Example 19. Analysis of two-span beam using NonLinear Engineering Design system

In this lesson you will learn how to:
– to carry out analysis with NonLinear Engineering Design (NL Engineering) system.

Description:
Model of the beam and its boundary conditions are presented in Figure 19.1.
Sections for elements of the beam are presented in Figure 19.2.
Material for beam – reinforced concrete B25.

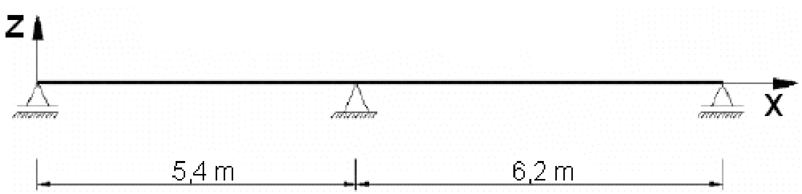


Figure 19.1 Model of the beam

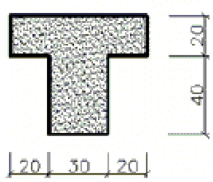


Figure 19.2 Sections for elements of the beam

Loads:

load case 1 – dead weight (see Figure 19.3);

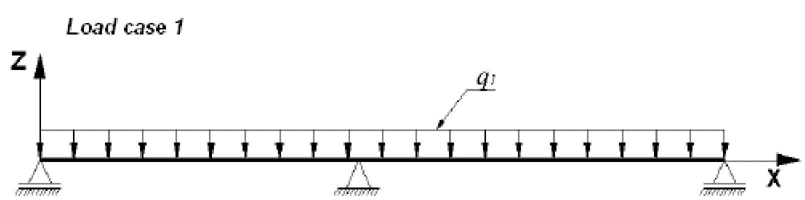


Figure 19.3 Load case No.1 for beam

load case 2 – uniformly distributed load q_2 = 0.3 t/m (see Figure 19.4);

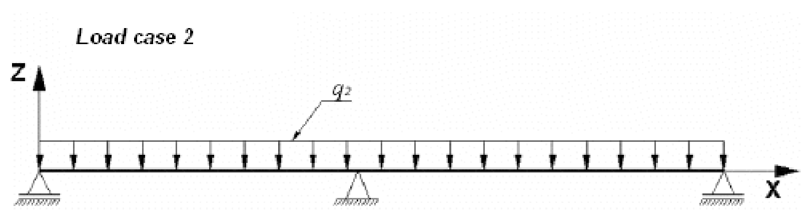


Figure 19.4 Load case No.2 for beam

load case 3 – uniformly distributed load in the first span q_3 = 0.87 t/m (see Figure 19.5);

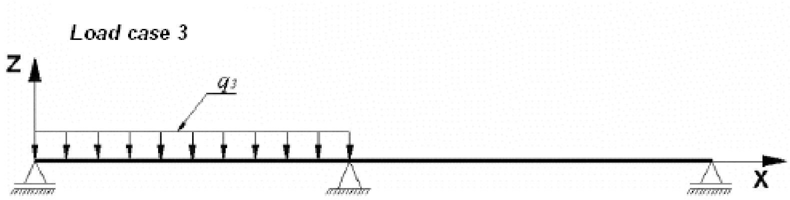


Figure 19.5 Load case No.3 for beam

load case 4 – uniformly distributed load in the second span q_4 = 0.87 t/m (see Figure 19.6);

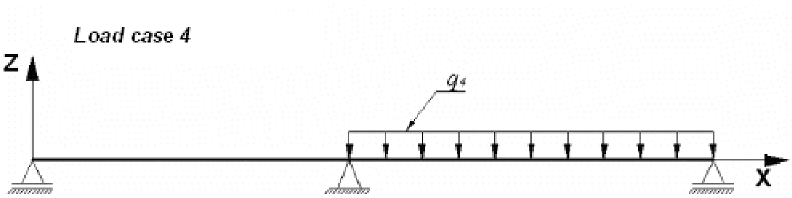


Figure 19.6 Load case No.4 for beam

- On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2020** and then click **LIRA-SAPR 2020**.

Step 1. Creating new problem

- On the FILE menu, click **New** (button on the toolbar).
- In the **Model type** dialog box (see Figure 19.7) specify the following data:
  - problem name – **Example19_1**;
  - problem code (by default it coincides with the problem name);
  - model type – **5 – Six degrees of freedom per node** (translations X, Y, Z and rotations Ux, Uy, Uz).
- Click **OK** .

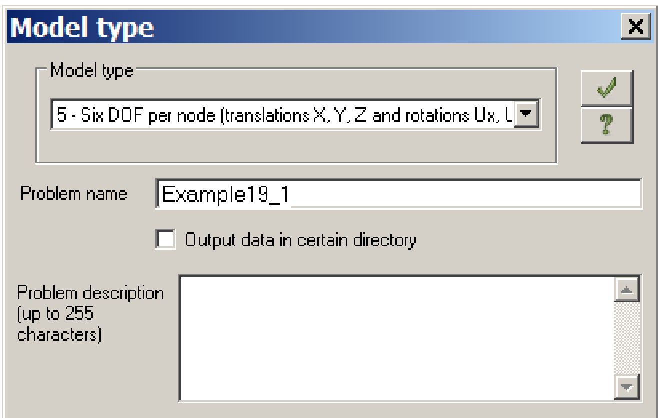


Figure 19.7 **Model type** dialog box

*It is also possible to open the **Model type** dialog box with a pre-defined type of model. To do this, on the **LIRA-SAPR menu** (Application menu), point to **New** and click **Model type 5 (Six DOF per node)** command . One more way to do the same: on the Quick Access Toolbar, click **New** and in the drop-down menu select **Model type 5 (Six DOF per node)** command . Then you should define only problem name.*

To save all output data files for the problem in certain directory, select appropriate check box. The directory name will coincide with the name of the problem. This directory will appear in the directory for files with analysis results. This is helpful if you have to find output data files for certain problem, then transfer these files or review and evaluate them with the help of Windows Explorer or other file managers.

Step 2. Generating model geometry

- On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** drop-down list and click the **Create frame** command.
- To divide the beam spans into 4 parts, in the **Create plane fragments and grids** dialog box specify the following data:
  - spacing along the first axis:

L(i)	**N**
1.35	4
1.55	4.

  - other parameters remain by default (see Figure 19.8).

- Click **Apply** .

