

Example 7. Nonlinear analysis of beam on two spans with account of creep in concrete

In this lesson you will learn how to:

- generate design model of beam on two spans;
- define parameters of physical nonlinearity for materials with account of creep in concrete and define parameters for reinforcement;
- model nonlinear load cases.

Description:

Model of the beam and its boundary conditions are presented in Figure 7.1.

Sections for elements of the beam are presented in Figure 7.2.

Material for beam – reinforced concrete B25, reinforcement A-III.

State of design model is evaluated after 365 and 730 days.

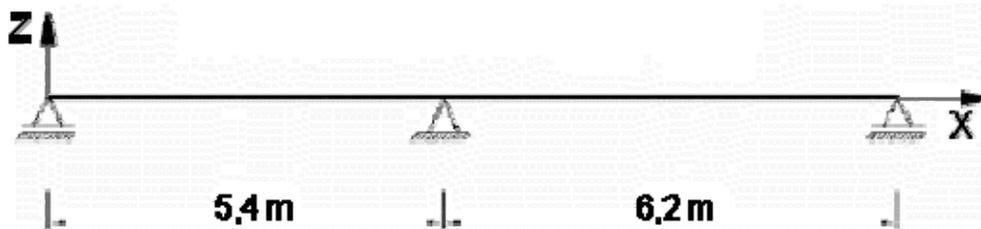


Figure 7.1 Model of the beam

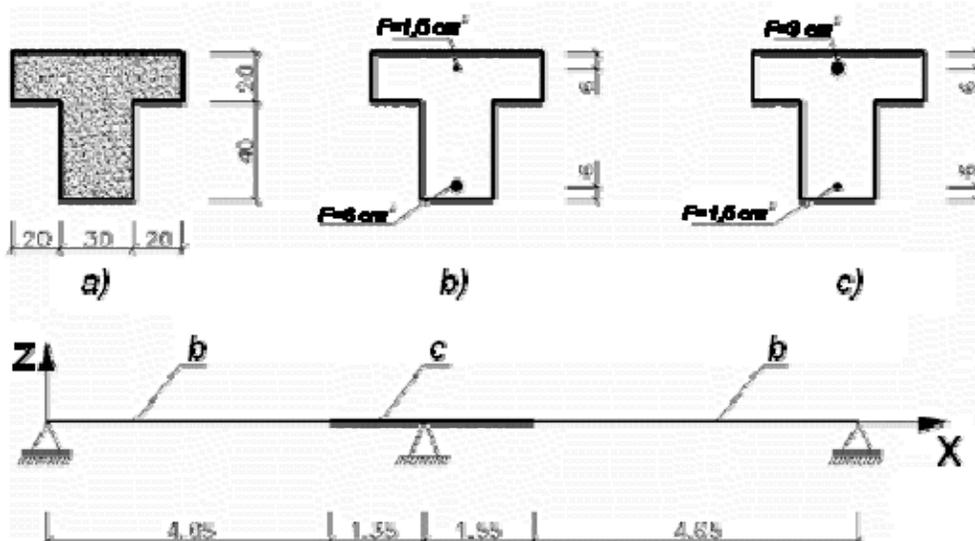


Figure 7.2 Sections for elements of the beam:

- a) section;
- b) arrangement of reinforcement in span section;
- c) arrangement of reinforcement at support section.

Loads:

- load case 1 – dead weight (see Figure 7.3);

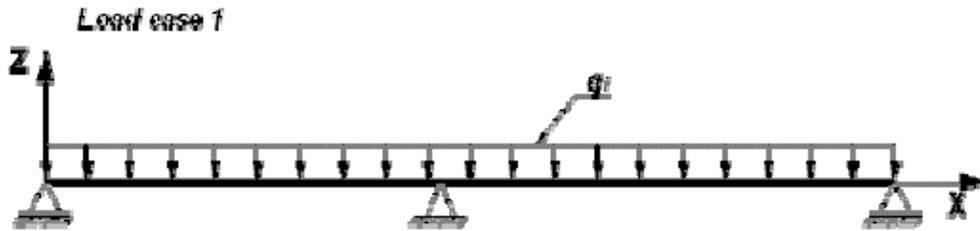


Figure 7.3 Load case No.1 for beam

- load case 2 – uniformly distributed load $q_2 = 0.3$ t/m (see Figure 7.4);

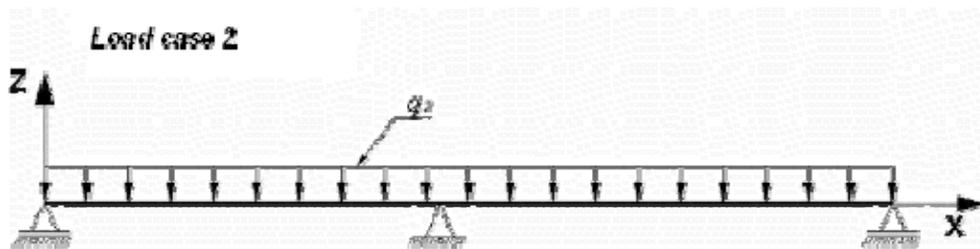


Figure 7.4 Load case No.2 for beam

- load case 3 – uniformly distributed load in the first span $q_3 = 0.87$ t/m (see Figure 7.5);

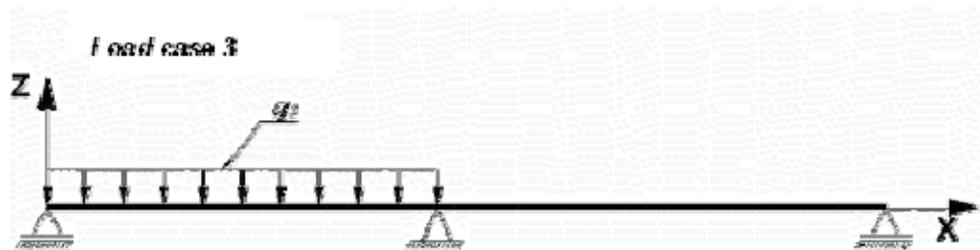


Figure 7.5 Load case No.3 for beam

- load case 4 – uniformly distributed load in the second span $q_4 = 0.87$ t/m (see Figure 7.6);

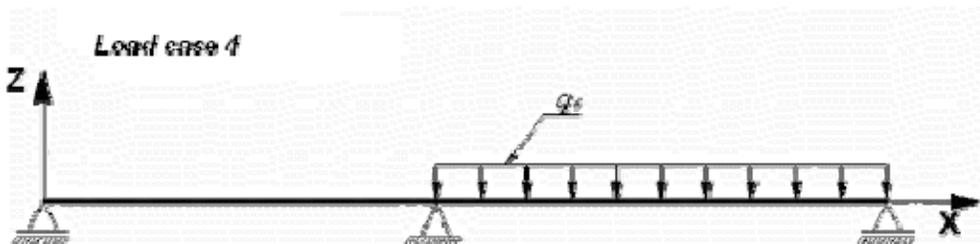


Figure 7.6 Load case No.4 for beam

- ⇒ On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2015** and then click **LIRA-SAPR 2015**.

Step 1. Creating new problem

- ⇒ On the FILE menu, click **New** (button  on the toolbar).
- ⇒ In the **Model type** dialog box (see Figure 7.7) specify the following data:
- problem name – **Example7**;
 - problem code (by default it coincides with the problem name);
 - model type – **2 – Three degrees of freedom per node** (two translations and rotation) X0Z.
- ⇒ Click **OK** .

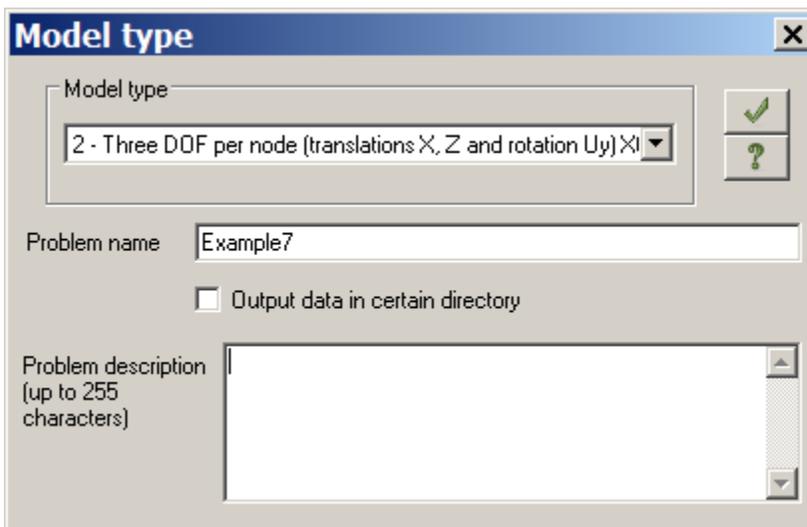


Figure 7.7 **Model type** dialog box



*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 2 (Three DOF per node)***

*command . One more way to do the same: on the Quick Access Toolbar, click **New** and in the drop-*

*down menu select **Model type 2 (Three DOF per node)** command . Then you should define only problem name.*



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 7.8).

