

Example 16. Analysis on progressive collapse

In this lesson you will learn how to:

- carry out analysis of structure in progressive collapse.

Description:

4-span 8-storey building.

Span dimensions – 6 m, columns spacing – 5.6 m, height of storeys – 3 m.

Columns are fixed at foundation slab.

Material – reinforced concrete B30, reinforcement A-III.

Loads:

load case 1 – uniformly distributed load $p = 1 \text{ t/m}^2$ applied to all floor slabs and to roof slab;



concentrated load $P = 30 \text{ t}$ applied to the upper corner of collapsed column.



This example illustrates novel approach to this situation using unique options of LIRA-SAPR software. At the first stage, physically and geometrically nonlinear analysis is carried out on emergency loads. At the second stage, forces (with account of dynamic factors) from excluded elements are applied to modified design model (in the ASSEMBLAGE-plus system). Stress-strain state determined at the first stage of analysis is considered as initial state for the second stage. At the second stage of analysis, physically and geometrically nonlinear analyses are also carried out.

- ⇒ On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2016** and then click **LIRA-SAPR 2016**.

Step 1. Creating new problem

- ⇒ On the FILE menu, click **New** (button  on the toolbar).
- ⇒ In the **Model type** dialog box (see Fig.16.1) specify the following data:
- problem name – **Example16**;
 - model type – **5 – Six degrees of freedom per node** (translations X, Y, Z and rotations Ux, Uy, Uz).
- ⇒ Click **OK** .

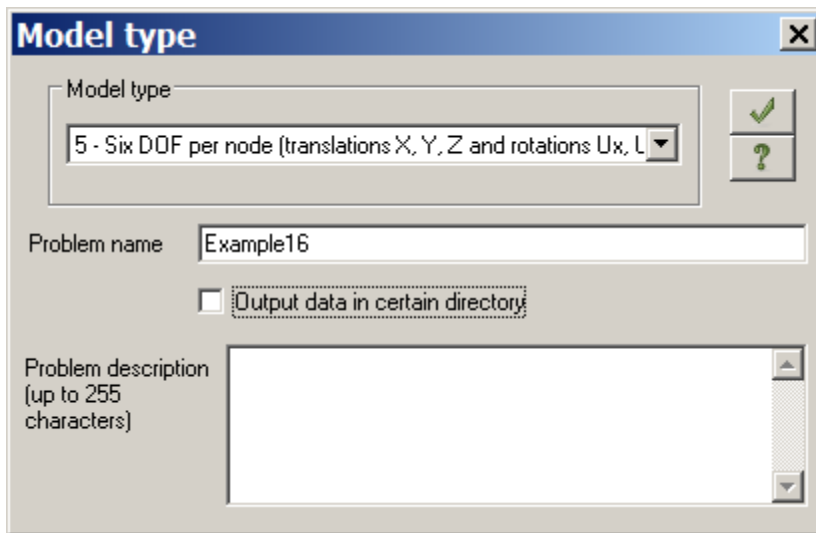




Figure 16.1 **Model type** dialog box



*It is also possible to open the **Model type** dialog box with a pre-defined type of model. To do this, on the **LIRA-SAPR menu** (Application menu), point to **New** and click **Model type 5 (Six DOF per node)***

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


*down menu select **Model type 5 (Six DOF per node)** command . Then you should define only problem name.*




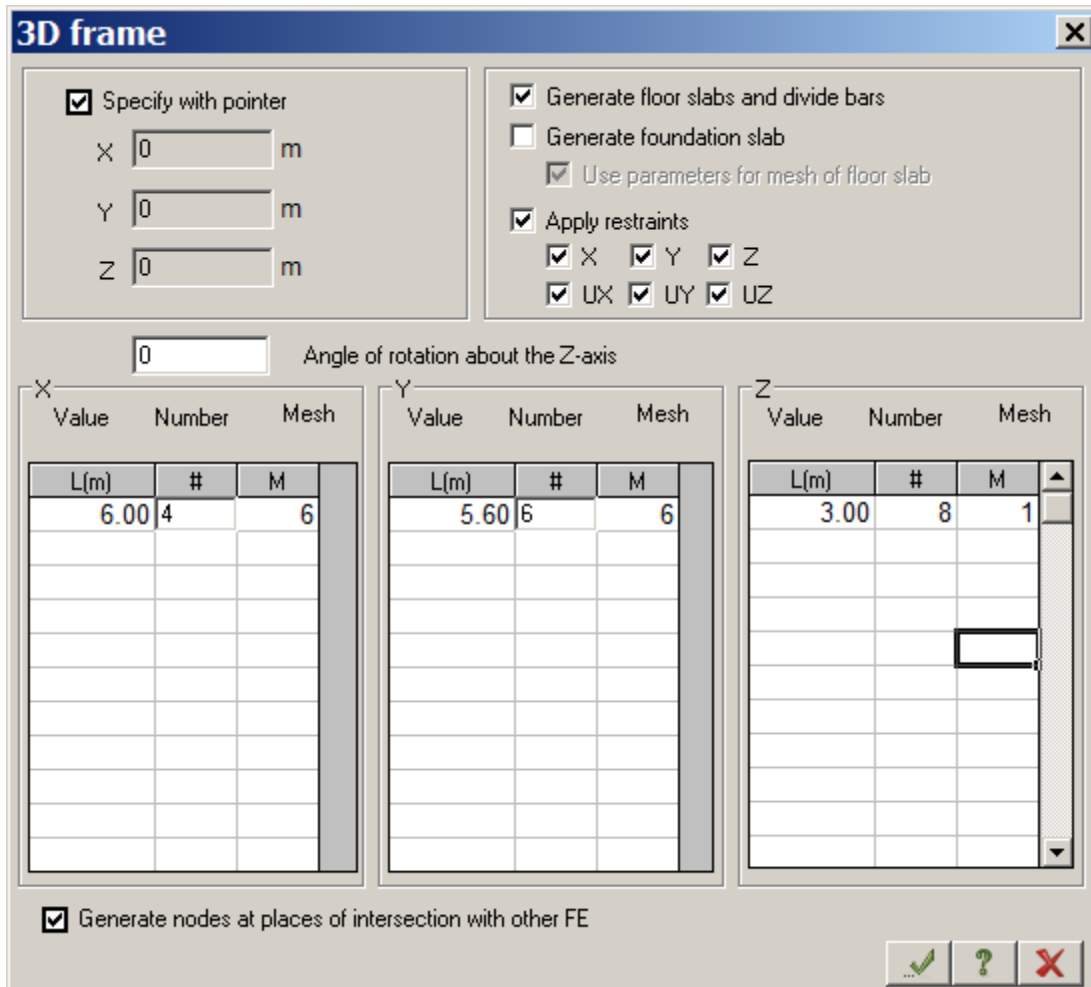
To save all output data files for the problem in certain directory, select appropriate check box. The directory name will coincide with the name of the problem. This directory will appear in the directory for files with analysis results. This is helpful if you have to find output data files for certain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