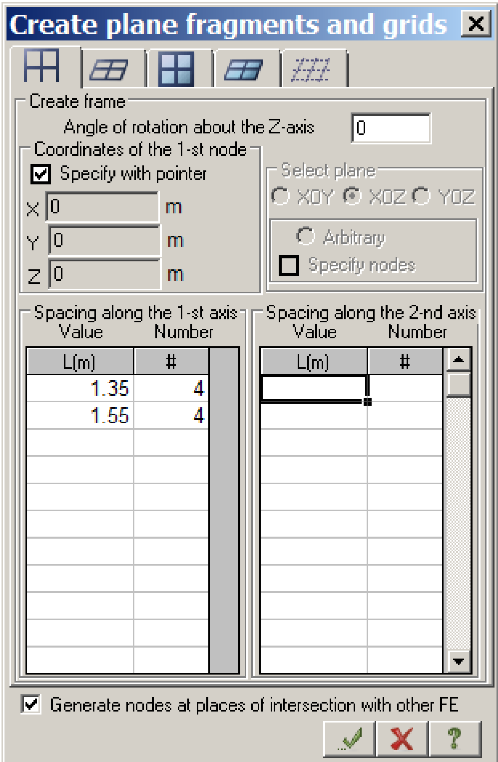


Figure 19.8 **Create plane fragments and grids** dialog box

<u>To save data about design model:</u>

- On the **LIRA-SAPR menu** (Application menu), click **Save** command .
- In the **Save as** dialog box specify the following data:
  - file name – **Example19_1**;
  - location where you want to save this file (**Data** folder is displayed by default).
- Click **Save**.

Step 3. Defining boundary conditions

<u>To present numbers of nodes and elements on the screen:</u>

- To switch to projection on the XOZ-plane, 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 **Flags of drawing** button .
- In the **Display** dialog box, select the **Element numbers** check box on the **Elements** tab.
- On the **Nodes** tab, select the **Node numbers** check box.
- Click **Redraw** .

<u>To select nodes No.1 and 9:</u>

- On the **Select** toolbar, point to **Select nodes** drop-down list and click **Select nodes** button .
- Select nodes No.1 and 9 with the pointer (the nodes will be coloured red).

To define boundary conditions for nodes No.1 and 9:

- 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.19.9) specify directions along which displacements of nodes are not allowed (Z). To do this, select appropriate check boxes.
- Click the **Add restraints at selected nodes** button (the nodes will be coloured blue).

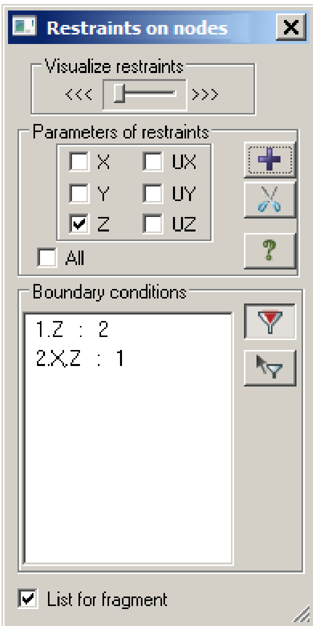


Figure 19.9 **Restraints on nodes** dialog box

To define boundary conditions for node No.5:

- Select node No.5 with the pointer.
- In the **Restraints on nodes** dialog box specify directions along which displacements of nodes are not allowed (X, Z). To do this, select appropriate check boxes.
- Click the **Add restraints at selected nodes** button .
- Close the 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.

The model with numbers of nodes and elements is presented in Fig.19.10.

Figure 19.10 Design model with numbers of nodes and elements

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.19.11), define parameters for the first design option:
  - in the **Analysis of sections by** list, select **DCL**;

- to select the DCL table, click **Add/Edit DCL table** button ;
- in the **Design combinations of loads** dialog box, in the drop-down list, select the building code SP 20.13330.2011/2016;
- click the **Save** command ;
- close the **Design combinations of loads** dialog box;
- in the **Design options** dialog box, select SP 63.13330.2012/2018 for RC analysis;
- other parameters remain by default.

➯ Click **Apply** .