⇒ Click **Apply**  .

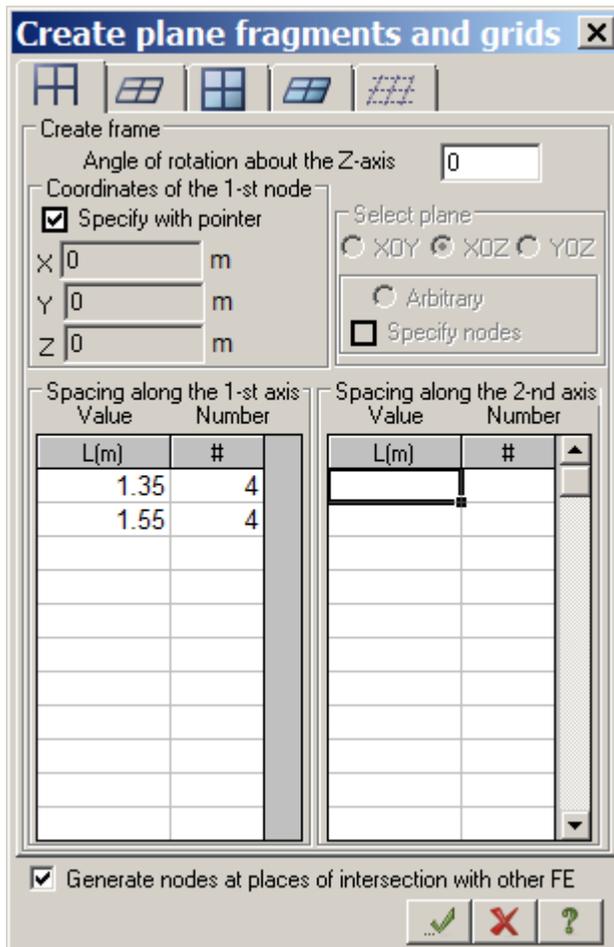


Figure 7.8 **Create plane fragments and grids** dialog box

To save data about design model:

- ⇒ On the **LIRA-SAPR menu** (Application menu), click **Save** command  .
- ⇒ In the **Save as** dialog box specify the following data:
 - file name – **Example7**;
 - location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

Step 3. Defining boundary conditions

To present numbers of nodes and elements on the screen:

- ⇒ On the **Select** toolbar, click **Flags of drawing** button  .

- ⇒ In the **Display** dialog box, select the **Element numbers** check box on the **Elements** tab.
- ⇒ On the **Nodes** tab, select the **Node numbers** check box.
- ⇒ Click **Redraw** .

The model with numbers of nodes and elements is presented in Fig.7.9.



Figure 7.9 Design model with numbers of nodes and elements

To select nodes No.1 and 9:

- ⇒ 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.7.10) specify directions along which displacements of nodes are not allowed (Z). To do this, select appropriate check boxes.
- ⇒ Click **Apply**  (the nodes will be coloured blue).

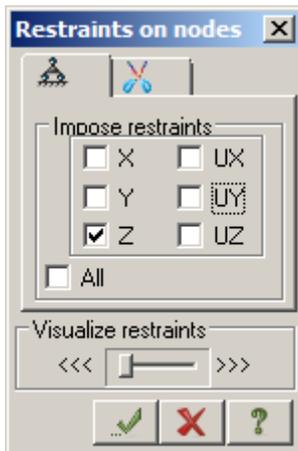


Figure 7.10 **Restraints on nodes** dialog box

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

Step 4. Defining material properties to elements of the beam

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.7.11a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.7.11b).

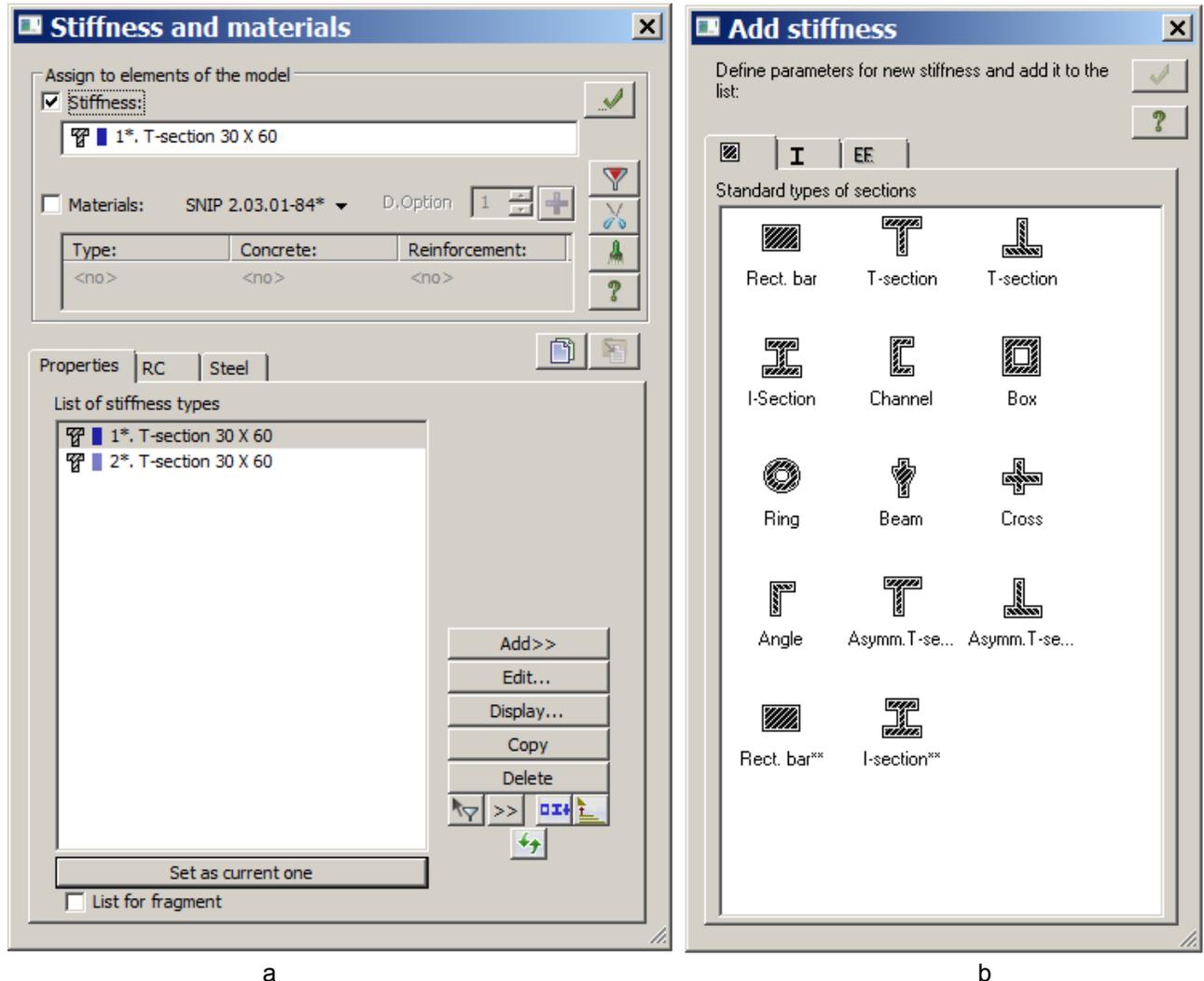


Figure 7.11 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ 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.7.12):
 - geometric properties – $B = 30$ cm; $H = 60$ cm; $B_1 = 70$ cm; $H_1 = 20$ cm;
 - unit weight of material – $R_o = 2.5$ t/m³.
- ⇒ In the **Define standard section** dialog box, select the **Nonlinear parameters** check box.

