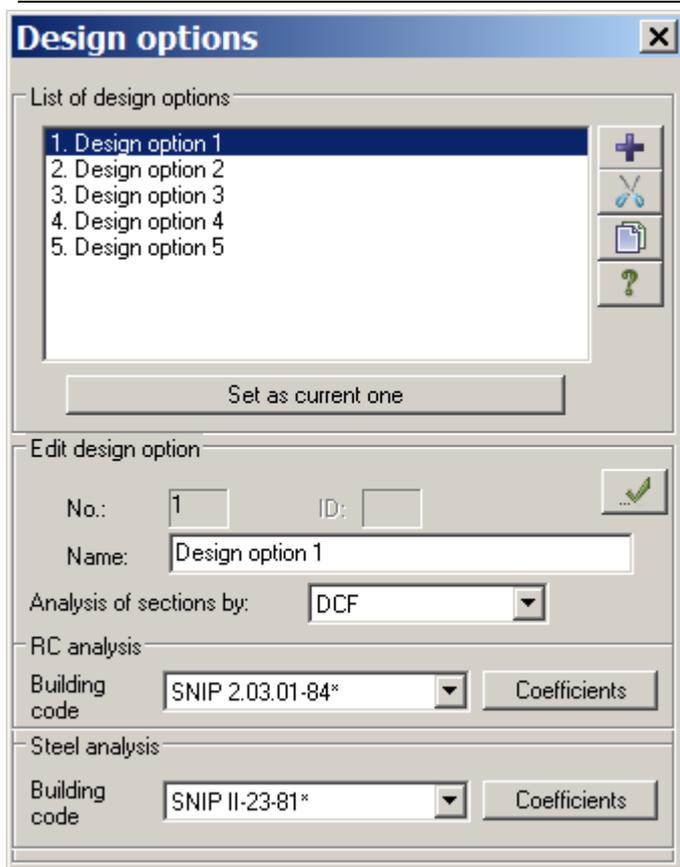
To save data about design model:

- ⇒ On the **LIRA-SAPR** menu (Application menu), click **Save** command .
- ⇒ In the **Save as** dialog box specify the following data:
 - file name – **Example21**;
 - location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

Step 3. Defining design options

To define the first design option:

- ⇒ On the **More edit options** ribbon tab, on the **Design** panel, click **Design options for main model** command .
- ⇒ In the **Design options** dialog box (see Fig.21.4), define parameters for the first design option:
 - make sure that **SNIP 2.03.01-84*** is selected for RC analysis;
 - in the **Analysis of sections by** list, select **DCF**;
 - other parameters remain by default.
- ⇒ Click **Apply** .

Figure 21.4 **Design options** dialog boxTo define the second design option:

- ⇒ To define the second design option, click **Create new design option** .
- ⇒ Then define data for the second design option:
 - make sure that **SNIP 2.03.01-84*** is selected for RC analysis;
 - in the **Analysis of sections by** list, select **DCF**;
 - other parameters remain by default.
- ⇒ Click **Apply** .

To define the third design option:

- ⇒ To define the third design option, click **Create new design option** .
- ⇒ Then define data for the third design option:
 - make sure that **SNIP 2.03.01-84*** is selected for RC analysis;
 - in the **Analysis of sections by** list, select **DCF**;
 - other parameters remain by default.
- ⇒ Click **Apply** .

To define the fourth design option:

- ⇒ To define the fourth design option, click **Create new design option** .
- ⇒ Then define data for the fourth design option:
 - select building code **SP 63.13330.2012** for RC analysis;
 - in the **Analysis of sections by** list, select **DCF**;

- other parameters remain by default.

⇒ Click **Apply**  .

To define the fifth design option:

⇒ To define the fifth design option, click **Create new design option**  .

⇒ Then define data for the fifth design option:

- make sure that **SNIP 2.03.01-84*** is selected for RC analysis;
- in the **Analysis of sections by** list, select **DCF**;
- other parameters remain by default.

⇒ Click **Apply**  .

⇒ To assign the first design option as a current one, in the **List of design options**, select appropriate row and click **Set as current type**.

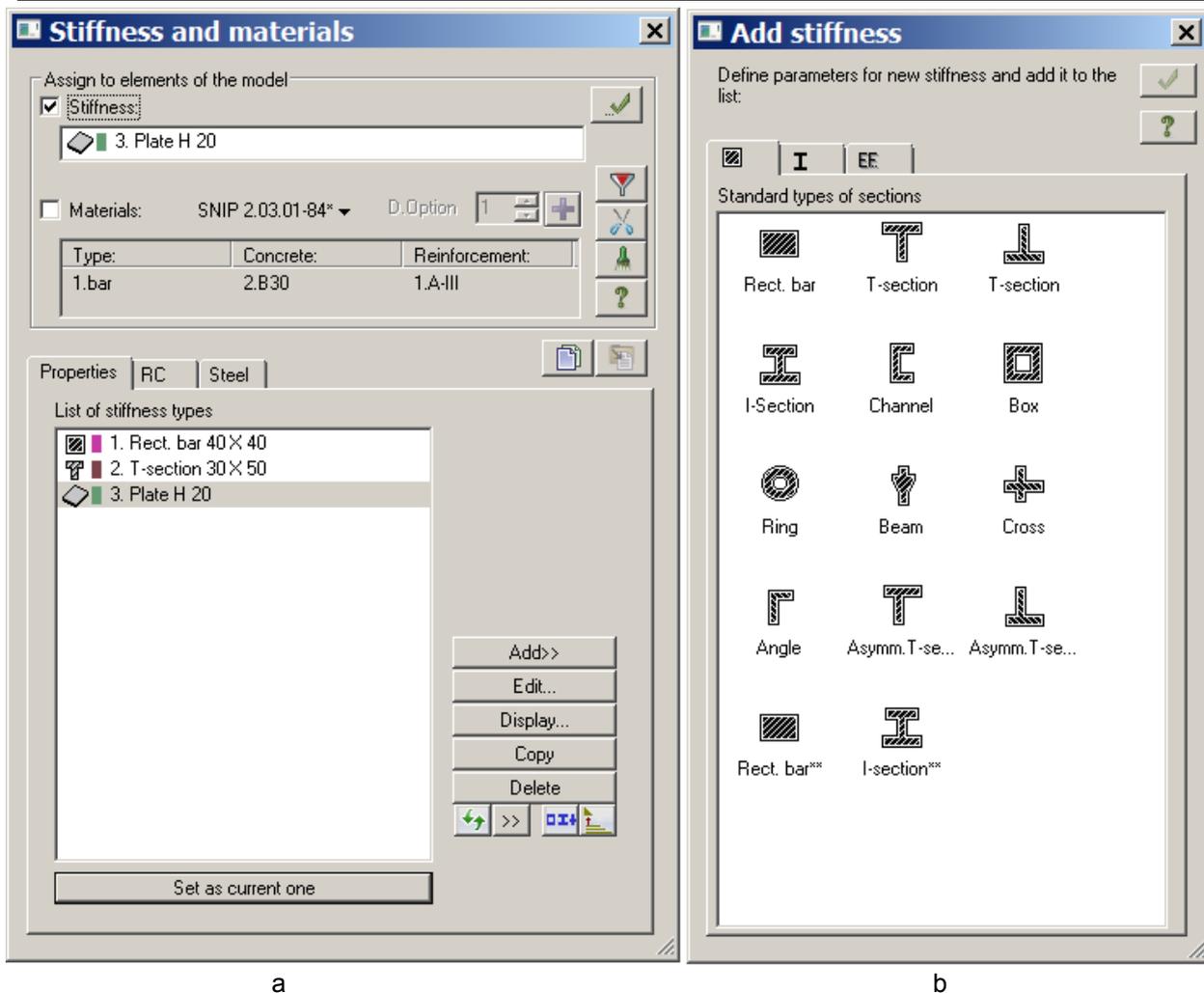
⇒ To close the **Design options** dialog box, click the **Close** button.