Step 2. Generating model geometry


To generate 3D frame:

- ⇒ On the **Create and edit** ribbon tab, on the **Create** panel, point to **Create regular fragments and grids** list and click the **3D frame**  command.

- ⇒ In the **Create plane fragments and grids** dialog box (see Fig.16.2), clear the **Generate foundation slab** check box.
- ⇒ Then specify the following data for 3D frame:
- | | | |
|--------------------|------------------|------------------|
| ▪ spacing along X: | spacing along Y: | spacing along Z: |
| L(m) N M | L(m) N M | L(m) N M |
| 6 4 6 | 5.6 6 6 | 3 8 1 |
- other parameters remain by default.
- ⇒ Click **Apply** .

Figure 16.2 **3D frame** dialog box

To delete elements of beams:



- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **Select horizontal elements**  .
- ⇒ Then 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.

- ⇒ To delete selected elements, on the **Create and edit** ribbon tab, on the **Edit** panel, click **Delete selected objects** .

To pack the model:

- ⇒ On the **Create and edit** ribbon tab, on the **Edit** panel, click **Pack model** .
- ⇒ In the **Pack model** dialog box (see Fig.16.3), click **Apply** . It is necessary to pack the model in order to 'throw together' coincident nodes and elements and to eliminate (that is, to remove completely) deleted nodes and elements from design model.

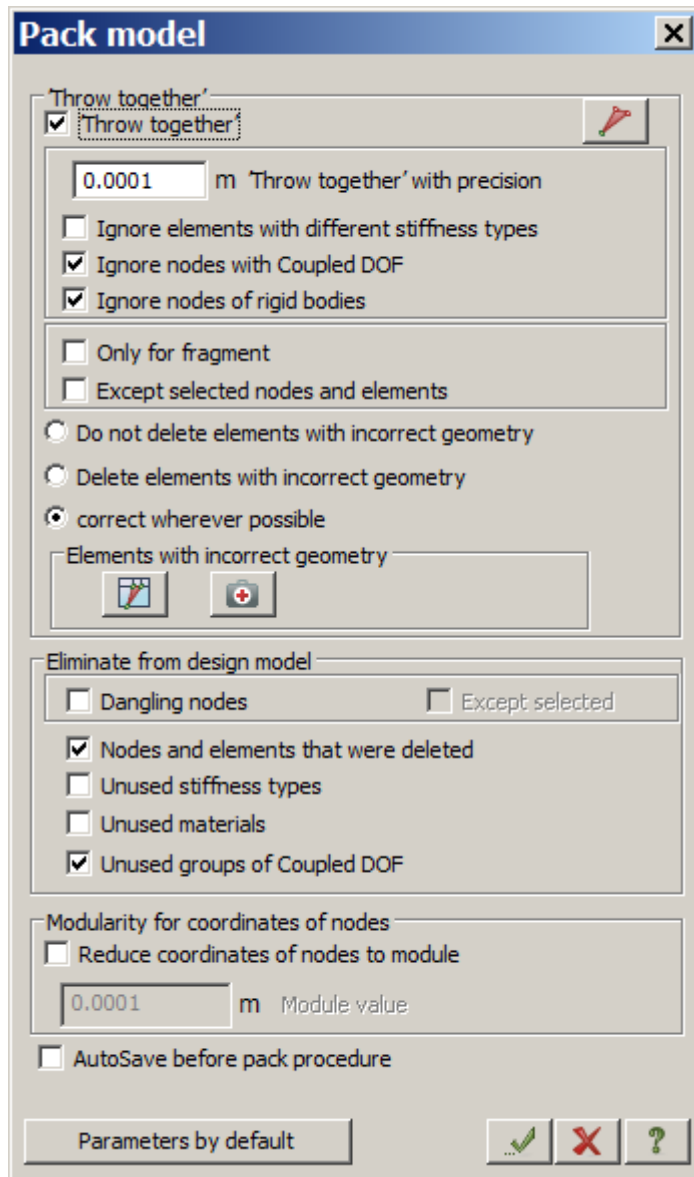


Figure 16.3 **Pack model** dialog box

The model shown in Fig.16.4 will be displayed on the screen.