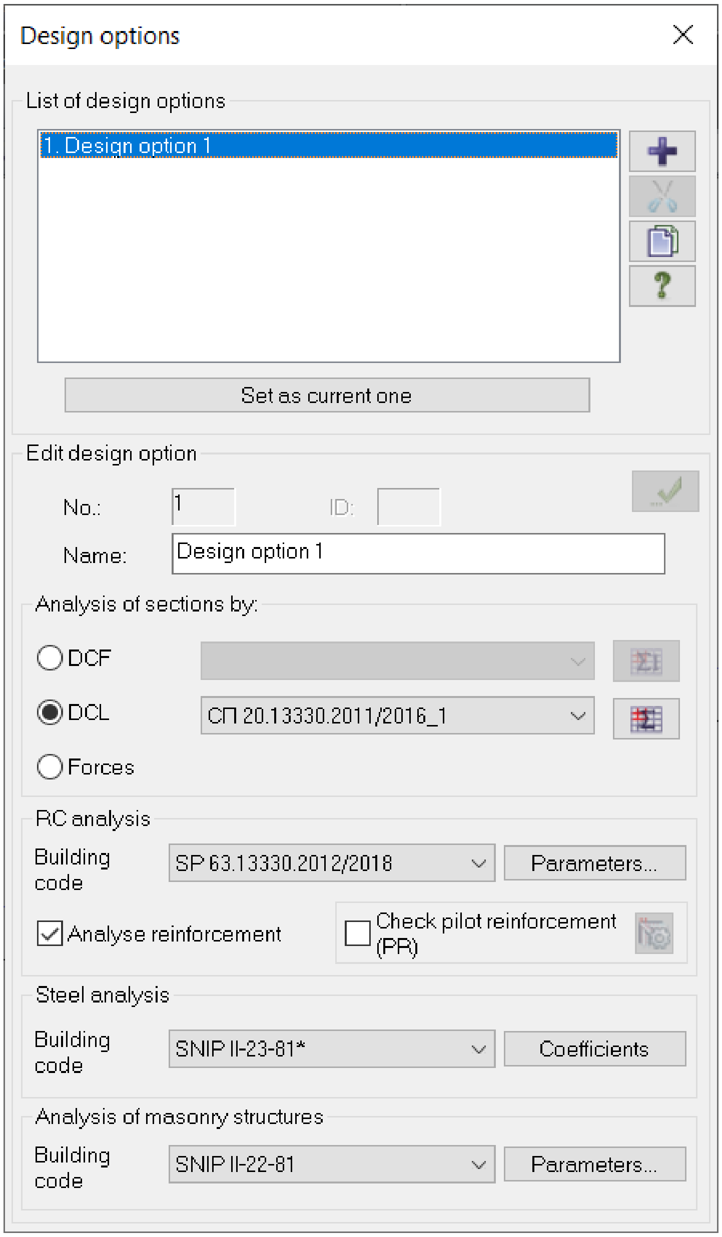


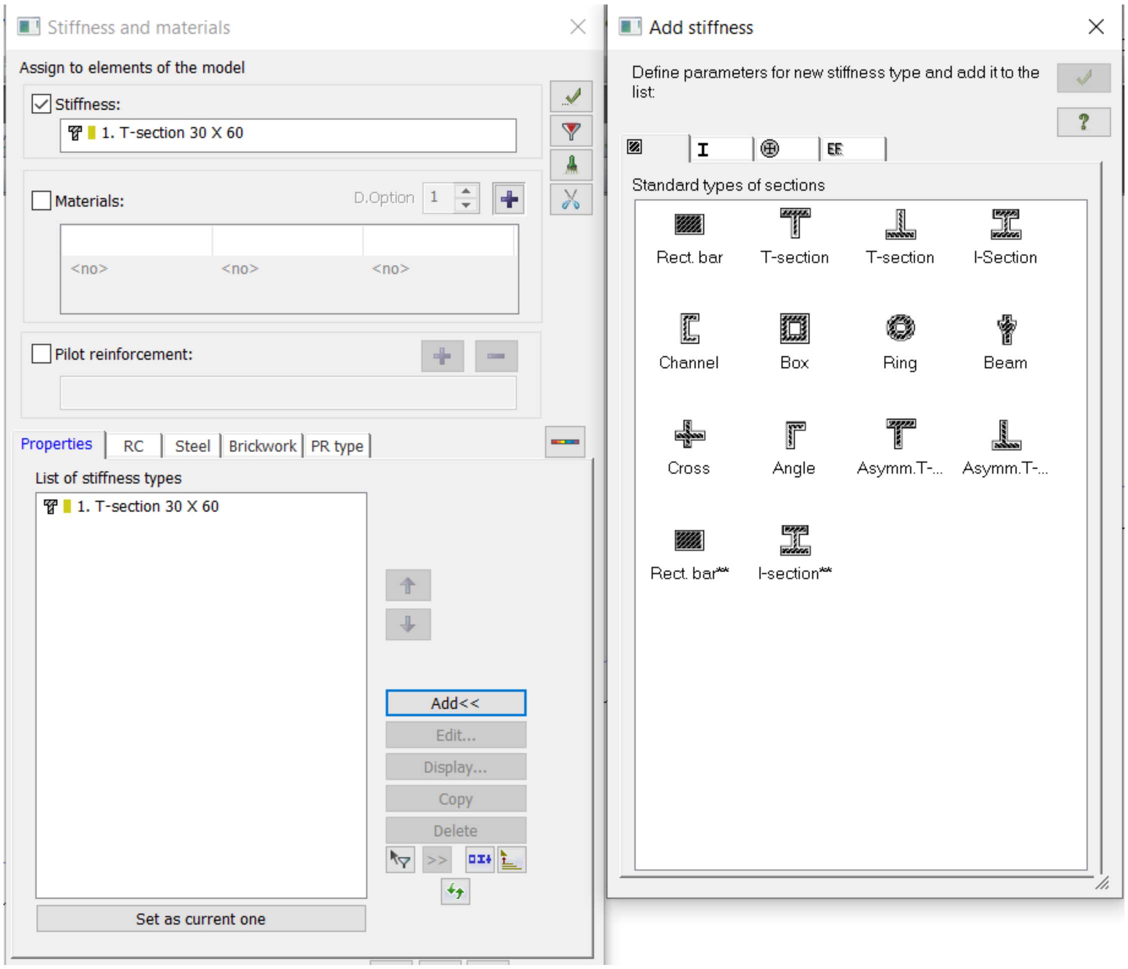
Figure 19.11 **Design options** dialog box

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

Step 5. Defining material properties to elements of the beam

<u>To create material data sets:</u>

- 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.19.12a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.19.12b).



a b

Figure 19.12 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- Double-click the **T-section (table at the top)** icon in the list.
- Then specify the following parameters for **T-section (table at the top)** (see Fig.19.13):
  - modulus of elasticity – E = 3e6 t/m^2 (for the U.S. keyboard layout);
  - Poisson's ratio – v = 0.25;
  - geometric properties – B = 30 cm; H = 60 cm; B1 = 70 cm; H1 = 20 cm;
  - unit weight of material – Ro = 2.5 t/m^3.
- To preview schematic presentation, click **Draw**.
- To confirm the specified data, click **OK** .

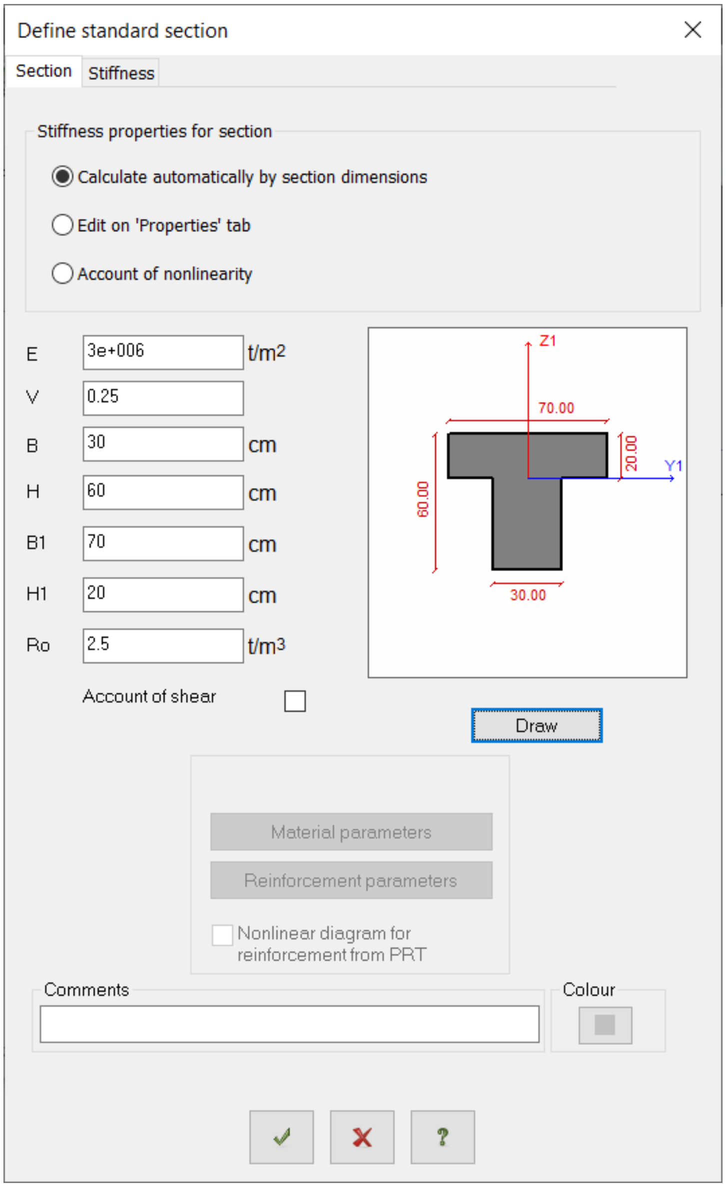


Figure 19.13 **Define standard section** dialog box

- To hide library of stiffness properties, in the **Stiffness of elements** 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**.
- In the **Material properties for analysis of RC structures** dialog box (see Fig.19.14), select the **Type: Bar** in the first line and then in the right part of the dialog box define the following parameters 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.