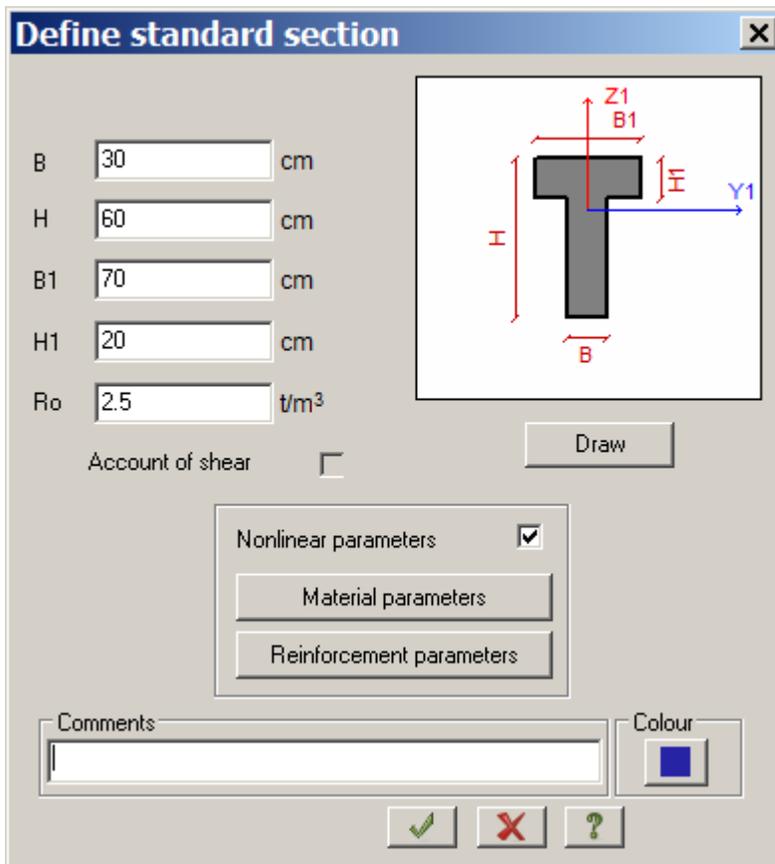


Figure 7.12 Define standard section dialog box

- ⇒ To define material, click **Material parameters**. The **Nonlinear stress-strain diagrams for materials** dialog box appears on the screen (see Figure 7.13).
- ⇒ In this dialog box for the main material select **25 – exponential (normative strength)** in the **Nonlinear stress-strain diagram** list box.
- ⇒ Under **Parameters for stress-strain diagram**, double-click appropriate field to specify the following parameters for main material (concrete):
 - concrete name – B25;
 - concrete type – TA.

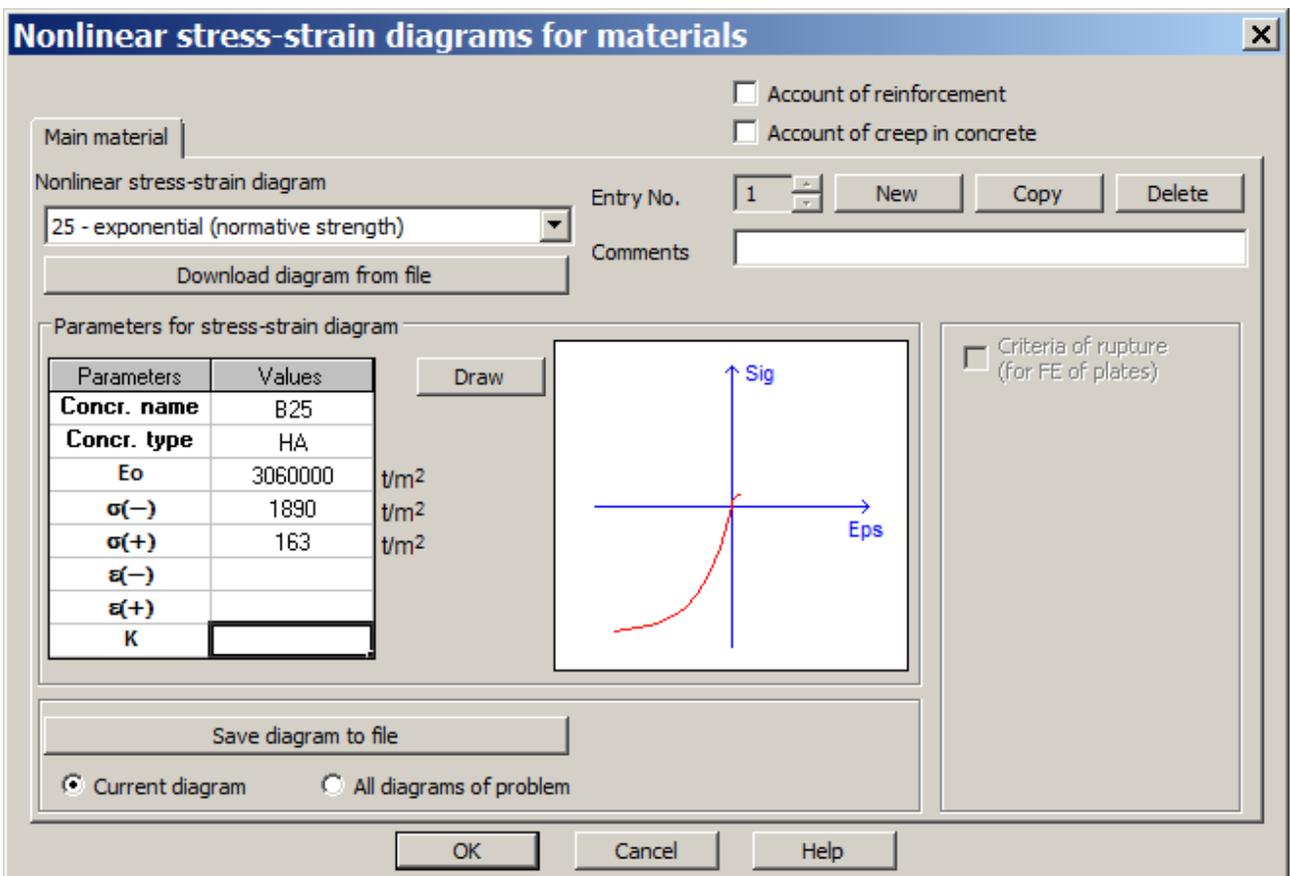


Figure 7.13 Nonlinear stress-strain diagrams for materials dialog box for main material

- ⇒ In the same dialog box, click **Account of reinforcement** (see Figure 7.14) and select the **Reinforcement** tab.
- ⇒ In the **Nonlinear stress-strain diagram** list box, select **11 – exponential**.
- ⇒ Under **Parameters for stress-strain diagram** specify the following parameters for reinforcement (for the U.S. keyboard layout):
 - modulus of elasticity – $Eo(-) = 2e7 \text{ t/m}^2$;
 - modulus of elasticity – $Eo(+) = 2e7 \text{ t/m}^2$;
 - ultimate stress $s(-) = -36000 \text{ t/m}^2$;
 - ultimate stress $s(+) = 36000 \text{ t/m}^2$.