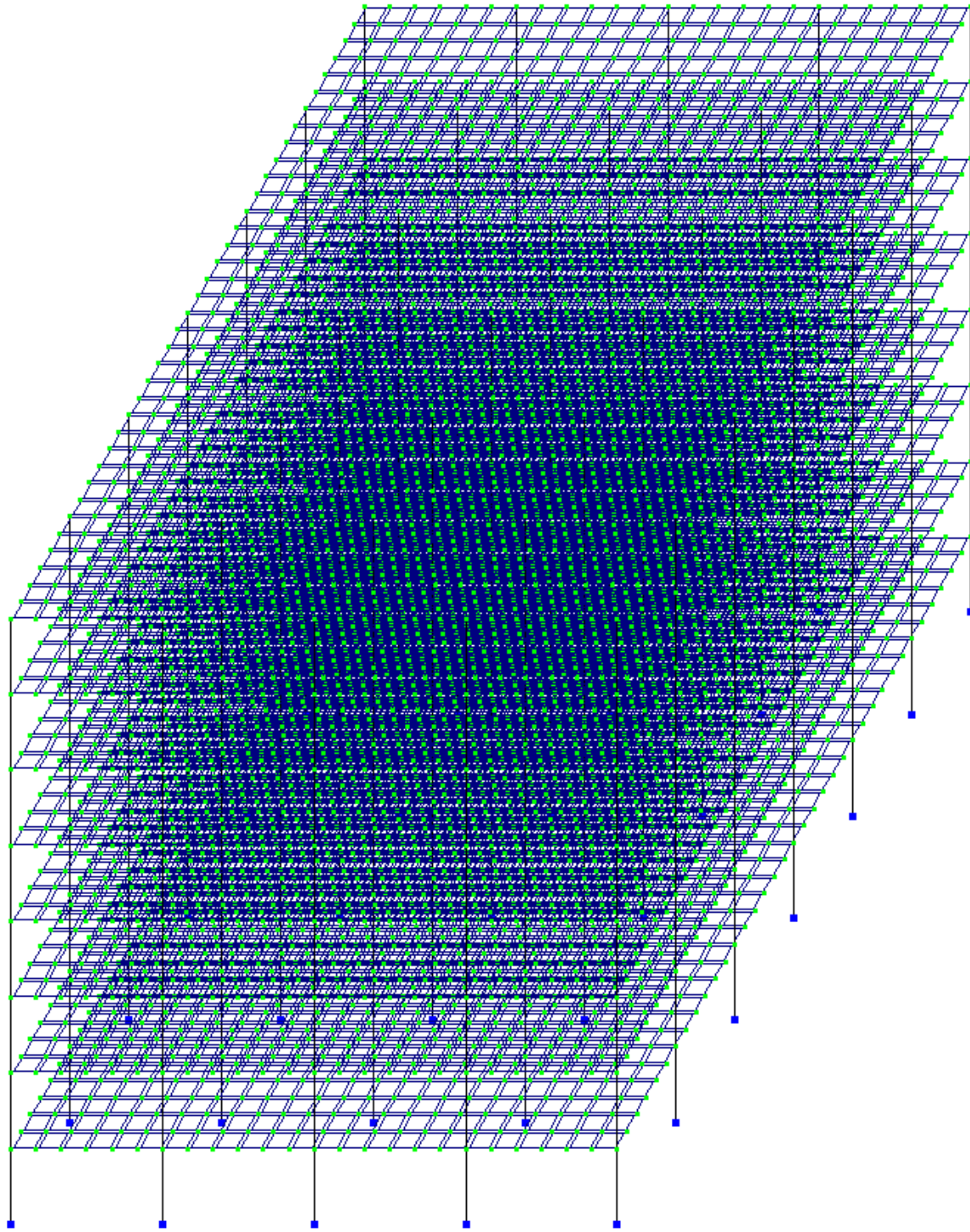



Figure 16.4 Design model of the framework


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 – **Example16**;

- location where you want to save this file (**Data** folder is displayed by default).
- ⇒ Click **Save**.

Step 3. 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.16.5a), click **Add**. The list of standard section types will be presented in the **Add stiffness** dialog box (see Fig.16.5b).