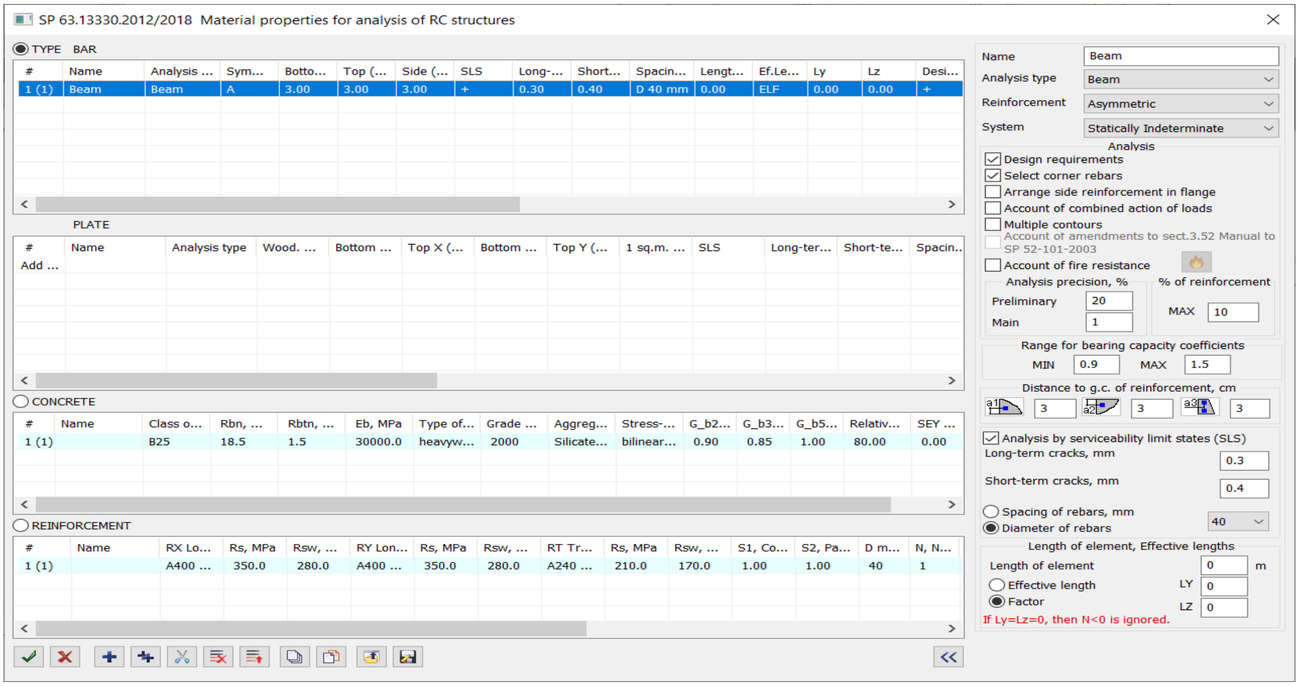


Figure 19.14 **Material properties for analysis of RC structures** dialog box

- 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 A240;
  - 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 '**1.T-section (table at the top)**' is defined as current one and type of materials '**1.Beam**', class of concrete **1.B25** and class of concrete **1.A400.A400.A240** are also defined as current ones.

- On the **Select** toolbar, click **Select horizontal bars** button .
- Select all horizontal elements of the model. 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.19.15), click **OK**. (This warning is displayed because type of reinforcement 'Beam' was applied to selected bars. Number of design sections in these bars is equal to 2, but for design of flexural element, forces should be computed in three or more sections.)

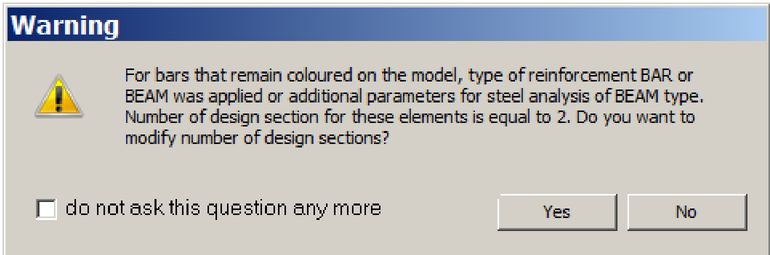


Figure 19.15 **Warning** box

Step 6. Defining design sections for beams

- In the **Design sections** dialog box (see Fig.19.16), specify number of sections N = 5.
- Click **Apply** .
- Close the dialog box.

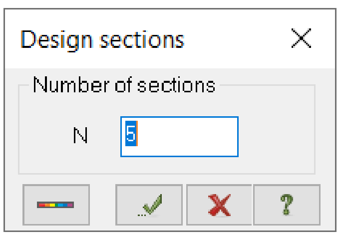


Figure 19.16 **Design sections** dialog box

Step 7. Applying loads

How to assign load case

It is allowed to define up to 300 load cases. Certain number and arbitrary name are assigned to every load case. Load case may include any number of loads. To assign number and name to a load case, on the ***Create and edit*** *ribbon tab, select the* ***Loads*** *panel and click* ***Edit load cases*** *. In the* ***Edit load cases*** *dialog box (see Fig.19.17) specify appropriate data. When you start a program, name* ***Load case 1*** *is displayed by default.*

How to define loads

To define loads on nodes and elements, on the ***Create and edit*** *ribbon tab, select the* ***Loads*** *panel and click appropriate command from the* ***Loads on nodes and elements*** *drop-down list. In the* ***Define loads*** *dialog box (see Fig.19.19) specify appropriate data.*
Define loads *dialog box contains tabs where you can specify loads on nodes, bars, plates, solids and super-elements, and loads for time history analysis. It is accepted by default that* **loads belong to the same active load case***; the number of this load case was specified beforehand. There is also a tabbed page where it is possible to edit or delete loads of the active load case.*
In the dialog box you can also specify coordinate system (global or local), direction of load X, Y, Z, W (W-warping), type of static or dynamic load (static load is coloured brown, initial displacement – yellow, dynamic load – pink, load for special FE turnbuckle – green). Menu of these buttons is changed automatically and depends on the type of FE. When you click these buttons, you will see another dialog box where it is necessary to specify parameters of load. Applied loads and effects will appear in the **Current load** *field of the dialog box.*

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.19.17).
- For load case 1 – in the **Edit selected load case** area, in the **Type** box, select **Dead** and click **Apply** .
- To add the second load case, in the **List of load cases** area, click **Add load case (to the end)** .
- For load case 2 – in the **Edit selected load case** area, in the **Type** box, select **Live** and click **Apply** .
- To add the third load case, in the **List of load cases** area, click **Add load case (to the end)** .
- For load case 3 – in the **Edit selected load case** area, in the **Type** box, select **Short-term** and click **Apply** .
- To add the fourth load case, in the **List of load cases** area, click **Add load case (to the end)** .
- For load case 4 – in the **Edit selected load case** area, in the **Type** box, select **Short-term** and click **Apply** .
- To continue generating the first load case, in the **List of load cases** area, select the row **1. Load case 1** and click **Set as current one**. You could also define the current load case by double-clicking appropriate row in the list.
- Close the **Edit load cases** dialog box.