Step 4. Defining material properties to elements of the model

To create material data sets:

⇒ On the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click **Material properties** button  .

⇒ In the **Stiffness and materials** dialog box (see Fig.21.5a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.21.5b).



a

b

Figure 21.5 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.21.6):
 - modulus of elasticity – $E = 2.4e6 \text{ t/m}^2$ (for the U.S. keyboard layout);
 - geometric properties – $B = 40 \text{ cm}$; $H = 40 \text{ cm}$;
 - unit weight of material – $R_0 = 2.75 \text{ t/m}^3$.
- ⇒ To preview schematic presentation, click **Draw**.
- ⇒ To confirm the specified data, click **OK** .

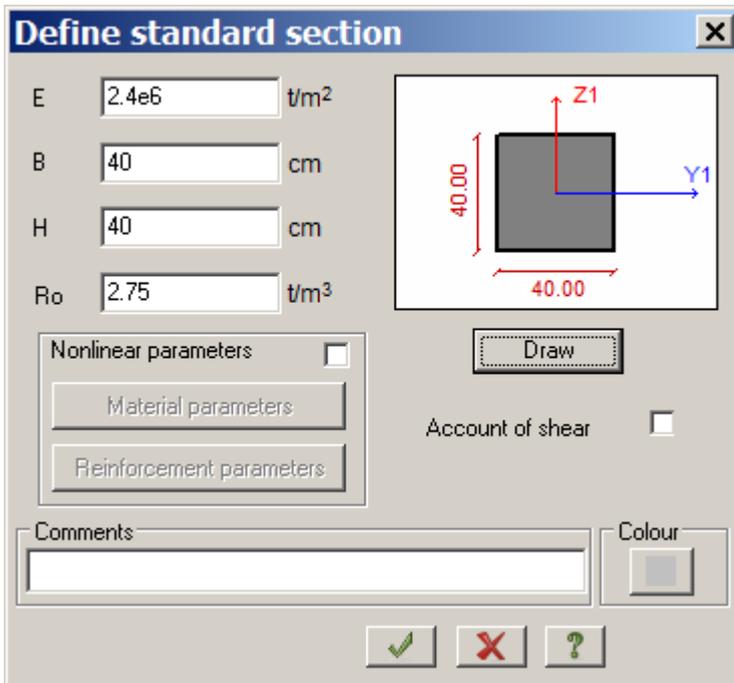


Figure 21.6 Define standard section dialog box

- ⇒ 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 – $E = 1.2e6 \text{ t/m}^2$;
 - geometric properties – $B = 30 \text{ cm}$; $H = 50 \text{ cm}$; $B1 = 50 \text{ cm}$; $H1 = 20 \text{ cm}$;
 - unit weight of material – $R_o = 2.75 \text{ t/m}^3$.
- ⇒ To confirm the specified data, click **OK** .
- ⇒ In the **Stiffness and materials** dialog box, select the third tab **Plates, solids, numerical** and double-click the **Plates** icon in the list.
- ⇒ In the **Stiffness for plates** dialog box (see Fig.21.7), specify the following parameters for **Plate** (floor slab):
 - modulus of elasticity – $E = 1.2e6 \text{ t/m}^2$ (for the U.S. keyboard layout);
 - Poisson's ratio – $\nu = 0.2$;
 - thickness – $H = 20 \text{ cm}$;
 - unit weight of material – $R_o = 2.75 \text{ t/m}^3$.
- ⇒ To confirm the specified data, click **OK** .

Figure 21.7 **Stiffness for plates** dialog box

⇒ To hide library of stiffness properties, in the **Stiffness and materials** dialog box click **Add** unfold button.

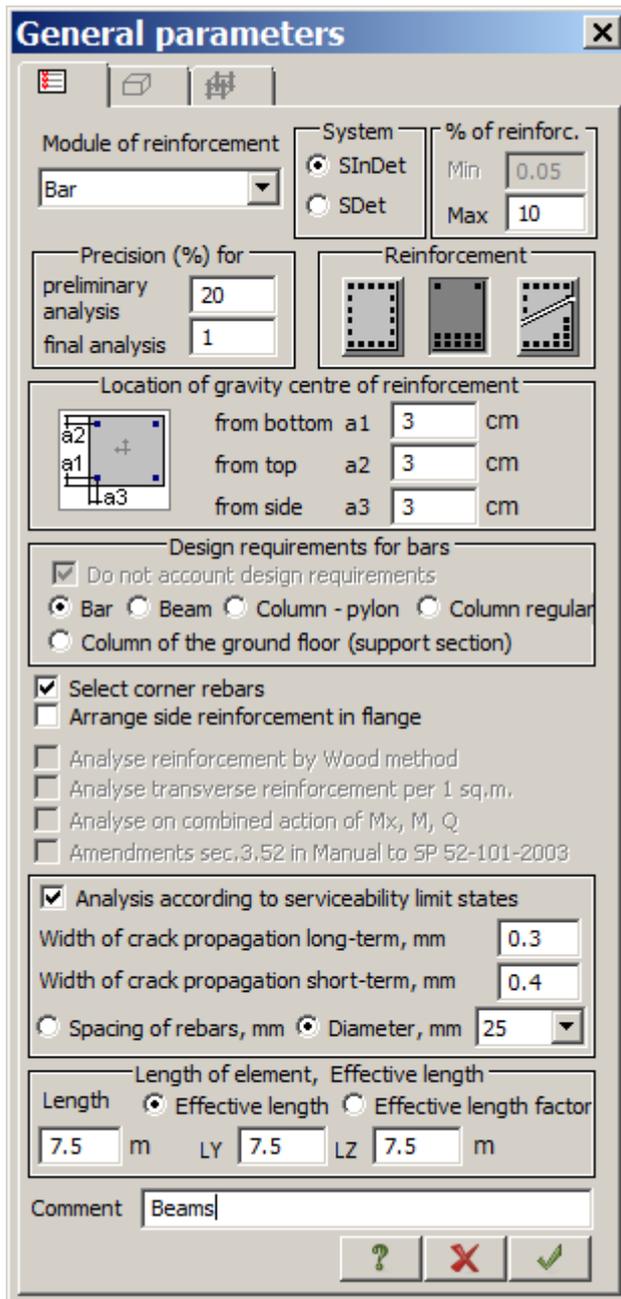
To define materials for the first design option for RC structures:

- ⇒ 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**.
- ⇒ In the **General parameters** dialog box (see Fig.21.8), 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 **Bar** option;
 - 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 $LY = 1$, $LZ = 1$;
 - in the **Comment** box, type comment - **Columns1**;
 - other parameters remain by default.
- ⇒ Click **OK** .