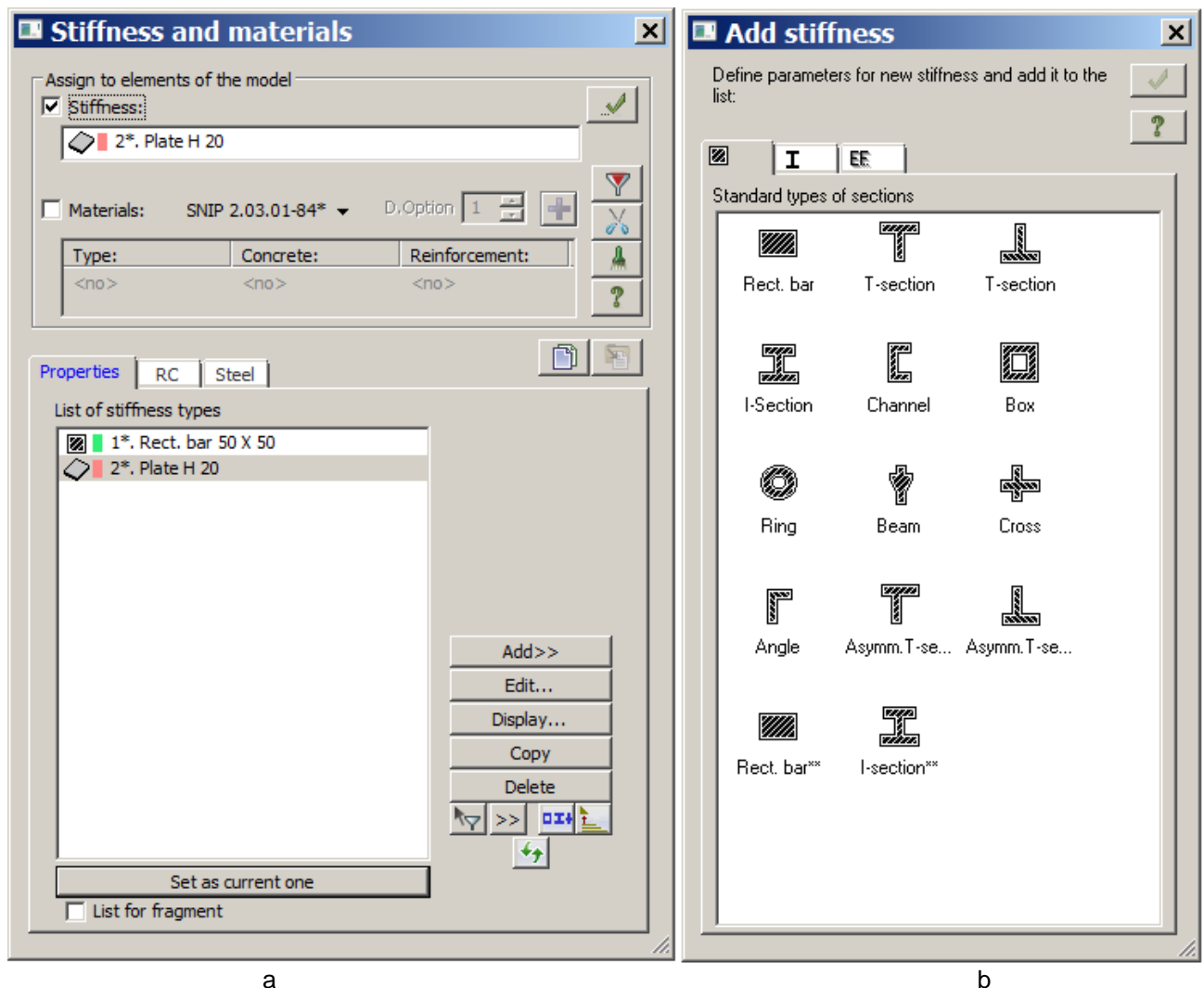


Figure 16.5 Dialog boxes: a – **Stiffness and materials**, b – **Add stiffness**

- ⇒ 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.16.6):
 - geometric properties – B = 50 cm; H = 50 cm.

⇒ Select the **Nonlinear parameters** check box.

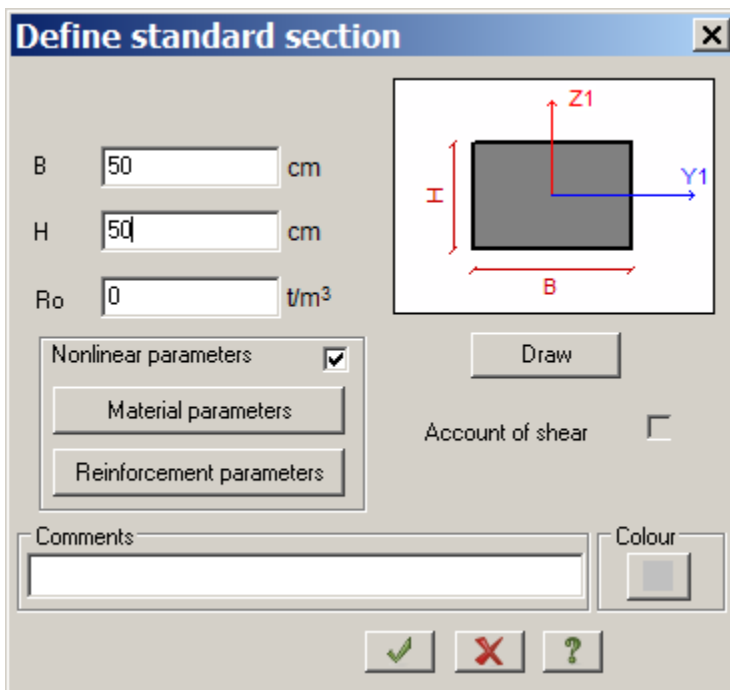


Figure 16.6 **Define standard section** dialog box

- ⇒ To define material, click **Material parameters**. The **Nonlinear stress-strain diagrams for materials** dialog box appears on the screen (see Fig.16.7).
- ⇒ In this dialog box for the main material select **31 – exponential (design 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 – B30;
 - concrete type – TA.
- ⇒ To preview schematic presentation, click **Draw**.

Nonlinear stress-strain diagrams for materials

☐ Account of reinforcement
☐ Account of creep in concrete

Main material

Nonlinear stress-strain diagram

31 - exponential (design strength)

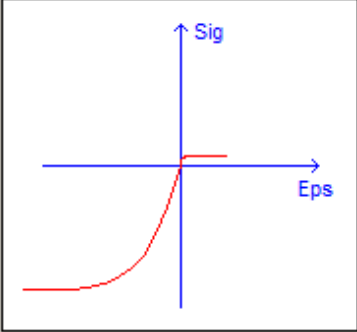
Entry No. 1 New Copy Delete

Comments

Parameters for stress-strain diagram

Parameters	Values
Concr. name	B30
Concr. type	HB
Eo	2960000 t/m ²
$\sigma(-)$	1730 t/m ²
$\sigma(+)$	122 t/m ²
$\epsilon(-)$	
$\epsilon(+)$	
K	

Draw



☐ Criteria of rupture (for FE of plates)

Save diagram to file

☒ Current diagram ☐ All diagrams of problem

OK Cancel Help

Figure 16.7 Nonlinear stress-strain diagrams for materials dialog box for main material

- ⇒ In the same dialog box, click **Account of reinforcement** (see Fig.16.8) 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 $s(-) = -37500 \text{ t/m}^2$;
 - ultimate stress $s(+) = 37500 \text{ t/m}^2$.
- ⇒ To preview graphical presentation of the diagram, click **Draw**.
- ⇒ To confirm the specified data, click **OK**.