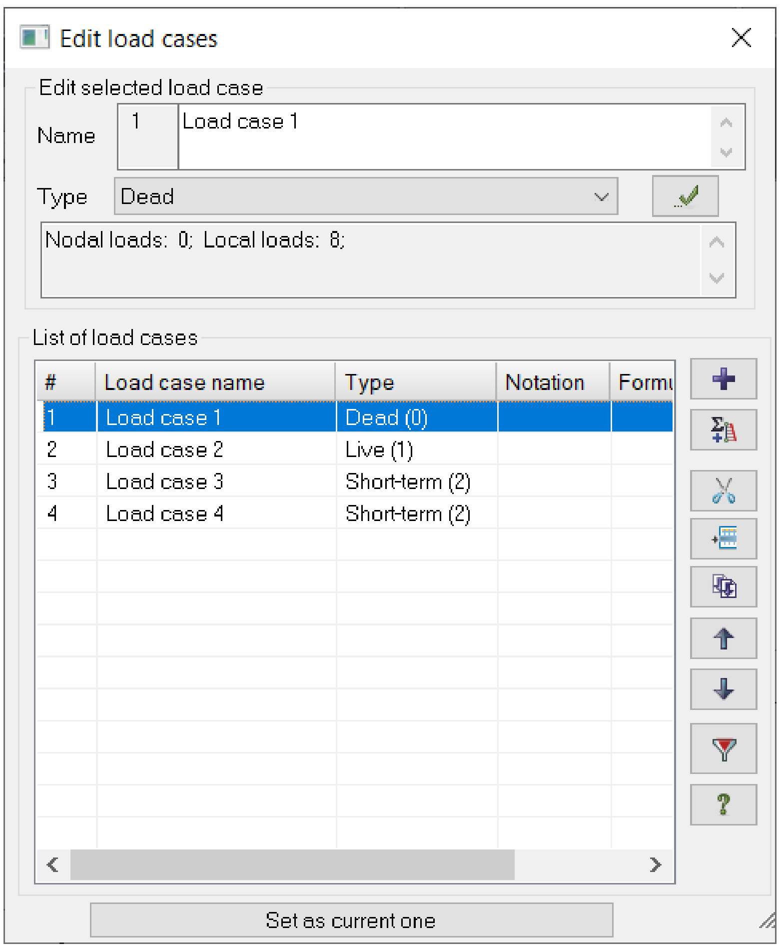


Figure 19.17 **Edit load cases** dialog box

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

To create load case No.1:

- 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.19.18), click **All elements** and specify **Load factor** as equal to **1.1**. (as the unit weight is specified as normative value, it should be converted to design value).
- Then click **Apply** (uniformly distributed load equal to unit weight of elements is automatically applied to all elements of the structure).

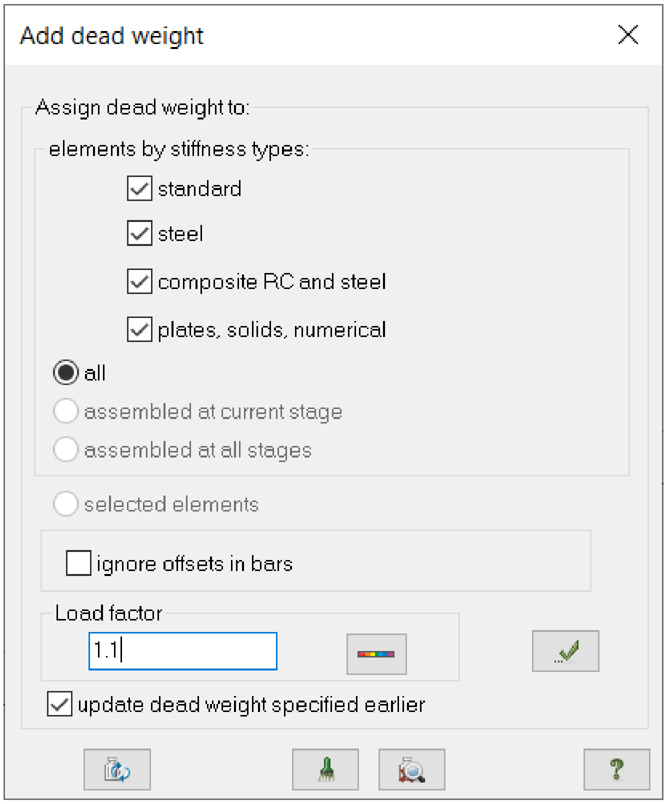


Figure 19.18 **Add dead weight** dialog box

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.
- Select all elements with the pointer.
- 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.19.19), specify **Global** coordinate system and direction along the **Z**-axis (default parameters).

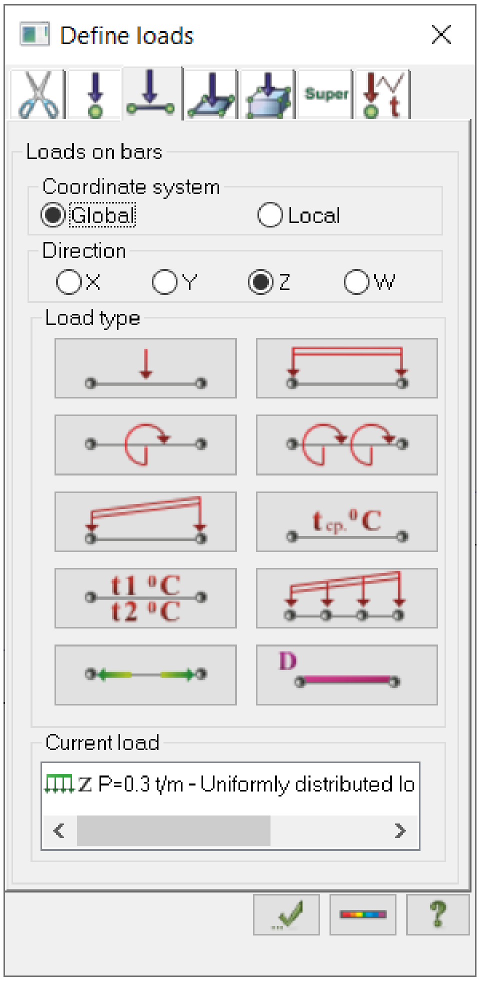
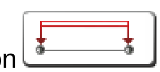


Figure 19.19 **Define loads** dialog box

- In the **Load type** area, click **Uniformly distributed load** button  .
- In the **Load parameters** dialog box specify P = 0.3 t/m (see Fig.19.20).
- Click **OK**  .

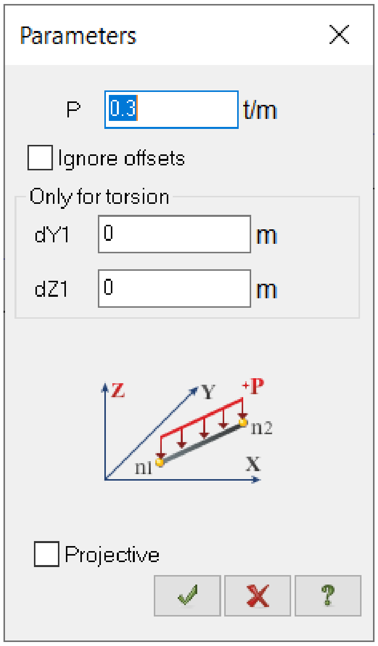


Figure 19.20 **Load parameters** dialog box

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 with the pointer elements of the first span: elements No.1, 2, 3 and 4.
- In the **Load type** area, click **Uniformly distributed load** button .
- In the **Load parameters** dialog box specify P = 0.87 t/m.
- Click **OK** .