Figure 21.8 **General parameters** dialog box (for columns)

- ⇒ In the **Stiffness and materials** dialog box, click **Add** once again.
- ⇒ In another **General parameters** dialog box, define parameters for beams (will be applied for all design options by SNIP 2.03.01-84*):
 - in the **Reinforcement** area, select **Asymmetric** type of reinforcement;
 - in the **Design requirements for bars** area, select **Bar** option;
 - 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** option;
 - define parameters LY = 7.5 m, LZ = 7.5 m;
 - in the **Length** box, define the max allowed value for beams - 7.5 m;
 - in the **Comment** box, type comment - **Beams**;
 - other parameters remain by default.

⇒ Click **OK** .



General parameters

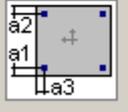
Module of reinforcement: Bar

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, Analyse transverse reinforcement per 1 sq.m., Analyse 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: Effective length Effective length factor, LY: 7.5 m, LZ: 7.5 m

Comment: Beams

? X 

Figure 21.9 **General parameters** dialog box (for beams)

- ⇒ In the **Stiffness and materials** dialog box, click **Add** once again.
- ⇒ In the **General parameters** dialog box (see Fig.21.10), define the following parameters for plate elements (will be applied for all design options by SNIP 2.03.01-84*):
- in the **Module of reinforcement** list, select **Shell**;
 - in the **Comment** box, type comment - **Plates**;
 - other parameters remain by default.
- ⇒ Click **OK** .

General parameters

Module of reinforcement: Shell

System: SInDet SDet

% of reinforc.: 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

Analyse transverse reinforcement per 1 sq.m.

Analyse on combined action of M_x , M_y , 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: 100

Length of element, Effective length:

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

Effective length: Effective length Effective length factor

Comment: Plates

Figure 21.10 General parameters dialog box (for plates)

- ⇒ 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 and click **Add**.
- ⇒ In the **Reinforcement** dialog box (see Fig.21.11), in the **Max diameter** list, select 25 mm and click **OK**



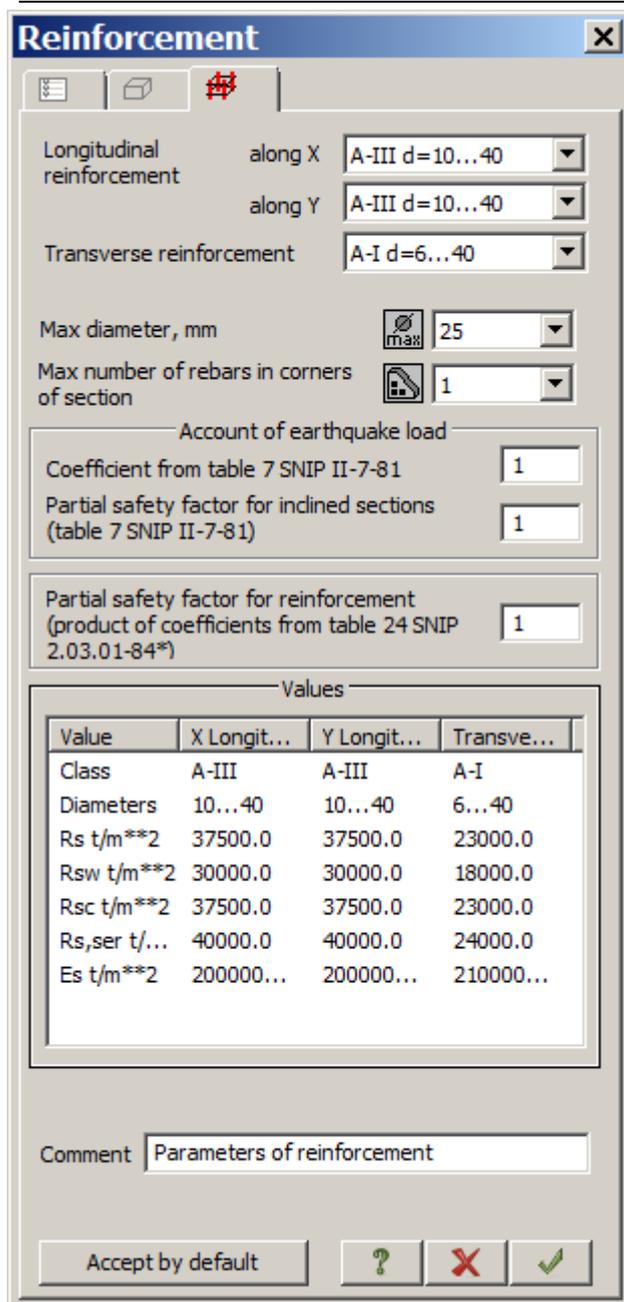


Figure 21.11 Reinforcement dialog box

To define material properties for the second design option for RC 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, define the following parameters for columns:
 - in the **Module of reinforcement** list, select **Bar**;
 - in the **Reinforcement** area, select **Asymmetric** type of reinforcement;
 - in the **Design requirements for bars** area, select **Bar** option;
 - 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 = 1, LZ = 1;

- in the **Comment** box, type comment - **Columns2**;
 - other parameters remain by default.
- ⇒ Click **OK** .

To define material properties for the third design option for RC structures:

- ⇒ To switch to the 3rd design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box, specify number 3.
 - ⇒ In the **Define parameters for reinforced concrete (RC) structures** area, select the **Type** option and then click **Add**.
 - ⇒ In the **General parameters** dialog box, 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 **Bar** 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 $LY = 1$, $LZ = 1$;
 - in the **Comment** box, type comment - **Columns3**;
 - other parameters remain by default.
- ⇒ Click **OK** .

To define material properties for the fourth design option for RC structures:

- ⇒ To switch to the 4th design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box, specify number 4.
 - ⇒ In the **Define parameters for reinforced concrete (RC) structures** area, select the **Type** option and then click **Add**.
 - ⇒ In the **General parameters** dialog box, 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 **Bar** option;
 - 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 $LY = 1$, $LZ = 1$;
 - in the **Comment** box, type comment - **Columns4**;
 - other parameters remain by default.
- ⇒ Click **OK** .