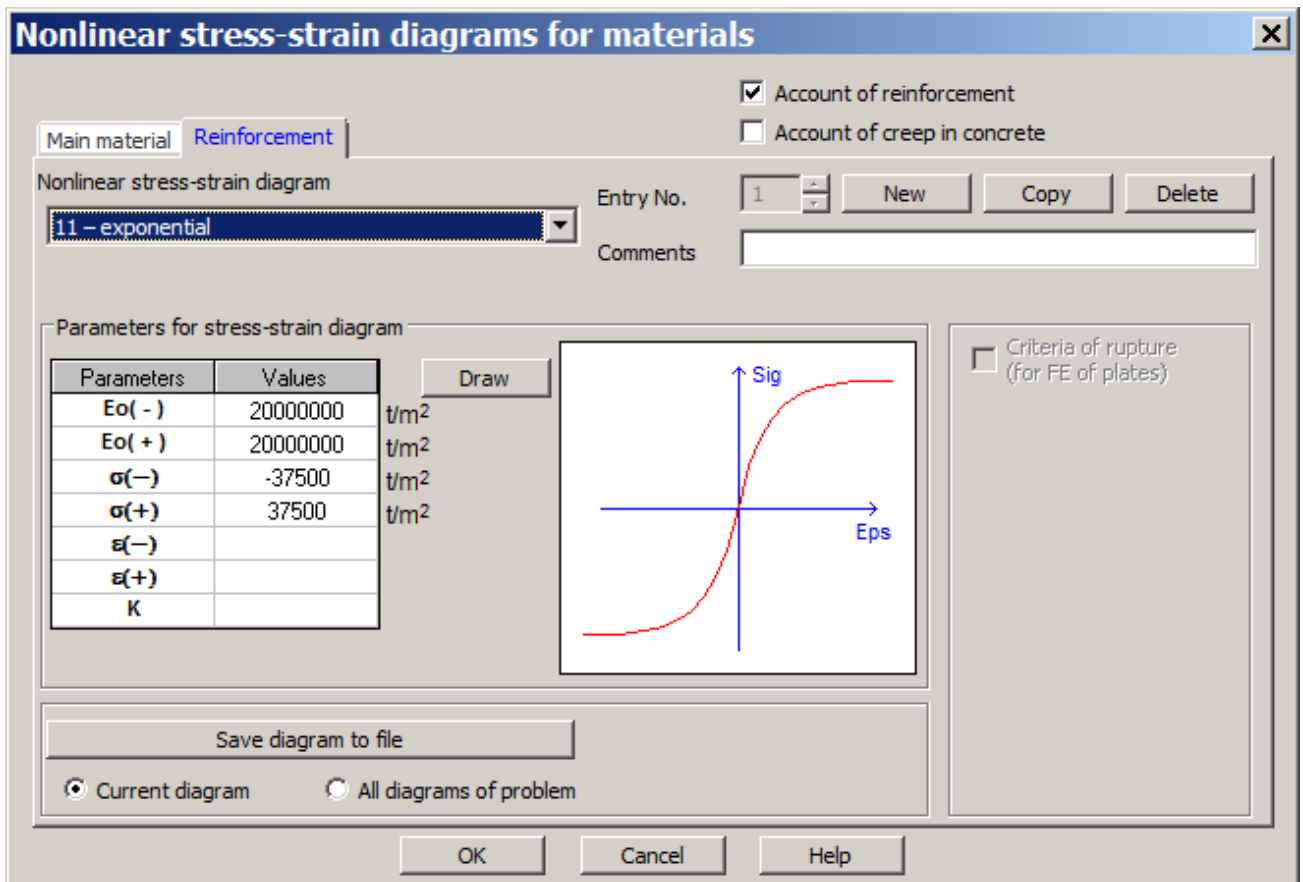


Figure 16.8 Nonlinear stress-strain diagrams for materials dialog box for reinforcement

⇒ To define location and area of reinforcement, in the **Define standard section** dialog box (see Fig.16.6), click **Reinforcement parameters**.

⇒ In the **Parameters of physical nonlinearity for bars** dialog box (see Fig.16.9), click the **Point**



⇒ Define parameters for the first layer of reinforcement:

- area of reinforcement – $F_a = 10 \text{ cm}^2$;
- distances from the Z-axis (Y_i) and from the Y-axis (Z_i) to the point – $y = -20 \text{ cm}$; $z = 5 \text{ cm}$.

⇒ Under the **Type of reinforcement**, in the **Number of reinforcement layer** list box select number 2.

⇒ Define parameters for the second layer of reinforcement:

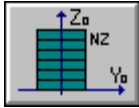
- area of reinforcement – $F_a = 10 \text{ cm}^2$;
- distances from the Z-axis (Y_i) and from the Y-axis (Z_i) to the point – $y = 20 \text{ cm}$; $z = 5 \text{ cm}$.

⇒ Under the **Type of reinforcement**, in the **Number of reinforcement layer** list box select number 3.

⇒ Define parameters for the third layer of reinforcement:

- area of reinforcement – $F_a = 10 \text{ cm}^2$;
- distances from the Z-axis (Y_i) and from the Y-axis (Z_i) to the point – $y = -20 \text{ cm}$; $z = 45 \text{ cm}$.

- ⇒ Under the **Type of reinforcement**, in the **Number of reinforcement layer** list box select number 4.
- ⇒ Define parameters for the fourth layer of reinforcement:
 - area of reinforcement – $F_a = 10 \text{ cm}^2$;
 - distances from the Z-axis (Y_i) and from the Y-axis (Z_i) to the point – $y = 20 \text{ cm}$; $z = 45 \text{ cm}$.
- ⇒ Under the **Type of cross-section division**, click **Division of cross-section into elementary strips** icon



- ⇒ To preview the section, click **Draw**.
- ⇒ Click **OK**.

Parameters of physical nonlinearity for bars

Type of reinforcement

Numb. of reforc. layer: 4

F_a 10 cm^2 y 20 cm

z 45 cm

Point reinforcement


Type of cross-section division

Division into elementary rectangles

NZ 5 NY 5

OK Draw Cancel Help

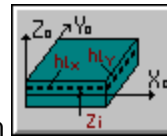
Figure 16.9 Parameters of physical nonlinearity for bars dialog box

- ⇒ In the **Define standard section** dialog box, click **OK** .
- ⇒ In the **Add stiffness** dialog box, click the tab with numerical description of stiffness (the third tab).
- ⇒ Double-click the **Plates** icon in the list. The **Specify stiffness for plates** dialog box opens. In this dialog box you can define material properties for selected type of the section.