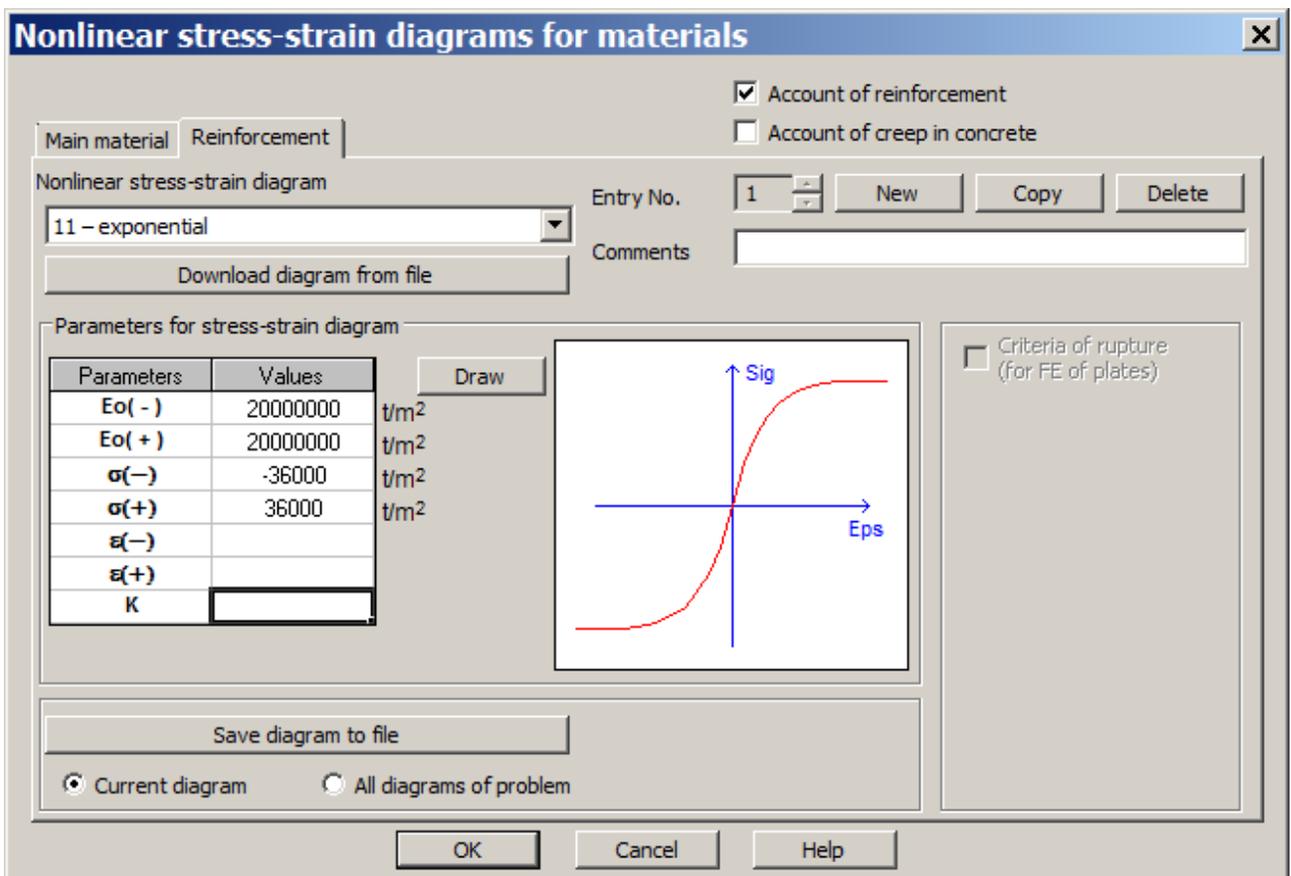


Figure 7.14 Nonlinear stress-strain diagrams for materials dialog box for reinforcement

- ⇒ In the same dialog box, select the **Account of creep in concrete** (see Figure 7.15) and click the **Creep in concrete** tab.
- ⇒ In the **Creep diagram in concrete** list box, select the line **41 – exponential for creep** (Eurocode prEN 1992-1-1).
- ⇒ Under **Parameters for stress-strain diagram** specify the following parameters:
 - notional creep coefficient – $\varphi_0 = 2$;
 - coefficient – $\beta_H = 657.82$.
- ⇒ To confirm the specified data, click **OK**.

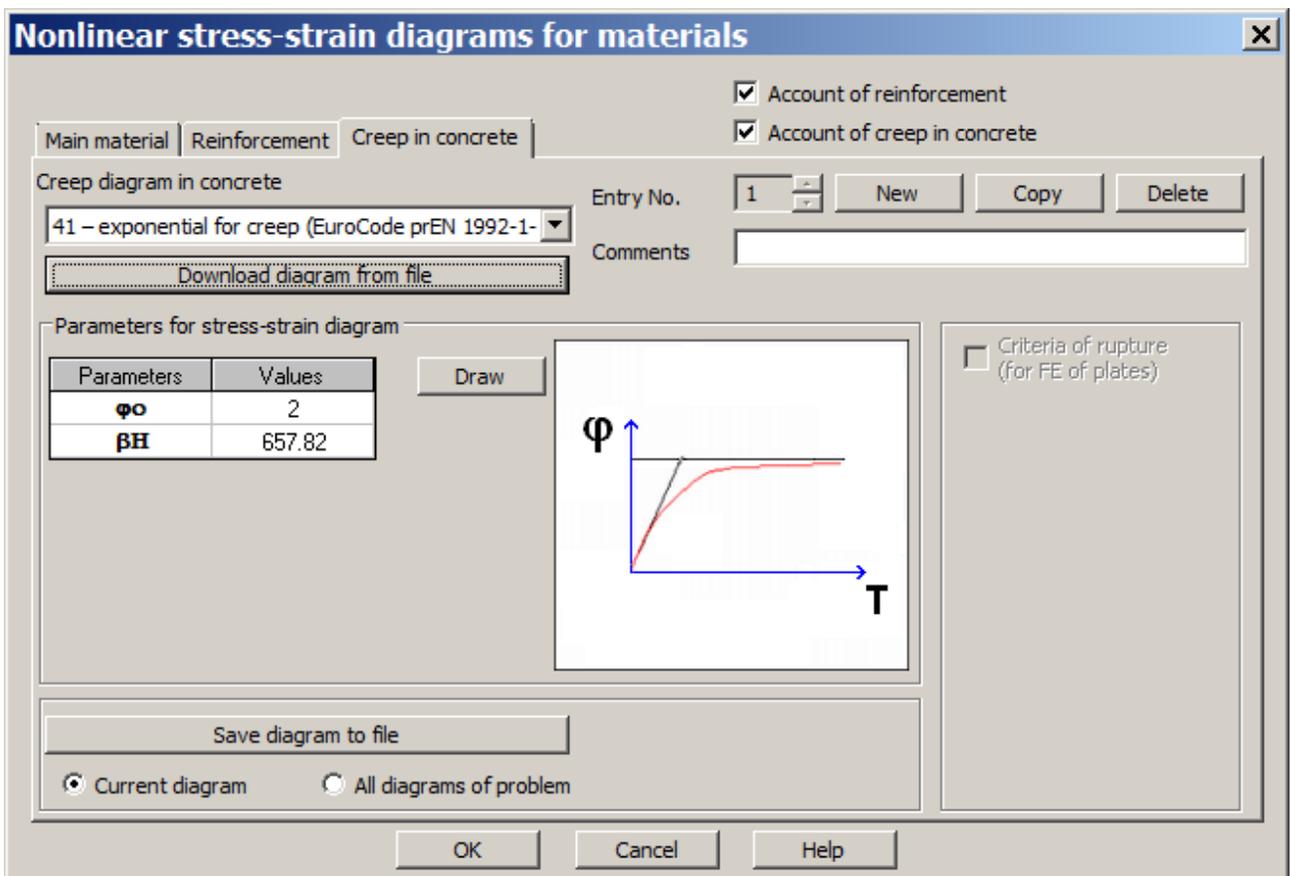
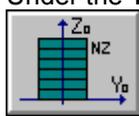


Figure 7.15 Nonlinear stress-strain diagrams for materials dialog box for creep in concrete

- ⇒ To define location and area of reinforcement, in the **Define standard section** dialog box (see Figure 7.12), click **Reinforcement parameters**.
- ⇒ In the **Parameters of physical nonlinearity for bars** dialog box (see Figure 7.16), click the **Point**



- ⇒ Define parameters for the first layer of reinforcement:
 - area of reinforcement – $F_a = 6 \text{ cm}^2$;
 - distances from the Z-axis (Y_i) and from the Y-axis (Z_i) to the point – $y = 0 \text{ cm}$; $z = 6 \text{ cm}$.
- ⇒ Under the **Type of reinforcement**, in the **Number of reinforcement layer** list box select number 2.
- ⇒ Define parameters for the second layer of reinforcement:
 - area of reinforcement – $F_a = 1.5 \text{ cm}^2$;
 - distances from the Z-axis (Y_i) and from the Y-axis (Z_i) to the point – $y = 0 \text{ cm}$; $z = 54 \text{ cm}$.
- ⇒ Under the **Type of cross-section division**, click **Division of cross-section into elementary strips** icon .
- ⇒ To preview the section, click **Draw**.
- ⇒ Click **OK**.
- ⇒ In the **Define standard section** dialog box, click **OK** .

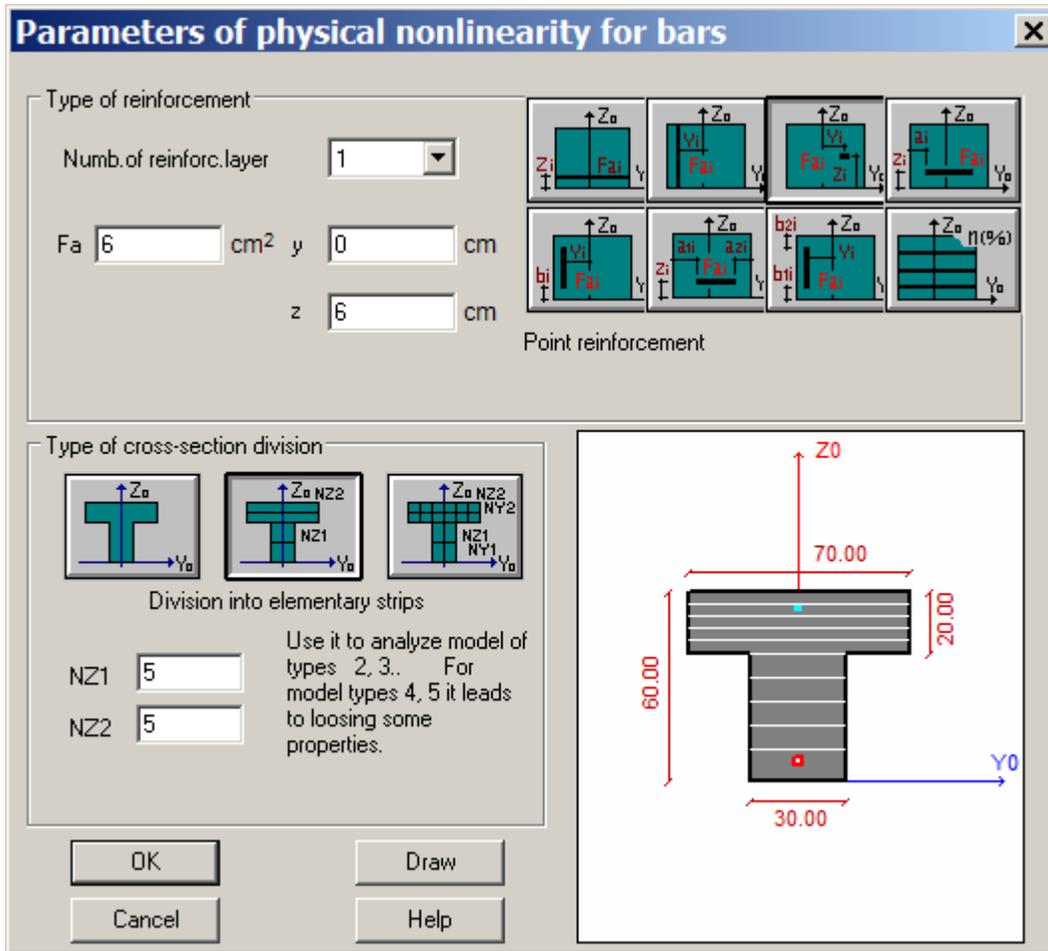
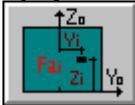