- ⇒ 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 **Bar** option;

- 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** option;
 - define parameters $LY = 7.5$ m, $LZ = 7.5$ m;
 - in the **Length** box, define the max allowed value for beams - 7.5 m;
 - in the **Comment** box, type comment - **Beams**;
 - other parameters remain by default.
- ⇒ Click **OK** .
- ⇒ In the **Stiffness and materials** dialog box, click **Add** once again.
- ⇒ In another **General parameters** dialog box, define parameters for plate elements:
- in the **Module of reinforcement** list, select **Shell**;
 - in the **Comment** box, type comment - **Plates**;
 - other parameters remain by default.
- ⇒ Click **OK** .
- ⇒ In the **Stiffness and materials** dialog box, select the **Concrete** option and click **Add**.
- ⇒ In the **Concrete** dialog box (see Fig.21.12), define the following parameters:
- define partial safety factors as equal to $\gamma_{b2} = 1$ and $\gamma_{b3} = 1$;
 - other parameters remain by default.
- ⇒ Click **OK** .

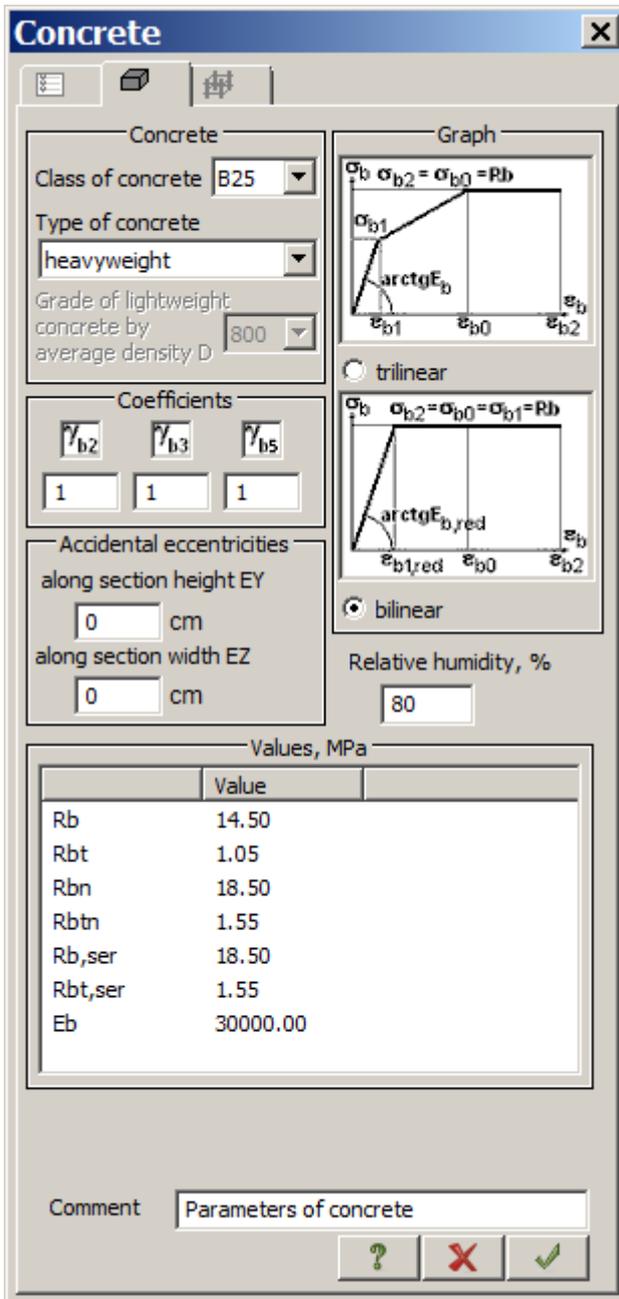


Figure 21.12 Concrete dialog box

- ⇒ In the **Stiffness and materials** dialog box, select the **Reinforcement** option and click **Add**.
- ⇒ In the **Reinforcement** dialog box (see Fig.21.13), in the **Max diameter** list, select 25 mm and click **OK** .

Reinforcement

Classes of reinforcement

Longitudinal reinforcement along X: A400

Longitudinal reinforcement along Y: A400

Transverse reinforcement: A400

Max diameter, mm: 25

Max number of rebar in corners of section (for bars): 1

Account of earthquake load

Coefficient from table 7 SNIP II-7-2010: 1

Partial safety factor for inclined sections (table 7 SNIP II-7-2010): 1

Values, MPa

Value	X Longit...	Y Longit...	Transve...
Class	A400	A400	A400
Diameters	6-40	6-40	6-40
Rsn	400.0	400.0	400.0
Rs_ser	400.0	400.0	400.0
Rs	350.0	350.0	350.0
Rsw	280.0	280.0	280.0
Rsc	350.0	350.0	350.0

Update

? X ✓

Figure 21.13 Reinforcement dialog box

To define material properties for the fifth design option for RC structures:

- ⇒ To switch to the 5th design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box, specify number 5.
- ⇒ In the **Define parameters for reinforced concrete (RC) structures** area, select the **Concrete** option and then click **Add**.
- ⇒ In the **Concrete** dialog box, define the following parameters:
 - in the **Class of concrete** box, select **B30**;
 - other parameters remain by default.
- ⇒ Click **OK** .

To assign material properties to elements of the model for the first design option:

- ⇒ In the **Stiffness and materials** dialog box (see Fig.21.14), in the **Define parameters for reinforced concrete (RC) structures** area, select the **Concrete** option.

- ⇒ In the list of concrete types, select the row '**1.B25**'.
- ⇒ Click **Set as current type**. Selected type of concrete parameters will be displayed in the **Materials** box under **Assign to elements of the model**. You can also specify the current type of concrete parameters by double-clicking the necessary type in the list.
- ⇒ In the **Stiffness and materials** dialog box, select the **Type** option. In the list of material properties types for RC structures, select the row '**3.shell Plates**'.
- ⇒ Click **Set as current type**.
- ⇒ To switch to the first design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box define number 1. (In this case the stiffness '**3.Plate H20**' should be defined as current type of stiffness).