- ⇒ In the **Specify stiffness for plates** dialog box (see Fig.16.10), specify the following parameters for **Plates** (for floor slab):
 - Poisson's ratio – $V = 0.2$;
 - thickness – $H = 20$ cm.
- ⇒ Select the **Nonlinear parameters** check box.

Figure 16.10 **Specify stiffness for plates** dialog box

- ⇒ To define material, click **Material parameters**. The **Nonlinear stress-strain diagrams for materials** dialog box appears on the screen.
- ⇒ In this dialog box for the main material select **31 – exponential (design strength)** in the **Nonlinear stress-strain diagram** list box.
- ⇒ Under **Parameters for stress-strain diagram**, you will see parameters for main material (concrete) defined for the previous stiffness.
- ⇒ In the same dialog box, click **Account of reinforcement** and select the **Reinforcement** tab.
- ⇒ In the **Nonlinear stress-strain diagram** list box, select **11 – exponential**.
- ⇒ Under **Parameters for stress-strain diagram**, you parameters for reinforcement defined for the previous stiffness will be displayed.
- ⇒ To confirm the specified data, click **OK**.
- ⇒ To define location and area of reinforcement, in the **Stiffness for plates** dialog box (see Fig.16.10), click **Reinforcement parameters**.
- ⇒ The **Type of reinforcement** dialog box appears on the screen (see Fig.16.11).



(physical equivalent of mesh).

- ⇒ In this dialog box click the **Bar-type reinforcement** icon
- ⇒ Define parameters for the first layer of reinforcement:
 - equivalent thickness of rebars in mesh along the Y-axis – $H_y = 0.1$ cm;
 - equivalent thickness of rebars in mesh along the X-axis – $H_x = 0.1$ cm;
 - distance from the mesh to the middle surface – $z = -6$ cm.
- ⇒ By default, number of rebars per 1 running metre (r.m.) is equal to 5.
- ⇒ In the same dialog box, in the **Number of reinforcement layer** list box select number 2.
- ⇒ Define parameters for the second layer of reinforcement:
 - equivalent thickness of rebars in mesh along the Y-axis – $H_y = 0.2$ cm;
 - equivalent thickness of rebars in mesh along the X-axis – $H_x = 0.2$ cm;
 - distance from the mesh to the middle surface – $z = 6$ cm.
- ⇒ By default, number of rebars per 1 running metre (r.m.) is equal to 5.
- ⇒ To preview the section, click **Draw**.
- ⇒ Click **OK**.

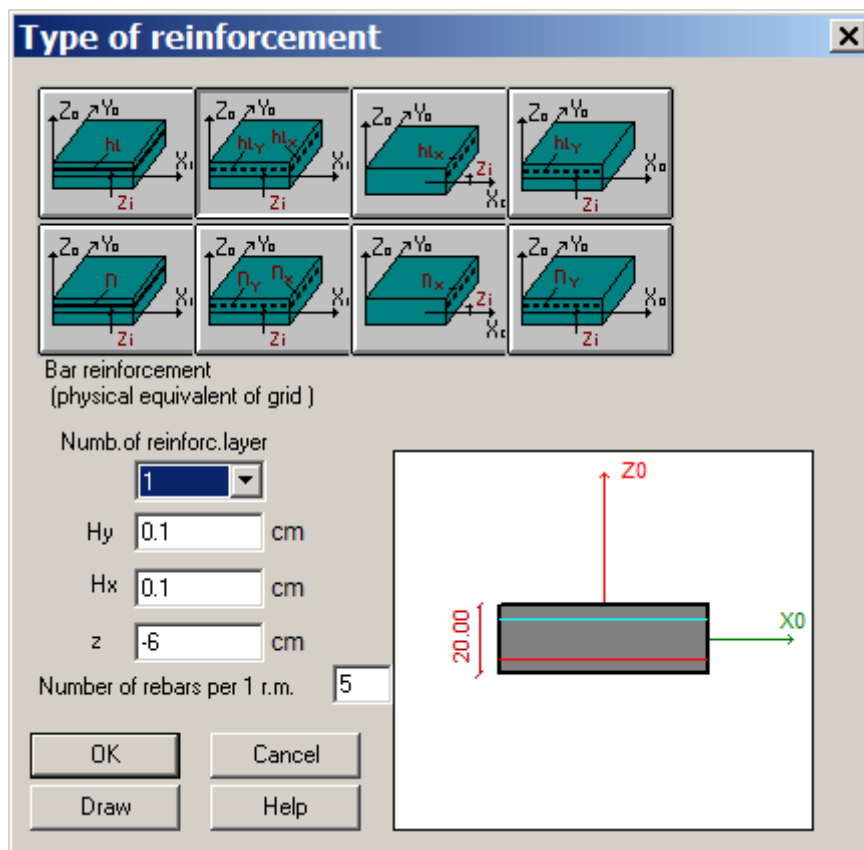





Figure 16.11 The **Type of reinforcement** dialog box

- ⇒ In the **Define standard section** dialog box, click **OK** .
- ⇒ To hide library of stiffness properties, in the **Stiffness of elements** dialog box, click **Add** unfold button.

To change FE type:

- ⇒ On the **Select** toolbar (by default, it is displayed at the bottom of the screen), click **Select elements** (button  on the toolbar).
- ⇒ Select all 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 Fig.16.12), in the **list of FE types**, select **FE type 241 – physically nonlinear arbitrary rectangular FE of shell**.
- ⇒ Click **Apply** .