To create load case No.4:

- Do not close the **Define loads** dialog box. Then change number of the current load case for 4 (button on the Status bar). In this case, current load will remain as equal to P = 0.87 t/m as it was defined for the previous load case.
- Select with the pointer elements of the second span: elements No.5, 6, 7 and 8.
- In the **Define loads** dialog box, click **Apply** .

Step 8. Generating DCL table

- On the **Analysis** ribbon tab, select the **More calculations** panel and click **DCL** button .
- In the **Warning** box (see Fig.19.21), click **OK**.

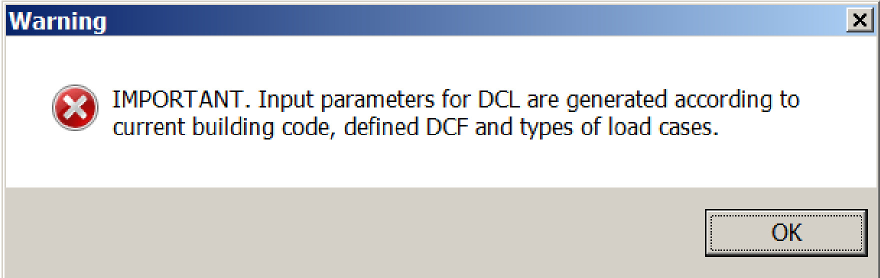


Figure 19.21 **Warning** box

As the type of load cases was defined in the **Edit load cases** *dialog box (see Fig.19.17), the DCL table is generated automatically with parameters accepted by default for every load case. Now you have to modify parameters for the third and the fourth load cases and define combinations.*

- In the **Design combinations of loads** dialog box (see Fig.19.22), select building code **SP 20.13330.2011/2016** and for load cases 3 and 4, in the **Mutually exclusive** cell, define **1**.
- To define combinations, do the following steps:
  - in the list of combinations, select the first row (**1 main**) and click **Add**.
- To save defined data, click **Save data** .
- To close the **Design combinations of loads** dialog box, click **Close**.

Design combinations of loads (DCL) are calculated as the sum of appropriate values of nodal displacements and forces (stresses) in elements. These values are added according to building codes (unlike calculation of DCF where extreme values of stresses at specific points of bar sections are used as criterion for determination of dangerous combinations).

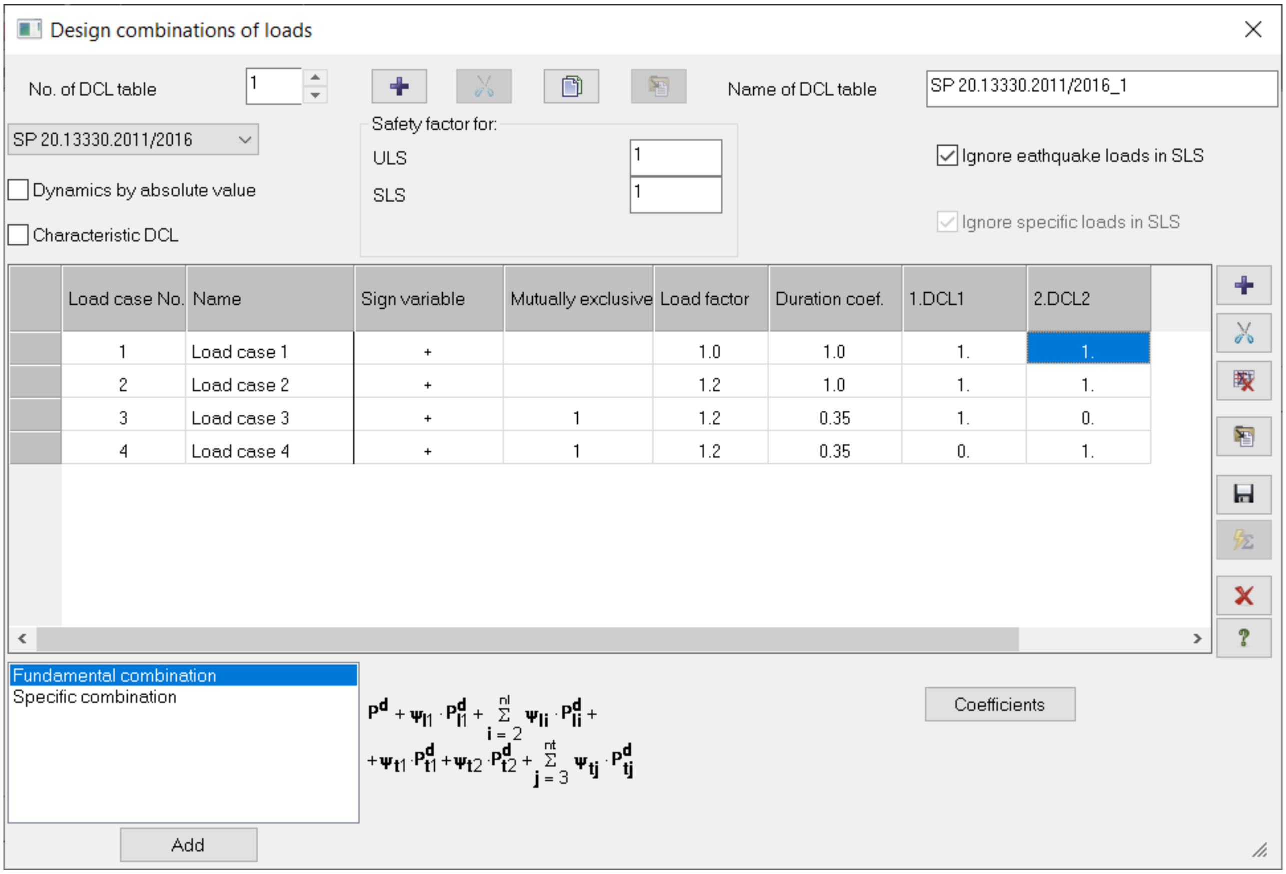


Figure 19.22 **Design combination of loads** dialog box

To save data about design model:

- On the **LIRA-SAPR menu** (Application menu), click **Save** command (or just click this button on the Quick Access Toolbar).

Step 9. 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** .

Step 10. Review and evaluation of static & dynamic analyses 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 (see Fig.19.23). To display the model without nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button .

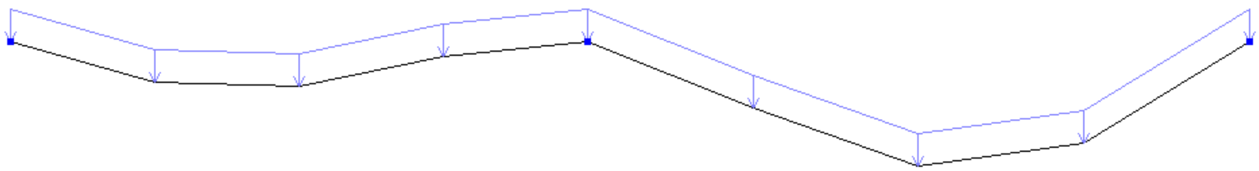


Figure 19.23 Design model with account of nodal displacements

To present displacement mosaic plot:

- To present mosaic plot of displacements along the Z-axis, on the **Results** ribbon tab, on the **Deformations** panel, select the **Displacement mosaic 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 **My** (see Fig.19.24), on the **Results** tab, select **Forces in bars** panel and click **Diagrams My** button .