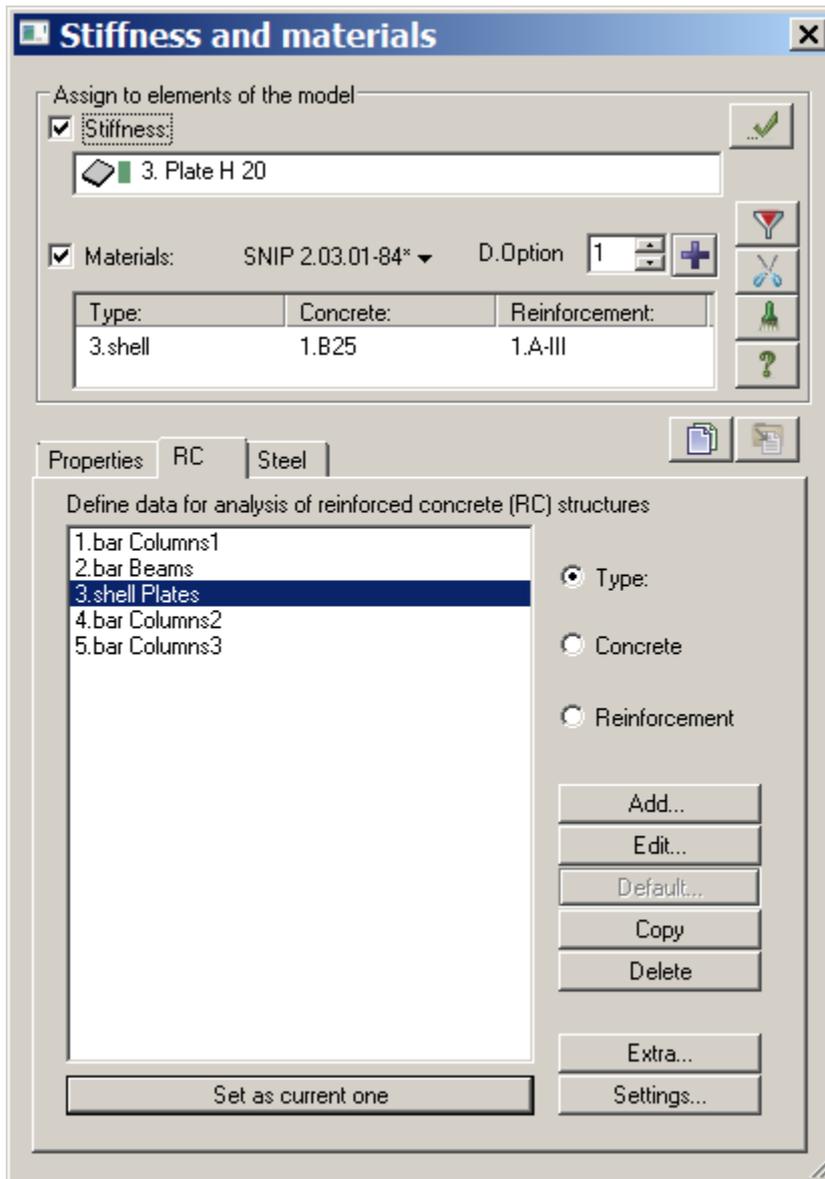


Figure 21.14 **Stiffness and materials** dialog box

- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), point to **Select elements** and 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** .
- ⇒ The **Warning** box is displayed (see Fig.21.15). Click **OK**.

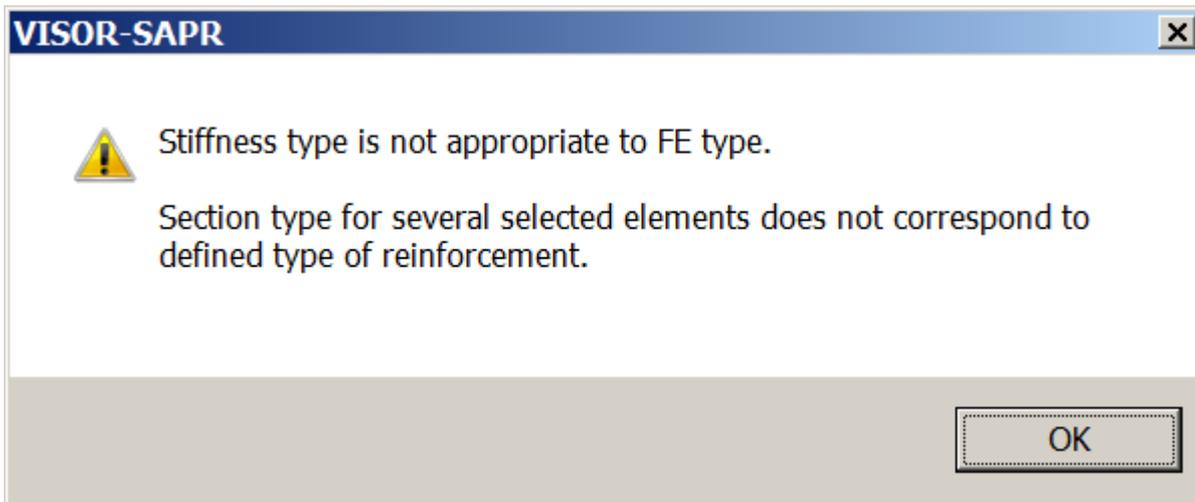


Figure 21.15 **Warning** dialog box

- ⇒ In the **Stiffness and materials** dialog box, on the **RC** tab, in the list of material parameters for RC structures, click the stiffness type '**2. bar beams**'.
- ⇒ Click **Set as current type**.
- ⇒ In the **Stiffness and materials** dialog box, on the **Properties** tab, in the **List of stiffness types**, click the stiffness type '**2.T-section 30x50**'.
- ⇒ Click **Set as current type**.
- ⇒ 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 40x40**'.
- ⇒ 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. bar Columns1**'.
- ⇒ 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 **Warning** box, click **OK**.

To assign material properties to elements of the model for the second design option:

- ⇒ 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.
- ⇒ To define materials to the second 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.shell Plates**'.
- ⇒ Click **Set as current one**.
- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), point to **Select elements** and click **Select elements** .

- ⇒ Select all elements of the model with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the **Warning** box, click **OK**.
- ⇒ In the list of materials for RC elements, select the row '**2.bar Beams**' and click **Set as current one**.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the list of materials for RC elements, select the row '**4.bar Columns2**' and click **Set as current one**.
- ⇒ 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** .

To assign material properties to elements of the model for the third design option:

- ⇒ To switch to the 3rd design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box, specify number 3.
- ⇒ In the list of materials for RC elements, select the row '**3.shell Plates**' and click **Set as current one**.
- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), point to **Select elements** and click **Select elements** .
- ⇒ Select all elements of the model with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the **Warning** box, click **OK**.
- ⇒ In the list of materials for RC elements, select the row '**2.bar Beams**' and click **Set as current one**.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the list of materials for RC elements, select the row '**5.bar Columns3**' and click **Set as current one**.
- ⇒ 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** .

To assign material properties to elements of the model for the fourth design option:

- ⇒ To switch to the 4th design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box, specify number 4. (In this case, make sure that in the list of current materials the following data should be defined as current one: type – **3.shell**, concrete class – **1.B25** and class of reinforcement – **1.A400**.)
- ⇒ 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 **Warning** box, click **OK**.
- ⇒ In the list of materials for RC elements, select the row '**2.bar Beams**' and click **Set as current one**.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the list of materials for RC elements, select the row '**1.bar Columns1**' and click **Set as current one**.

- ⇒ 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** .