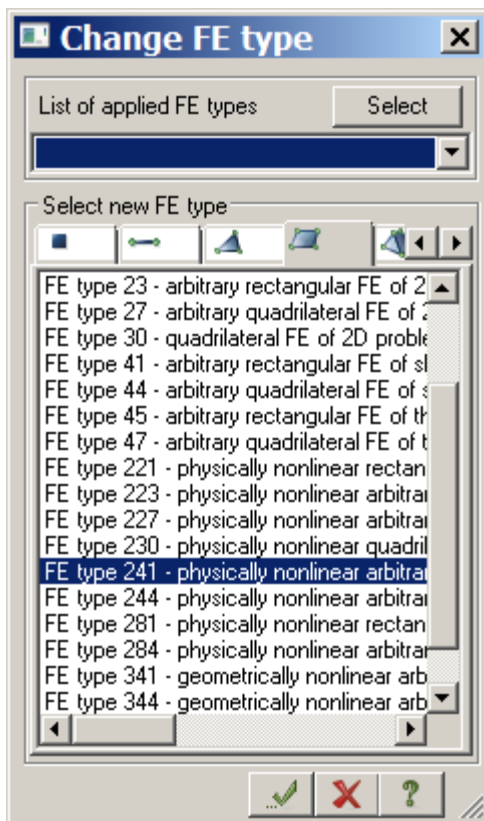


Figure 16.12 **Change FE type** dialog box

- ⇒ The **Warning** box is displayed (see Fig.16.13). Click **OK**.

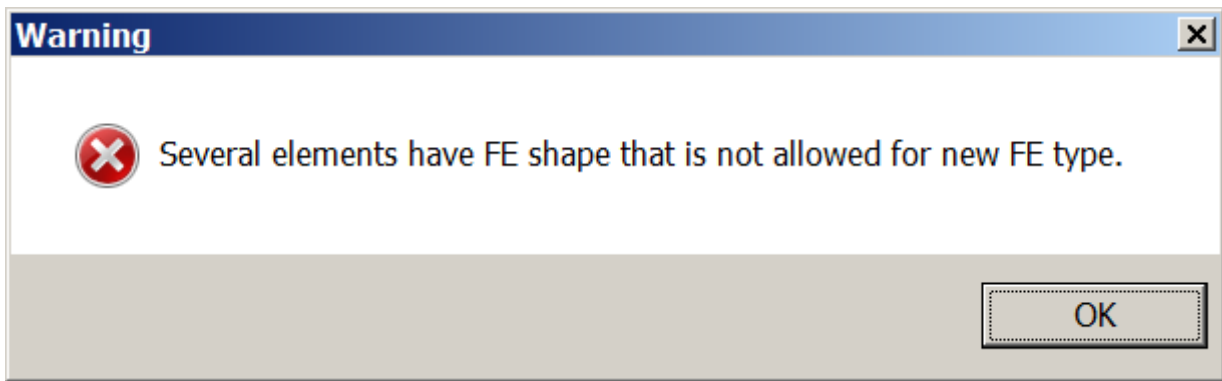






Figure 16.13 Warning box

- ⇒ In the **Change FE type** dialog box, click the second tab (2-node FE) and in the **list of FE types**, select **FE type 210 – physically nonlinear arbitrary 3D bar**.
- ⇒ Click **Apply** .


To assign stiffness to elements of the model:




- ⇒ 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.16.5a), in the **List of stiffness types**, select stiffness type '2*.Plate H20'.
- ⇒ 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 all elements of the model.
- ⇒ In the **Stiffness and materials** dialog box, click **Apply** .
- ⇒ The **Warning** box is displayed. Click **OK**.
- ⇒ In the **Stiffness of elements** dialog box, in the **List of stiffness types**, select '1*.Rect. bar 50x50'.
- ⇒ Click **Set as current type**.
- ⇒ In the **Stiffness of elements** dialog box, click **Apply**.
- ⇒ On the **Select** toolbar, click **Select elements** button  once again in order to make this command not active.

Step 4. Defining boundary conditions



Analysis on progressive collapse will be carried out for the example when one of columns of the first floor is collapsed. To avoid geometrically unstable columns (above the first floor) about the Z-axis, additional boundary conditions are imposed on all nodes of floor slab of the first floor.

- ⇒ 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, point to **Select nodes** drop-down list and click **Select nodes** button . With the pointer select all nodes of the floor slab of the first floor.
- ⇒ 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.16.14), specify directions along which displacements of nodes are not allowed (UZ). To do this, select appropriate check boxes.
- ⇒ Click **Add restraints at selected nodes**  button.

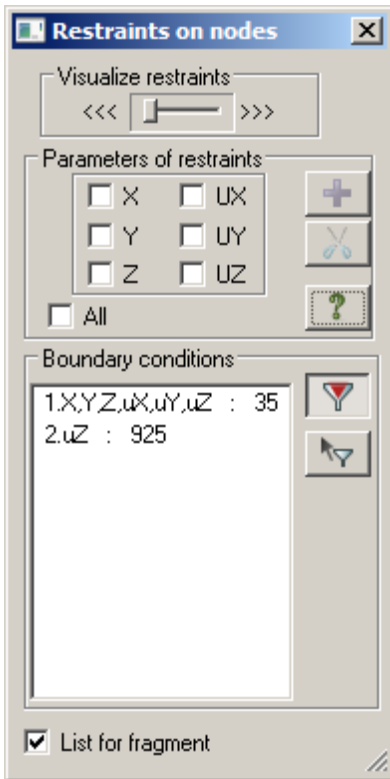


Figure 16.14 **Restraints on nodes** dialog box

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

Step 5. Applying loads



Analysis on progressive collapse is carried out with normative loads.

For the second load case where progressive collapse (destruction of column) is simulated, it is possible to define dynamic factor equal to 1.1. To do this, at the upper corner of the column, define 10% of the internal force in this column from the combination of previous load cases.

To create load case No.1:

- ⇒ To select floor slabs and roof slab, in the **PolyFilter** dialog box (see Fig.16.15), on the **Filter for elements** tab, select **By orientation of FE** check box and specify || XOY.
- ⇒ Click **Apply** .