Figure 7.16 Parameters of physical nonlinearity for bars dialog box

- ⇒ In the **Stiffness of elements** dialog box (see Figure 7.11), in the **List of stiffness types**, select '1*.T-section 30x60'.
- ⇒ Click **Copy**.
- ⇒ In the **Stiffness of elements** dialog box, in the **List of stiffness types**, select '2*.T-section 30x60'.
- ⇒ Click **Edit**.
- ⇒ In another **Define standard section** dialog box, click **Reinforcement parameters**.
- ⇒ In the **Parameters of physical nonlinearity for bars** dialog box (see Figure 7.17), click the **Point**

reinforcement icon .

- ⇒ Define parameters for the first layer of reinforcement:
 - area of reinforcement – $F_a = 1.5 \text{ cm}^2$;
 - distances from the Z-axis (Y_i) and from the Y-axis (Z_i) to the point – $y = 0 \text{ cm}$; $z = 6 \text{ cm}$.
- ⇒ Under the **Type of reinforcement**, in the **Number of reinforcement layer** list box select number 2.
- ⇒ Define parameters for the second layer of reinforcement:
 - area of reinforcement – $F_a = 9 \text{ cm}^2$;
 - distances from the Z-axis (Y_i) and from the Y-axis (Z_i) to the point – $y = 0 \text{ cm}$; $z = 54 \text{ cm}$.

- ⇒ Click **OK** in the **Parameters of physical nonlinearity for bars** dialog box.
- ⇒ In the **Define standard section** dialog box, click **OK** .

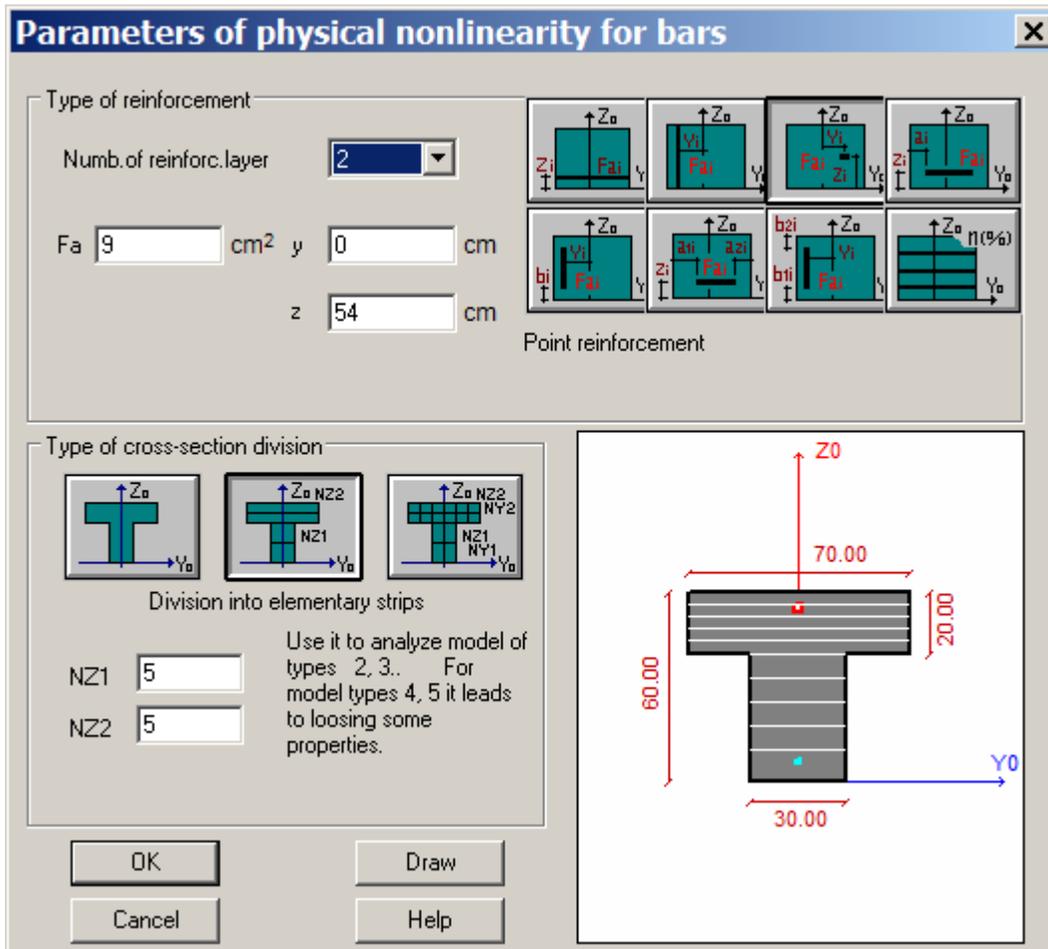


Figure 7.17 **Parameters of physical nonlinearity for bars** dialog box

To change FE type:

- ⇒ On the **Select** toolbar, click **Select horizontal bars** (button  on the toolbar).
- ⇒ Select all horizontal elements of the model with the pointer.
- ⇒ On the **Advanced edit options** ribbon tab, on the **Model** panel, click **Change FE type** (button  on the toolbar).
- ⇒ In the **Change FE type** dialog box (see Figure 7.18), in the **list of FE types**, select **FE type 210 – physically nonlinear arbitrary 3D bar**.
- ⇒ Click **Apply** .

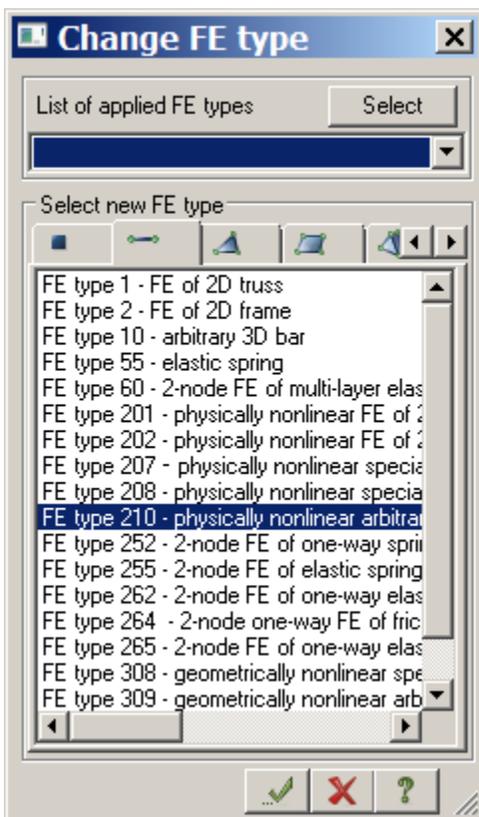


Figure 7.18 Change FE type dialog box

To assign stiffness to elements of the beam:

- ⇒ 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.7.11a), in the **List of stiffness types**, select stiffness type '1*.T-section 30x60'.
- ⇒ Click **Set as current type**. In this case selected type will be displayed in the **Stiffness** box in the **Assign to elements of the model** area. To assign current type of stiffness, you could also double-click appropriate row in the list.
- ⇒ Select with the pointer elements No.1, 2, 3, 6, 7 and 8.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ In the **Stiffness of elements** dialog box, in the **List of stiffness types**, select '2*.T-section 30x60'.
- ⇒ Click **Set as current type**.
- ⇒ Select with the pointer elements No.4 and 5.
- ⇒ In the **Stiffness of elements** 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.7.19), click **All elements** and specify **Load factor** as equal to **1**. Then click **Apply**  (dead weight of elements is added automatically).