To assign material properties to elements of the model for the fifth design option:

- ⇒ To switch to the 5th design option, in the **Stiffness and materials** dialog box, in the **No. of current design option** box, specify number 5.
- ⇒ In the list of materials for RC elements, select the row '**3.shell Plates**' and click **Set as current one**.
- ⇒ On the **Select** toolbar, point to **Select elements** and click **Select elements** .
- ⇒ Select all elements of the model with the pointer.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the **Warning** box, click **OK**.
- ⇒ In the list of materials for RC elements, select the row '**2.bar Beams**' and click **Set as current one**.
- ⇒ In the **Stiffness and materials** dialog box, in the **Define parameters for reinforced concrete (RC) structures** area, select the **Concrete** option.
- ⇒ In the list of concrete types, select the row '**1.B30**' and click **Set as current type**.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the **Stiffness and materials** dialog box, select the **Type** option. In the list of material properties types for RC structures, select the row '**1.bar Columns1**'.
- ⇒ 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** .

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.21.16), click **All elements** and specify **Load factor** as equal to **1**. Then click **Apply**  (dead weight is automatically applied to elements).

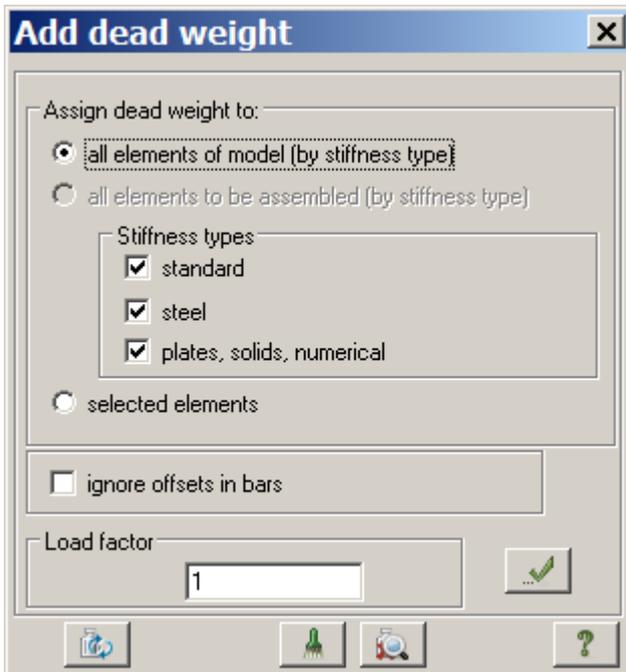


Figure 21.16 **Add dead weight** dialog box

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 (by default, it is displayed at the bottom of the screen), point to **Select elements** and click **Select elements** .
- ⇒ 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 **Define loads** dialog box (see Fig.21.17), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

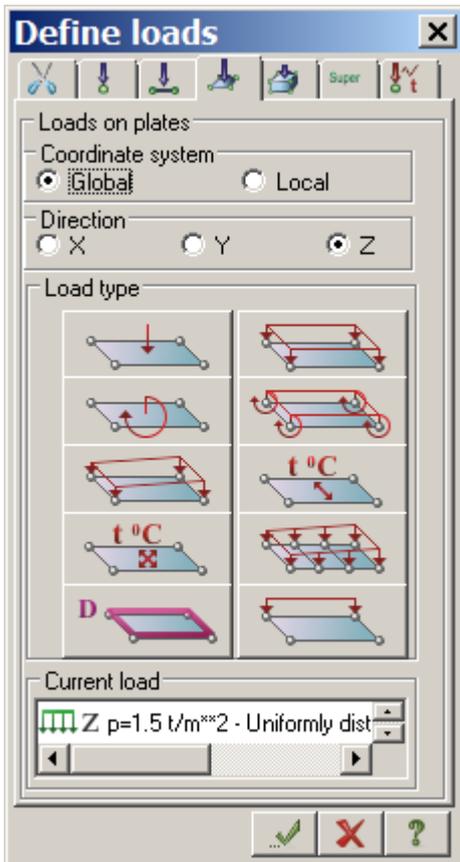
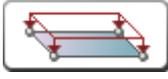


Figure 21.17 Add dead weight dialog box

- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 1.5 \text{ t/m}^2$ (see Fig.21.18).
- ⇒ Click **OK** .

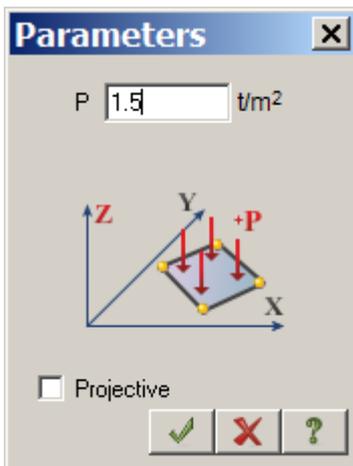


Figure 21.18 Load parameters dialog box

- ⇒ The **Warning** message box is displayed (see Fig.21.19). Click **OK**.

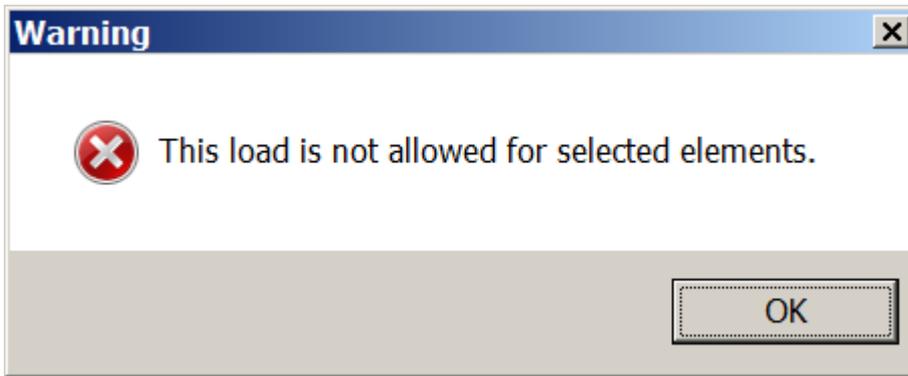


Figure 21.19 Warning message box



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 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.
- ⇒ Select all elements of the model with the pointer once again.
- ⇒ In the **Define loads** dialog box, in the **Load type** area, click the **Uniformly distributed load** button.
- ⇒ In the **Load parameters** dialog box specify $P = 0.35 \text{ t/m}^2$.
- ⇒ Click **OK**.

To create load case No.4:

- ⇒ To change the number of the current load case, click the **Next load case** button  located on the Status bar or on the toolbar.
- ⇒ On the **Projection** toolbar (by default, it is displayed at the bottom of the screen), click **Projection on XOZ-plane** .
- ⇒ On the **Select** toolbar, click **Select vertical bars** button . Then select extreme left row of columns.
- ⇒ In the **Define loads** dialog box, on the **Load on bars** tab, specify direction along the **X-axis**.
- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = -0.5 \text{ t/m}$.
- ⇒ Click **OK** .

To create load case No.5:

- ⇒ 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 present design model in projection on the YOZ-plane, on the **Projection** toolbar, click **Projection on YOZ-plane** .
- ⇒ Select columns located in front of the structure (extreme left row on projection).
- ⇒ In the **Define loads** dialog box, on the **Load on bars** tab, specify direction along the **Y-axis**.
- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = -0.5 \text{ t/m}$.