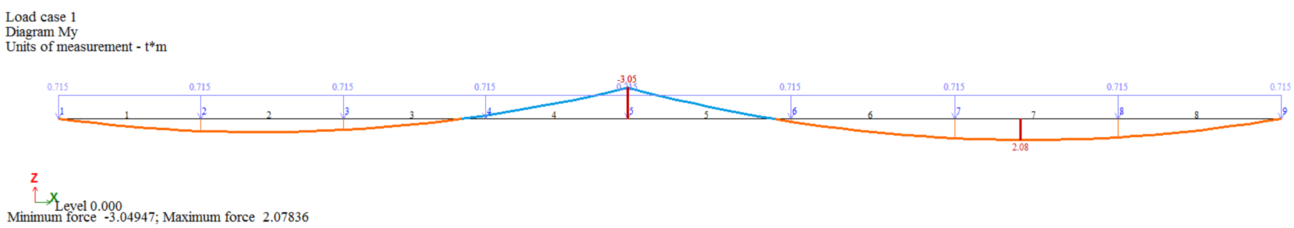


Figure 19.24 Diagram of bending moments **My**

- To display diagram **Qz** (see Fig.19.25), on the **Results** tab, select **Forces in bars** panel and click **Diagrams Qz** button .

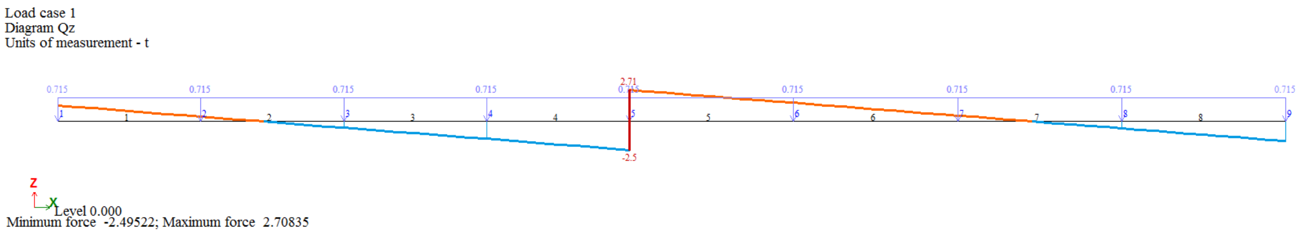


Figure 19.25 Shear force diagram **Qz**

- To display mosaic plots **Qz**, 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 evaluate analysis results by DCL:

- To present analysis results by DCL, on the Status bar (displayed at the bottom of the screen), click **Results by DCL** button .
- Diagrams of internal forces are presented on the screen in the same way as described above.

- To switch to another DCL number, on the Status bar in the **Load case No.** list, select appropriate row and click **Apply** .

<u>To generate and review tables of analysis results:</u>

- To present table with design combinations of loads 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.19.26), select **DCL design** in the list.
- Click **Apply**.

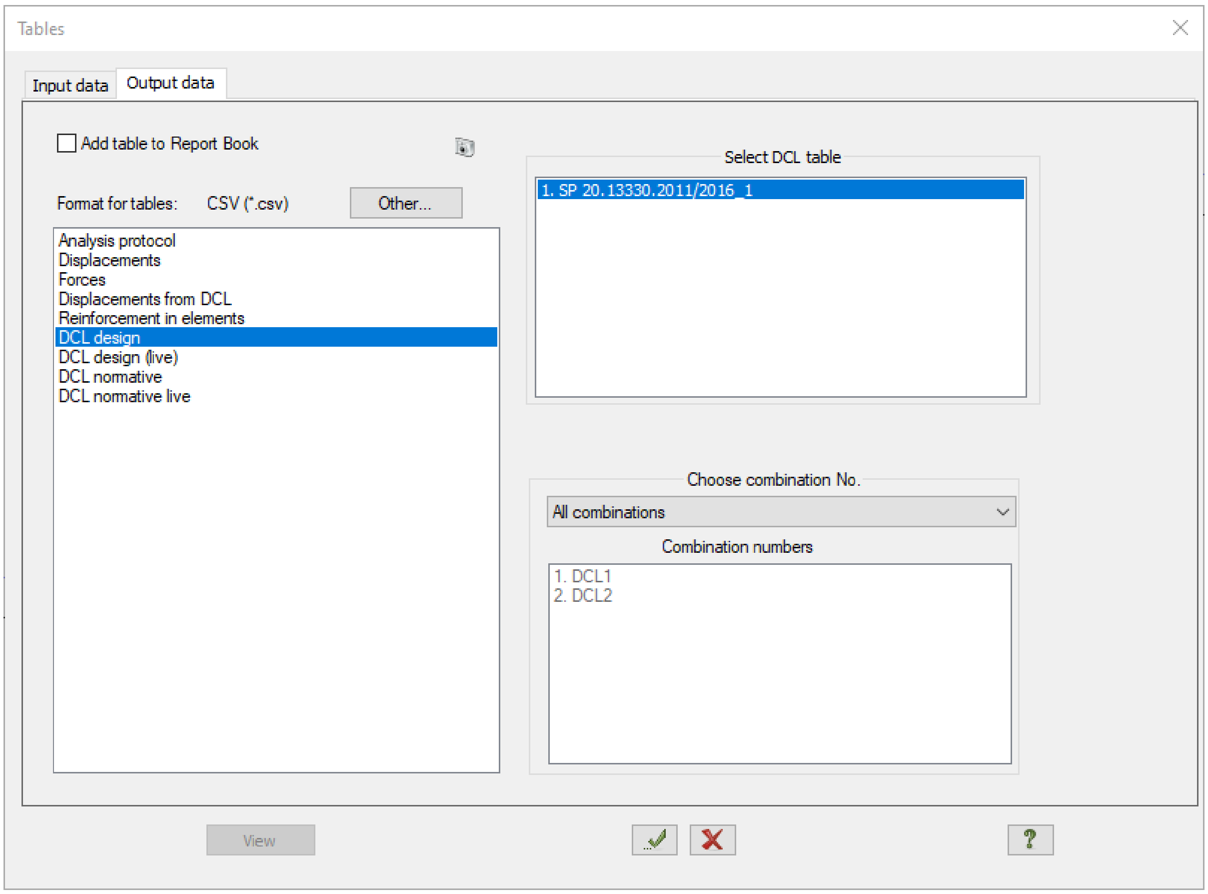


Figure 19.26 **Standard tables** dialog box

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* **Other**. *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.

- In the **DCL design (SP 20.13330.2011/2016_1)** window, select the **Bars** tabbed page.
- Click the **Filter** button . In the **Filter for tables** dialog box, define the following data:

- in the left part of the window, in the **Column names and their properties** area, select the MY check box;
- in the right part of the window, select the **Calculate minimum and maximum automatically** check box (denoted as **A**);
- click **Apply**.