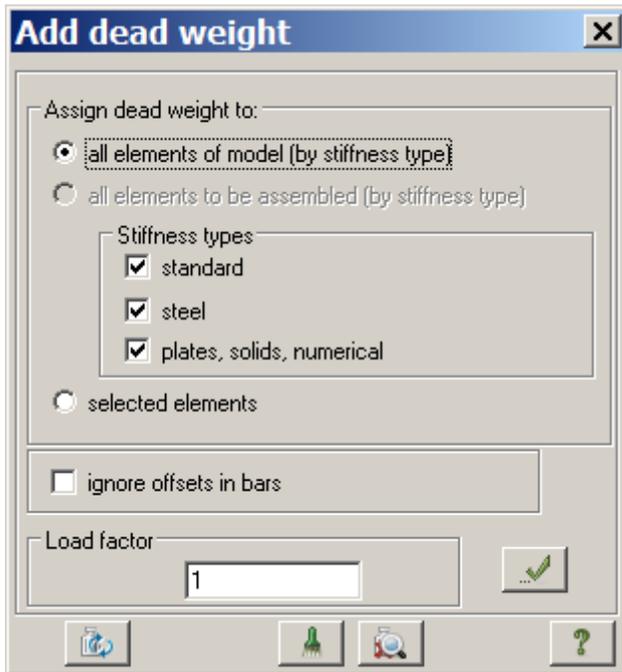


Figure 7.19 **Add dead weight** dialog box

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.7.20), specify **Global** coordinate system and direction along the **Z-axis** (default parameters).

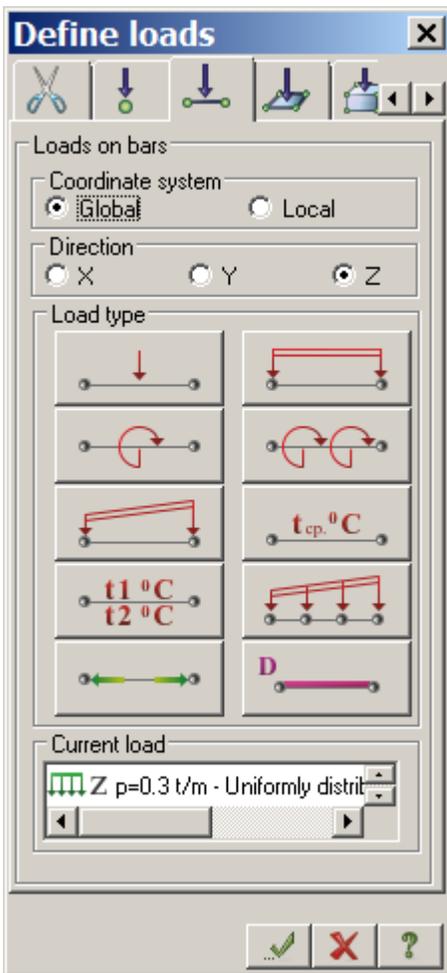


Figure 7.20 Define loads dialog box

- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 0.3 \text{ t/m}$ (see Fig.7.21).
- ⇒ Click **OK** .

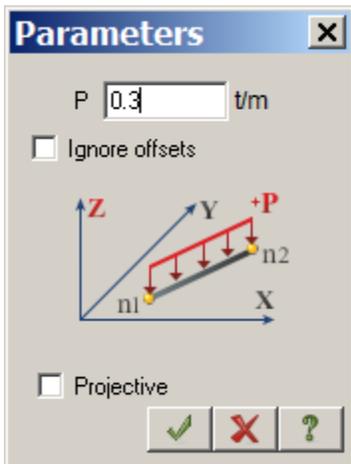


Figure 7.21 Load parameters dialog box

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

To create load case No.3:

- ⇒ To change the number of the current load case, click the **Next load case** button  located on the Status bar or on the toolbar.
- ⇒ 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 6. Modelling nonlinear load cases with account of creep in concrete

To analyse beam with load in different spans, it is necessary to apply loads in two sequences.

To define the first sequence of load application – load cases 1, 2 and 3 (load in the first span):

- ⇒ On the **Analysis** ribbon tab, on the **Nonlinearity** panel, click **Step-type method** (button  on the toolbar).
- ⇒ In the **Model nonlinear load cases of structure** dialog box (see Figure 7.22), click the **Add** button  (in the left part of the dialog box, under **History**, the first load history is added and the row with load case indicated with question mark will become selected automatically).
- ⇒ For the first load case define the following parameters:
 - load case No. – 1;
 - select **Step (1)** in the **Analysis method** list box;
 - select **Equal steps** option and define number of steps **5**;
 - select **Displacement and forces after every step** in the **Print results** list box;
 - in the **Display intermediate results** list, select **Display all** option.
- ⇒ Click **Apply** .
- ⇒ To add rows for parameters of the second load case, select the row for the first load case and then click the **Add** button .
- ⇒ Select the row for the second load case and define the following parameters:
 - load case No. – 2;

- select **Step (1)** in the **Analysis method** list box;
 - under the **Values of load factors for steps**, click **Equal steps**. Then define number of steps equal to **30**.
 - select **Displacement and forces after every step** in the **Print results** list box;
 - in the **Display intermediate results** list, select **Display all** option.
- ⇒ Click **Apply** .
- ⇒ To add rows for parameters of the third load case, select the row for the second load case and then click the **Add** button .
- ⇒ Select the row for the third load case and define the following parameters:
- load case No. – **3**;
 - select **Step (1)** in the **Analysis method** list box;
 - under the **Values of load factors for steps**, click **Equal steps**. Then define number of steps equal to **30**.
 - select **Displacement and forces after every step** in the **Print results** list box;
 - in the **Display intermediate results** list, select **Display all** option.
- ⇒ Click **Apply** .

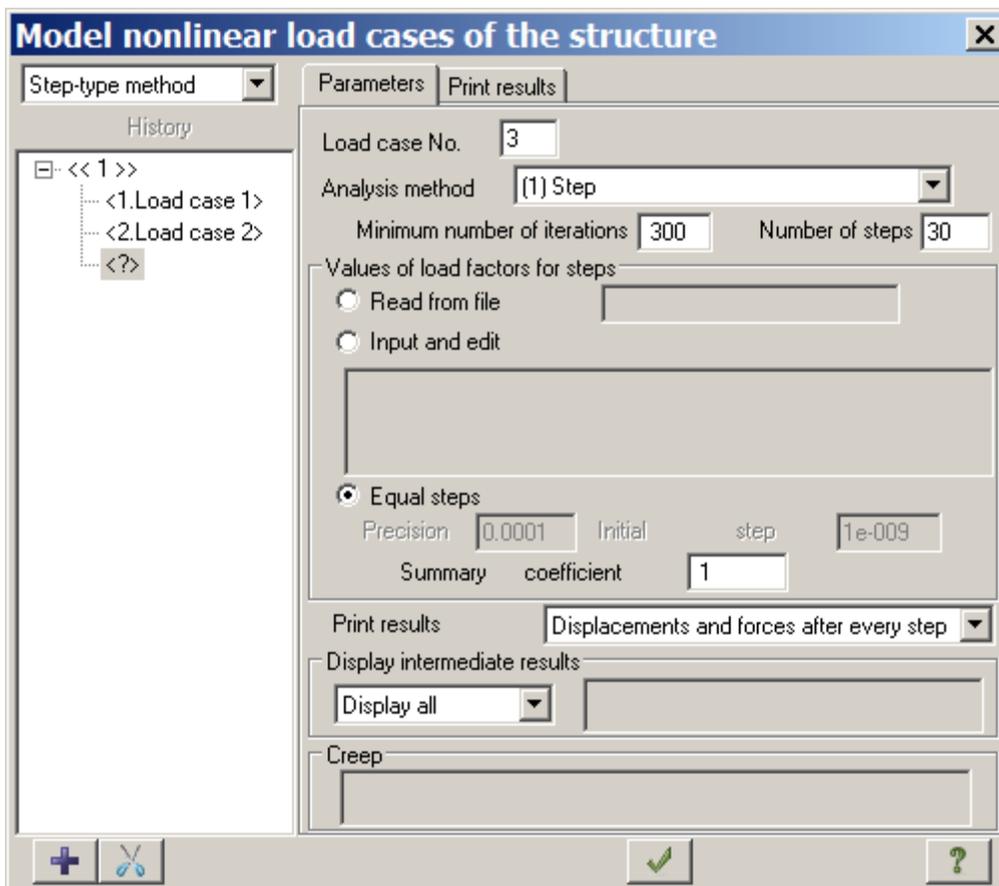


Figure 7.22 **Model nonlinear load cases of structure** dialog box

To define the second sequence of load application – load cases 1, 2 and 4 (load in the second span):