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

To define detailed information about load cases:

- ⇒ On the **Create and edit** ribbon tab, select the **Loads** panel and click **Edit load cases** . The **Edit load cases** dialog box is displayed on the screen (see Fig.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** .
- ⇒ For load case 3 – in the **Edit selected load case** area, in the **Type** box, select **Short-term** and click **Apply** .
- ⇒ For load case 4 – in the **Edit selected load case** area, in the **Type** box, select **Instant** and click **Apply** .
- ⇒ For load case 5 – in the **Edit selected load case** area, in the **Type** box, select **Instant** and click **Apply** .

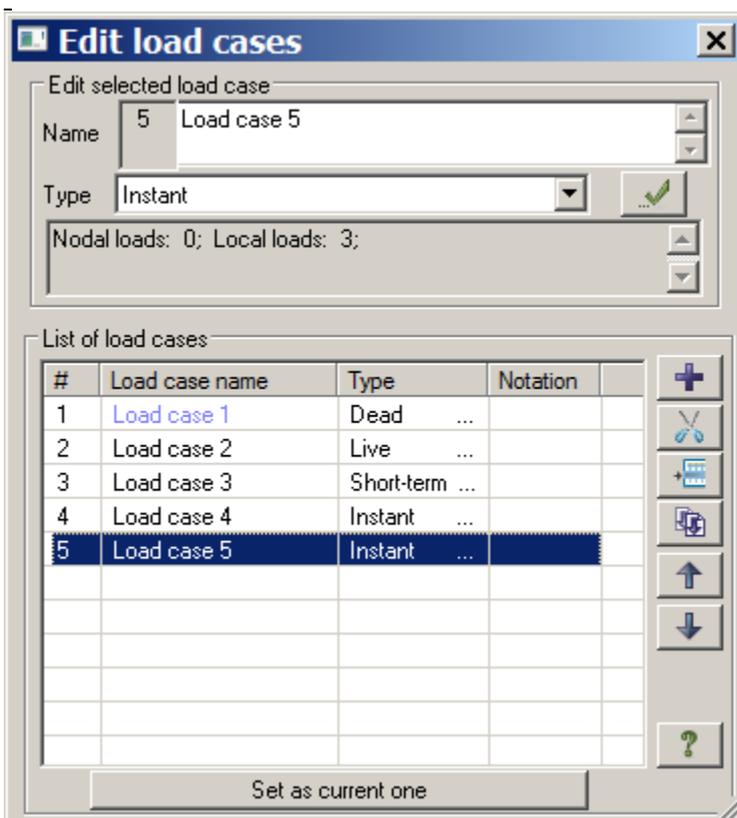


Figure 21.20 **Edit load cases** dialog box

Step 6. 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.21.21), the DCF table is generated automatically with parameters accepted by default for every load case. Now you have to modify parameters for the third, the fourth and the fifth load cases.

- ⇒ In the **Design combinations of forces** dialog box (see Fig.21.21), select building code **SNIP 2.01.07-85*** and specify the following data:
 - for Load case 3 – in the **Summary table for DCF calculation**, define **Load factor** as equal to 1.4 and then click **Apply** .
 - for Load case 4 – select the **Account of sign variability** box, then in the **No. of group of mutually exclusive load cases** box specify 1 and click **Apply** .
 - for Load case 5 – select the **Account of sign variability** box, then in the **No. of group of mutually exclusive load cases** box specify 1 and click **Apply** .
- ⇒ Click **OK** .

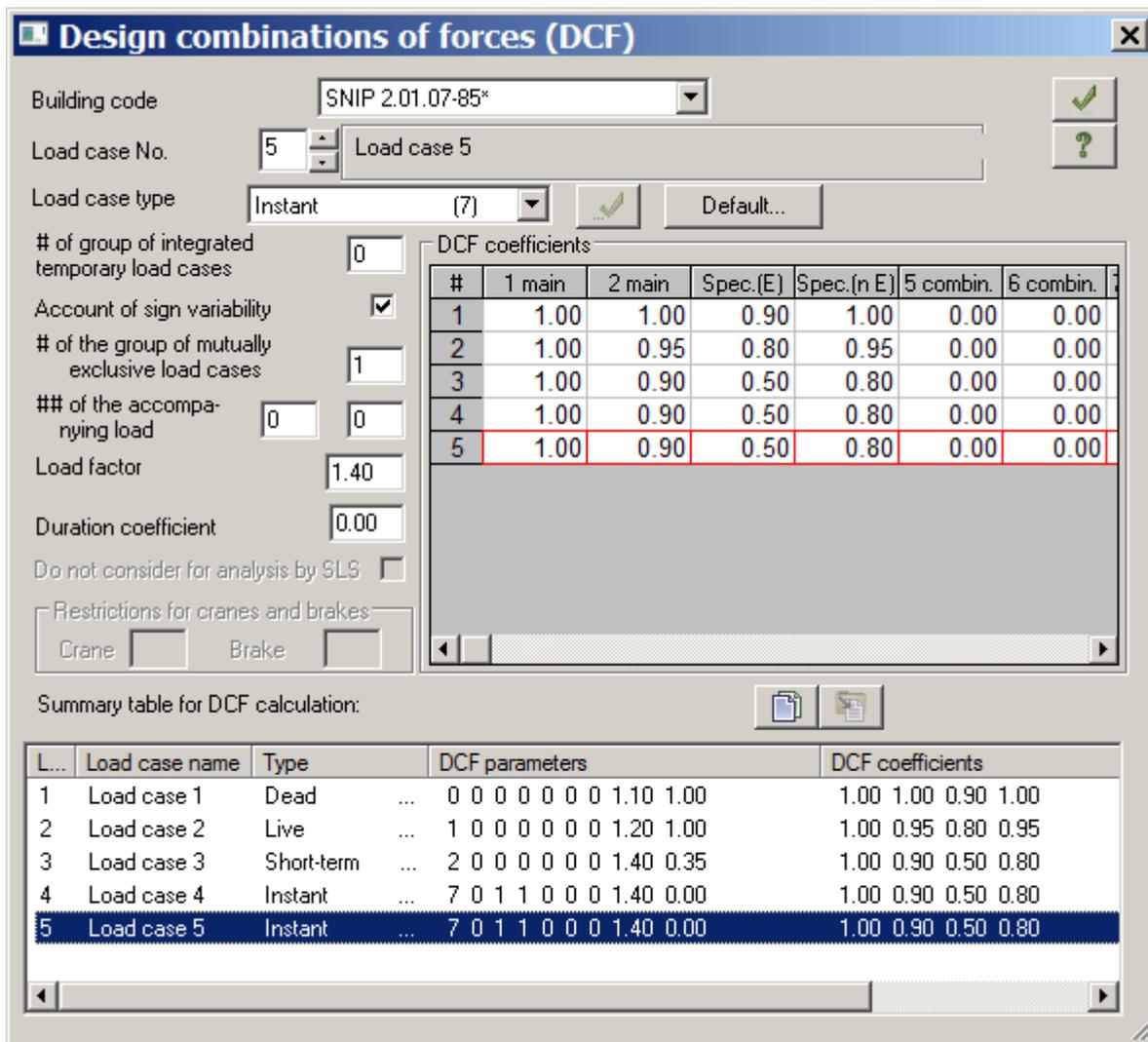


Figure 21.21 Design combinations of forces dialog box

Step 7. 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 problem** drop-down list, click **Complete analysis** .