➪ Close the **Filter for tables** dialog box.

➪ Close the **DCL design (SP 20.13330.2011/2016_1)** window.

With the Filter it is possible to display appropriate values in colour, to hide the table rows that contain values outside the specified range from Min to Max, to hide the specified columns and to reduce the number of decimal places for figures in appropriate column or in all columns.

Step 11. Modelling nonlinear load cases

Engineering Design (NL Engineering) – new approach to analysis of structures with account of physical nonlinearity. According to forces in characteristic combination (DCF and DCL are not considered), step-type analysis is carried out with analysis of reinforcement in element sections at every step. By results obtained at the last step in analysis of reinforcement, specific stiffness parameters are calculated. These parameters are considered the final ones and are automatically assigned to elements of the model.
Then, the program will automatically perform ordinary linear analysis for all load cases (including dynamic ones). In this case, displacements, DCF and DCL are calculated and analysis of reinforcement is carried out.
Such approach enables you to analyse reinforcement that is to a greater extent approximate to the real work of structure. In this case, time for defining input data and time for analysis (relative to step-type physically nonlinear analysis) is considerably reduced.

➪ On the **Analysis** ribbon tab, on the **More calculations** panel, click **NL Engineering** EN .The **Model nonlinear load cases** dialog box is displayed on the screen (see Fig.19.27).

In the dialog box it is necessary to generate the table of load cases that will be 'characteristic' in analysis of specific stiffness. To define the load case as included into 'characteristic' combination, specify coefficient different from zero. If coefficient value is equal to zero, it means that this load case is not included into 'characteristic' combination.
Combination of load cases 1, 2 and 4 is considered as 'characteristic' combination for this model.

➪ In the **Model nonlinear load cases of the structure** dialog box, to define 'characteristic combination of load cases' (load cases 1, 2 and 4), perform the following steps:

- In the **Coef.** column for the first, the second and the fourth load cases, define 1.

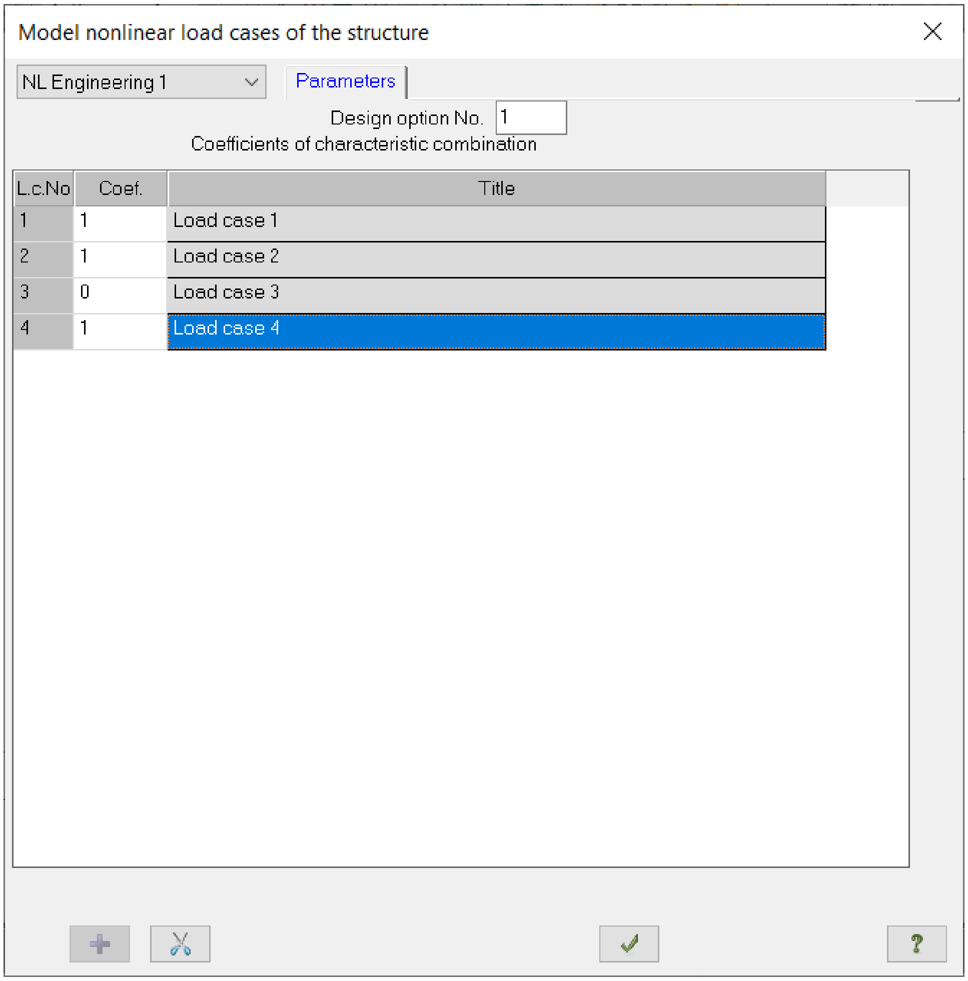


Figure 19.27 **Model nonlinear load cases of structure** dialog box

- To confirm data, click **OK** .

To save data about design model:

- On the **LIRA-SAPR menu** (Application menu), click **Save** command (or just click this button on the Quick Access Toolbar) .
- In the **Save as** dialog box specify the following data:
  - file name – **Example19_2**;
  - location where you want to save this file (**Data** folder is displayed by default).
- Click **Save**.

Step 12. Analysis of beam with NonLinear Engineering Design (NL Engineering) system

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

Step 13. NL Engineering: review and evaluation of analysis results

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

To present displacement mosaic plots:

- To present mosaic plot of displacements along the Z-axis, on the **Results** ribbon tab, on the **Deformations** panel, select the **Displacement mosaic 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 **My**, on the **Results** tab, select **Forces in bars** panel and click **Diagrams My** button .
- To display diagram **Qz**, on the **Results** tab, select **Forces in bars** panel and click **Diagrams Qz** button .

To present mosaic plots of nonlinear stiffness:

- To present mosaic plot for axial stiffness of bar EF, on the FORCES menu, point to **Nonlinear stiffness** and click **EF - Stiffness of bar**.
- To present mosaic plot for flexural stiffness of bar EIy about the local Y1-axis, on the FORCES menu, point to **Nonlinear stiffness** and click **EIy - Stiffness of bar**.

Step 14. Review and evaluate results from analysis of reinforcement

*When analysis procedure is complete, to review and evaluate results from analysis of reinforcement, select the **RC** ribbon tab (for Ribbon Plus 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 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 the **Close** button.

- To switch to the mode for presentation of asymmetric reinforcement in rebars, on the **RC** ribbon tab, select **Reinforcement in bars** panel and click **Asymmetric 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 **RC** ribbon tab, the **Reinforcement in 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 **RC** ribbon tab, the **Reinforcement in bars** panel).

To generate and review table with analysis results for reinforcement:

- On the **RC** 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, select the **Reinforcement in elements** option in the list.
- Click **Apply** .
- After review and evaluation, close the table window.