- ⇒ In the **Model nonlinear load cases of structure** dialog box (see Figure 7.23), under **History**, select the first history of loadings and click the **Add** button .
- ⇒ When the line indicated with question mark is selected, define the following parameters:
- load case No. – 1;
 - select **Step (1)** in the **Analysis method** list box;
 - under the **Values of load factors for steps**, click **Equal steps**. Then define number of steps equal to 5.
 - select **Displacement and forces after every step** in the **Print results** list box;
 - in the **Display intermediate results** list, select **Display all** option.
- ⇒ Click **Apply** .
- ⇒ To add rows for parameters of the second load case, select the row for the first load case and then click the **Add** button .
- ⇒ Select the row for the second load case and define the following parameters:
- load case No. – 2;
 - select **Step (1)** in the **Analysis method** list box;
 - under the **Values of load factors for steps**, click **Equal steps**. Then define number of steps equal to 30.
 - select **Displacement and forces after every step** in the **Print results** list box;
 - in the **Display intermediate results** list, select **Display all** option.
- ⇒ Click **Apply** .
- ⇒ To add rows for parameters of the fourth load case, select the row for the second load case and then click the **Add** button .
- ⇒ Select the row for the fourth load case and define the following parameters:
- load case No. – 4;
 - select **Step (1)** in the **Analysis method** list box;
 - under the **Values of load factors for steps**, click **Equal steps**. Then define number of steps equal to 30.
 - select **Displacement and forces after every step** in the **Print results** list box;
 - in the **Display intermediate results** list, select **Display all** option.
- ⇒ Click **Apply** .

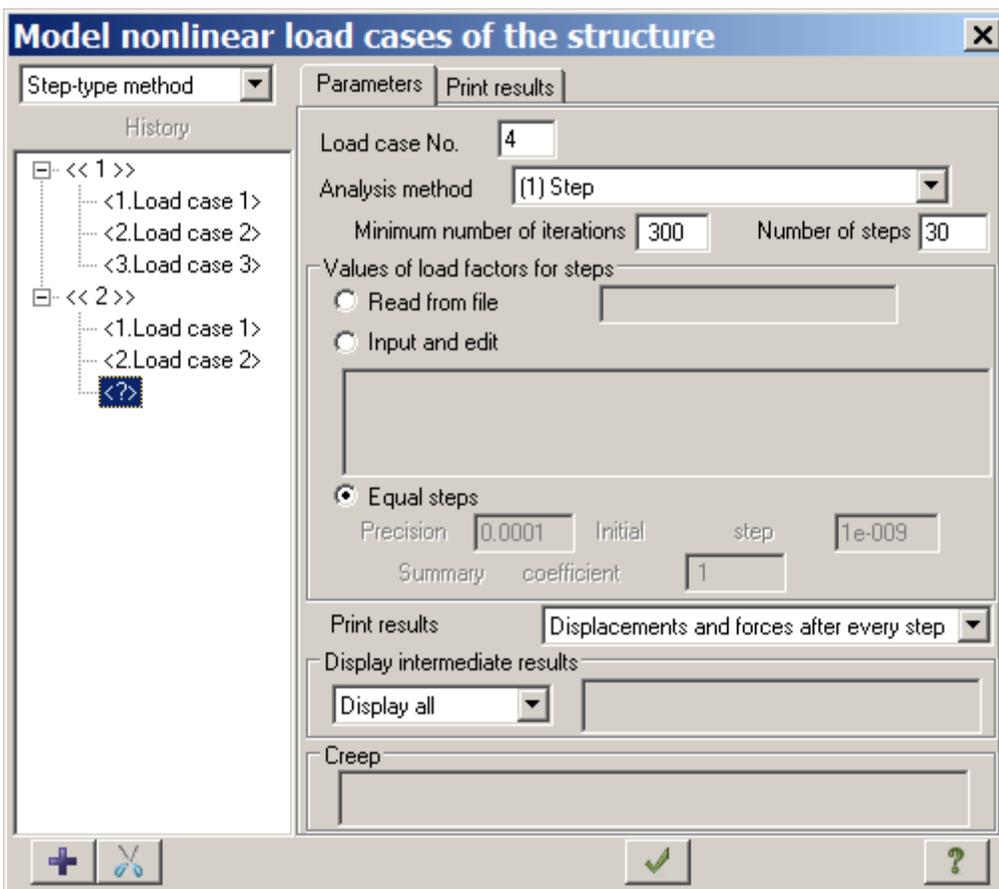


Figure 7.23 **Model nonlinear load cases of structure** dialog box

To take account of creep in concrete:

- ⇒ To take account of creep in concrete for the first sequence of load application, under **History**, select the first history of loads (see Figure 7.24).
- ⇒ Then in the **Creep** box, define with spaces number of days equal to 365 and 730 (after these number of days, creep in concrete will be considered in analysis).
- ⇒ To take account of creep in concrete for the second sequence of load application, under **History**, select the second history of loads.
- ⇒ Then in the **Creep** box, define with spaces number of days equal to 365 and 730 (after these number of days, creep in concrete will be considered in analysis).

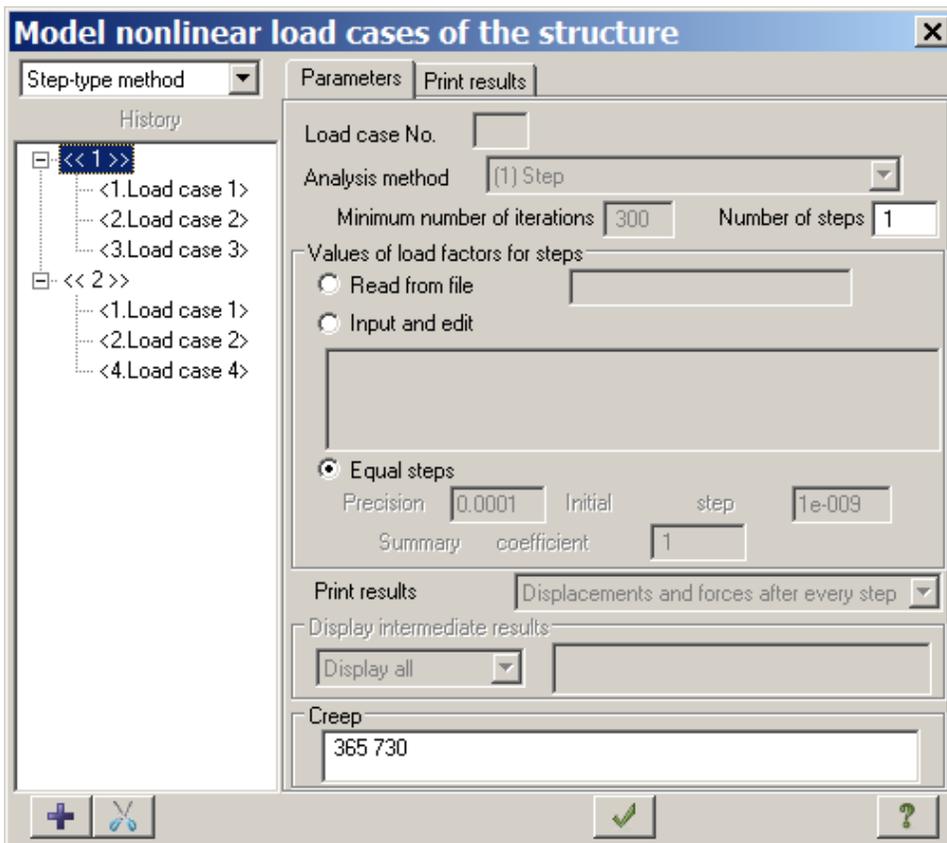


Figure 7.24 Model nonlinear load cases of structure dialog box

⇒ To input defined data, click **OK** .

Step 7. Physically nonlinear analysis of beam

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

Step 8. Review and evaluation of analysis results



*When analysis procedure is complete, to review and evaluate results of physically nonlinear analysis, select the **Results** and **Advanced results** ribbon tabs.*

⇒ In the mode of analysis results visualization, by default design model is presented with account of nodal displacements (see Fig.7.25). To display the model without nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button .

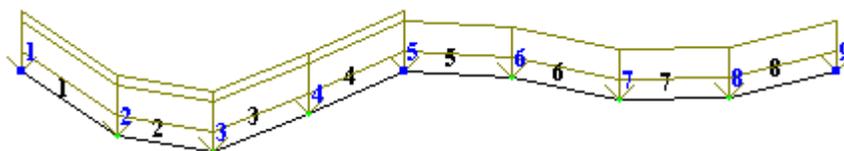


Figure 7.25 Design model with account of nodal displacements

To hide presentation of node numbers and of loads on design model:

- ⇒ On the **Select** toolbar, click **Flags of drawing** button .
- ⇒ In the **Display** dialog box, clear the **Node numbers** check box on the **Nodes** tab.
- ⇒ In the same dialog box, clear the **Loads** check box on the **General** tab.
- ⇒ Click **Redraw** .