Step 8. 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.

To hide presentation of loads on design model:

- ⇒ On the **Select** toolbar, click **Flags of drawing** button . In the **Display** dialog box, clear the **Loads** check box on the **General** tab.
- ⇒ Click **Redraw** .
- ⇒ In the mode of analysis results visualization, by default design model is presented with account of nodal displacements.

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 diagrams of internal forces:

- ⇒ To display diagram M_y , on the **Results** tab, select **Forces in bars** panel and click **Moment diagrams (M_y)** button .
- ⇒ To display diagram Q_z , on the **Results** tab, select **Forces in bars** panel and click **Shear force diagrams (Q_z)** button .
- ⇒ To display diagram **N**, on the **Results** tab, select **Forces in bars** panel and click **Axial force diagrams (**N**)** button .
- ⇒ To display mosaic plots **N**, on the **Results** tab, select **Forces in bars** panel and click **Mosaic plot of forces in bars** command in the **Force diagrams/Mosaic plots** drop-down list.

To change number of active load case:

- ⇒ On the status bar (displayed at the bottom of the screen), in the **Load case No.** list, select No. **2** and click **Apply** .

To 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 N_x , click **Stress N_x** button on the same panel.

To generate and review tables of analysis results:

- ⇒ To present table with 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.21.22), select **DCF, 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 21.22 **Standard tables** dialog box

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

Step 9. Review and evaluate results from analysis of reinforcement



*When analysis procedure is complete, to review and evaluate results from analysis of 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 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.
- ⇒ 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 of analysis results** dialog box (see Fig.21.23), 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).

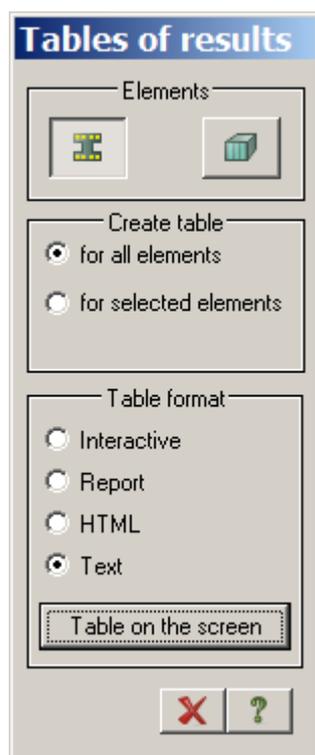


Figure 21.23 **Tables of analysis results** dialog box

To change the No. of design option:

- ⇒ On the Status bar, in the **Design option No.** box, select the row corresponding to the second design option.



To review and evaluate analysis results for another design option, do one of the following:

- either use the **Design options**  command. Then in the **Design options** dialog box, select

appropriate row in the **List of design options** and click **Set as current type**;
 - or on the **Status bar**, in the **Design option No.** box, select the row corresponding to appropriate design option. The design option with this number will become current one.



Analysis results for the second design option are evaluated in the same way as for the first design option.

To modify section dimensions:



After review of analysis results for reinforcement by all design options, we accept the fourth design option as the final one. For this design option, we will carry out analysis of reinforcement with increased section dimensions for bars.

- ⇒ On the **Design** ribbon tab, on the **Design** panel, click **Material properties** button .
- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select **1. Rect.bar 40x40**.
- ⇒ Click **Edit**.
- ⇒ In the **Define standard section** dialog box specify the following parameters for **Rectangular bar**:
 - geometric properties – B = 50 cm; H = 50 cm.
- ⇒ To confirm the specified data, click **OK** .
- ⇒ In the **Stiffness and materials** dialog box, in the **List of stiffness types**, select **2. T-section 30x50**.
- ⇒ Click **Edit**.
- ⇒ In the **Define standard section** dialog box specify the following parameters for **T-section (table at the top)**:
 - geometric properties – H = 60 cm.
- ⇒ To confirm the specified data, click **OK** .

To carry out analysis of reinforcement for modified section dimensions:

- ⇒ Switch to the fourth design option.
- ⇒ To carry out analysis of reinforcement, on the **Design** ribbon tab, on the **RC: Analysis** panel, click **Analyse reinforcement** button .
- ⇒ In the **RC analysis** dialog box (see Fig.21.24), click **Analyse**.
- ⇒ When analysis is complete, close the dialog box.

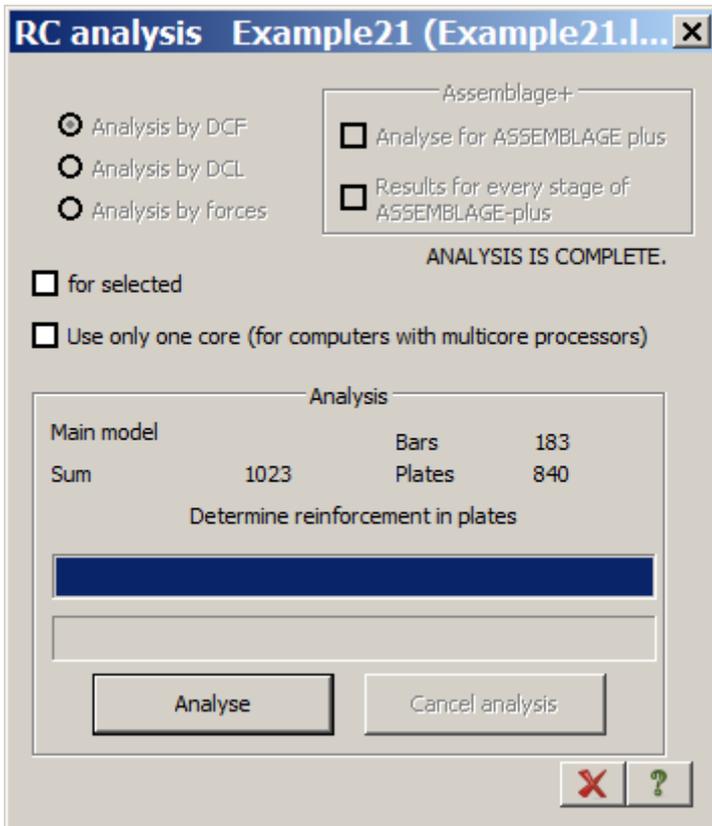


Figure 21.24 RC analysis dialog box



Results for analysis of reinforcement with modified section dimensions are evaluated in a similar way as described above.