To present diagrams of internal forces:

- ⇒ To display diagram **My** (see Fig.7.26), on the **Results** tab, select **Forces in bars** panel and click **Diagrams My** button .

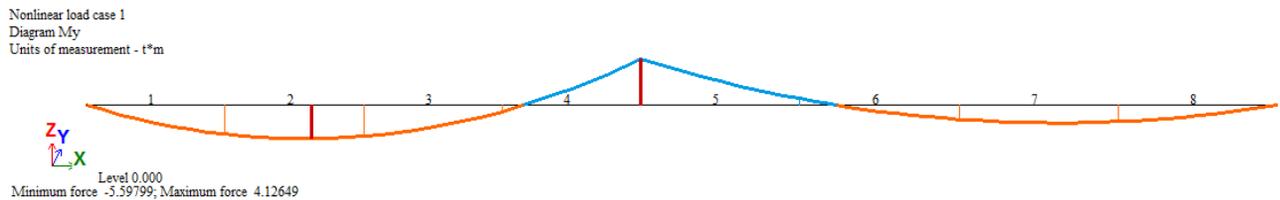


Figure 7.26 Diagram of bending moments **My**

- ⇒ To display diagram **Qz** (see Fig.7.27), on the **Results** tab, select **Forces in bars** panel and click **Diagrams Qz** button .

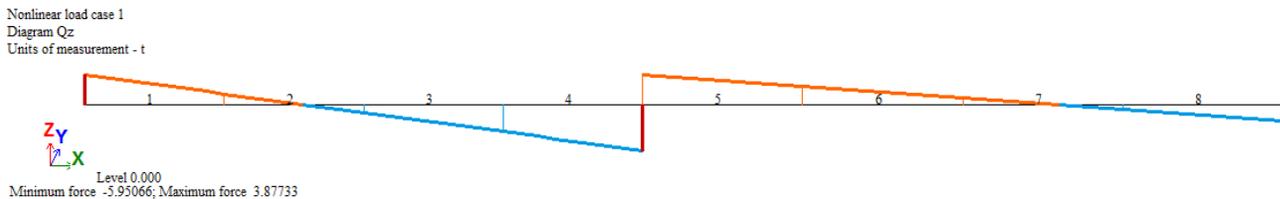


Figure 7.27 Shear force diagram **Qz**

- ⇒ To display mosaic plots **My**, 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. Then click  button.

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 change number of time interval for account of creep in concrete:

- ⇒ On the Status bar, in the **Mode shape No. (component, time interval)** list, select the row for the first time interval for account of creep in concrete (**1**) and click **Apply** .

To display intermediate results:

- ⇒ To display analysis results for 20% load application from the first load case, in the **Mode shape No. (component, time interval)** list, select the row **1 (20%)** and click **Apply**  .



To select the history of nonlinear load case automatically, click the **Apply current load case No.**

automatically button  .

To display analysis results for this load case, use the **Next** button at the **Change load case No.** list or at the **Mode shape No. (component, time interval)** list.

To display results by cracks in bar elements:

- ⇒ On the Status bar, in the **Mode shape No. (component, time interval)** list, select the row for zero time interval for account of creep in concrete (**0**).
- ⇒ To display analysis results by cracks in bar elements, on the **Advanced results** tab, on the **Fracture pattern** panel, select the **Cracks in bars** command  in the **Plates/Bars** drop-down list.
- ⇒ To display the depth of crack propagation, click the **Depth of crack propagation (Bars)** button  .
- ⇒ To display the width of crack propagation, click the **Width of crack propagation** button  .
- ⇒ To preview information about cracks in a certain element, on the **Select** toolbar, click the **Information about nodes and elements** button  and then specify with the pointer certain element, e.g. element No.4.
- ⇒ In the **Information about element** dialog box (see Fig. 7.28), to display parameters of section with cracks, select the **Cracks** check box.

Element 4

Nodes No.
4, 5

Eler 4 Block # 1 Selected

Stiffness type
2* T-section 30 X 60

FE type 210 Sect.number 2 Orthotropy

Length, centre of gravity coordinates
L=1.35m, Xc=4.725m, Yc=0m, Zc=0m

Load case Load. # 2
 DCL Time 0

Cracks
 Diagrams
 Export forces

N	0	t
Mx	0	t*m
My	-2.3627	t*m
Qz	-2.50713	t
Mz	0	t*m
Qy	0	t
Ry	0	t/m
Rz	0	t/m

Figure 7.28 Information about element No.4 dialog box

- ⇒ To display analysis results for the second section, in the **Show section** list, select number 2.
- ⇒ To modify the time interval for account of creep in concrete, use the **Time interval** list (after the second time interval you will see intermediate results).
- ⇒ To switch to another nonlinear load case, use the **Load case No.** list.

In the Fig.7.29 you will see intermediate results for the state of cracks for the first section of element No.8 of the 2nd nonlinear load case (the second sequence of load application) when 76.67% of load from the 4th load case is applied.

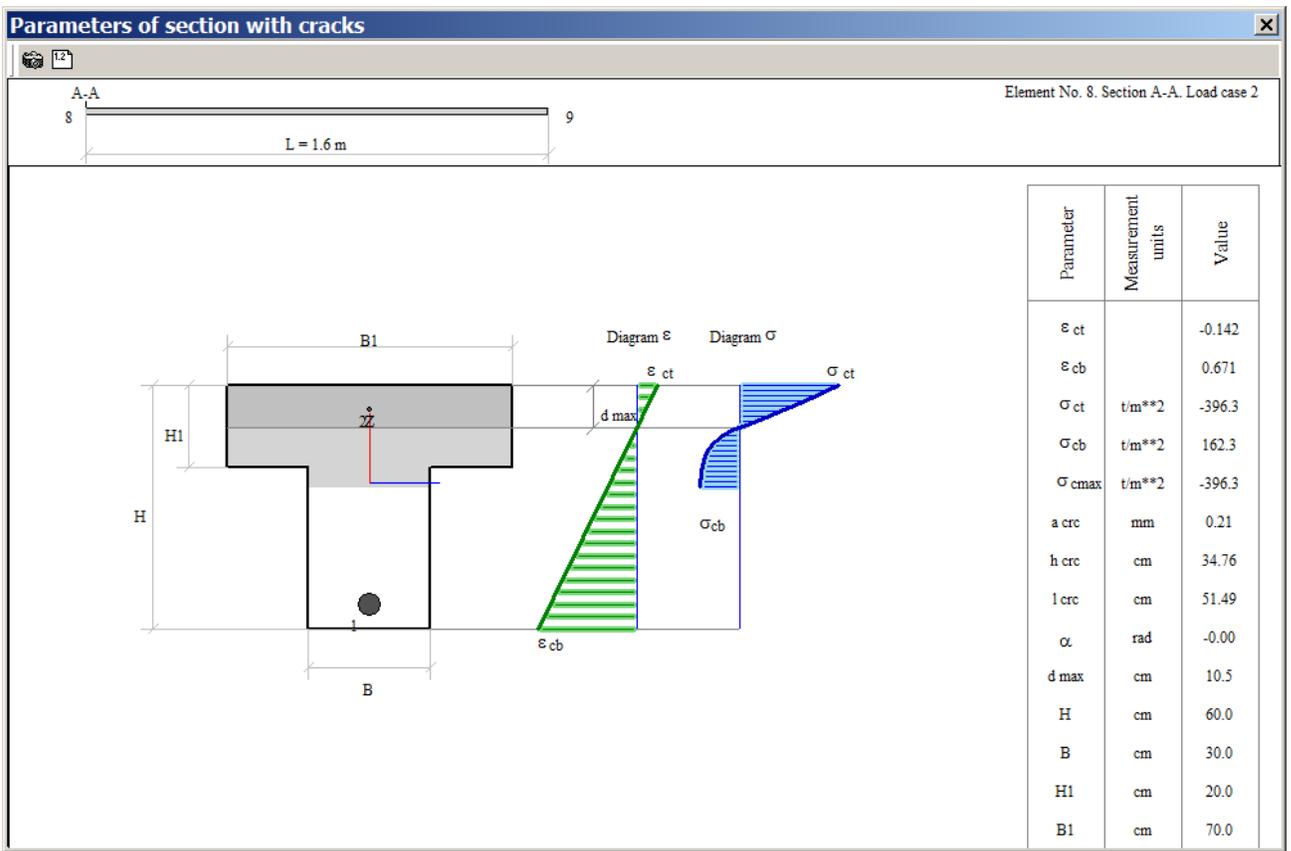


Figure 7.29 Parameters of section with cracks dialog box