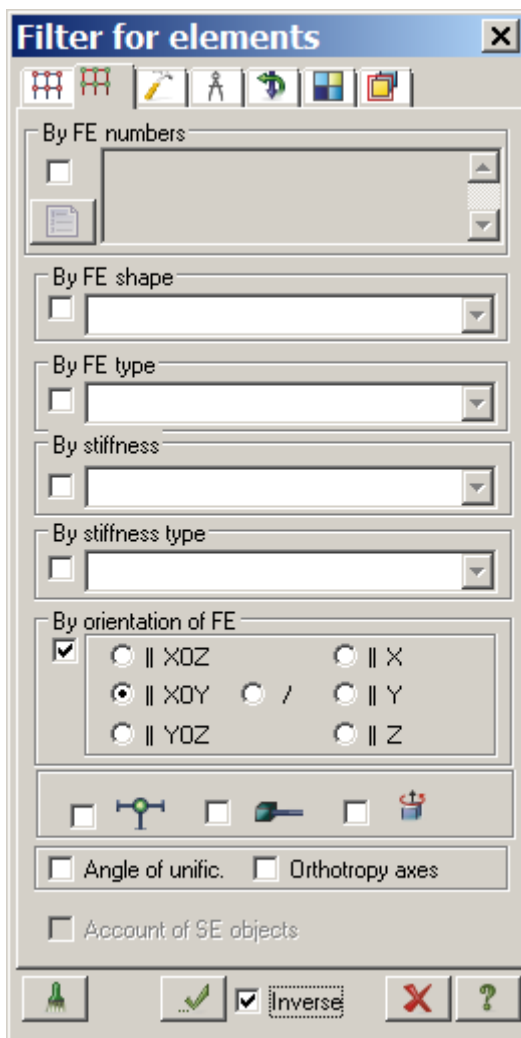



Figure 16.15 Filter for elements tab

- ⇒ 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.16.16), specify **Global** coordinate system and direction along the **Z-axis** (default parameters).

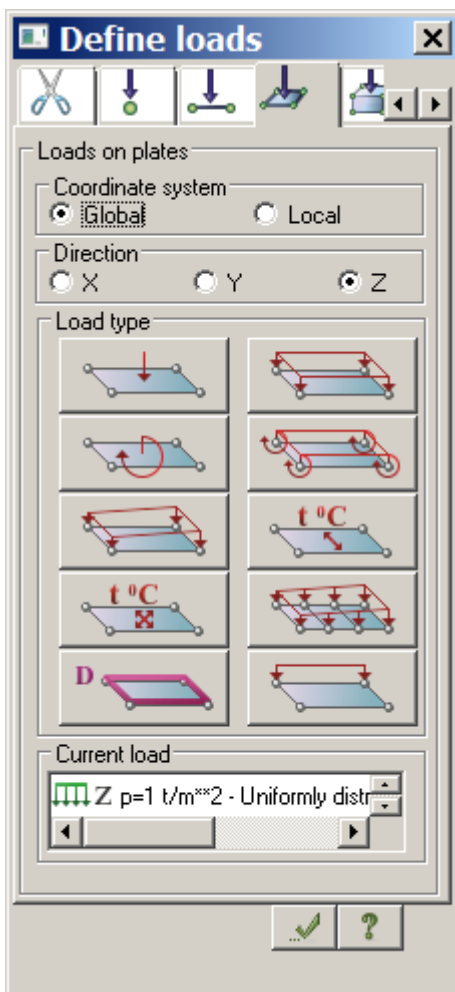
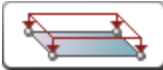



Figure 16.16 Define loads dialog box

- ⇒ In the **Load type** area, click **Uniformly distributed load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 1 \text{ t/m}^2$ (see Fig.16.17).
- ⇒ Click **OK** .

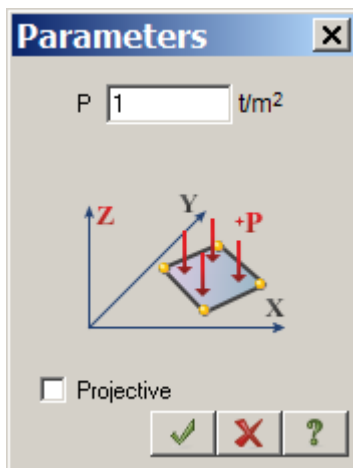













Figure 16.17 Load parameters dialog box

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.
- ⇒ In the **PolyFilter** dialog box (see Figure 16.15), on the **Filter for elements** tab, clear **By orientation of FE** check box.
- ⇒ Select **By FE numbers** check box and specify the element number '7'.
- ⇒ Click **Apply** .
- ⇒ To present on the screen only selected node of the model, on the **Select** toolbar, click **Fragmentation** .
- ⇒ Select upper node of column.
- ⇒ In the **Define loads** dialog box (see Fig.16.16), click the **Load on nodes**  tab.
- ⇒ In the **Load type** area, click **Concentrated load** button .
- ⇒ In the **Load parameters** dialog box specify $P = 30 \text{ t}$.
- ⇒ Click **OK** .
- ⇒ To restore design model in initial view after fragmentation, on the **Select** toolbar, click **Restore model** .
- ⇒ On the **Select** toolbar, click **Select nodes** button  once again in order to make this command not active.

Step 6. Modelling stages of collapse and nonlinear load cases

To model stages of collapse:

- ⇒ On the **Analysis** ribbon tab, on the **Nonlinearity** panel, click **Assemblage** (button  on the toolbar).
- ⇒ In the **Model nonlinear load cases of structure** dialog box (see Fig.16.18), select the third tab **Stages** and click the **Add** button  (in the left part of the dialog box, under **History**, the first load history is added and the row with first stage of assemblage will become selected automatically).
- ⇒ On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button .
- ⇒ Select all elements of the model with the pointer.
- ⇒ When elements are selected, in the **Model nonlinear load cases of structure** dialog box, in the **Elements to be assembled** area, click **All selected**. Numbers of elements selected on the model will be automatically displayed in the list.

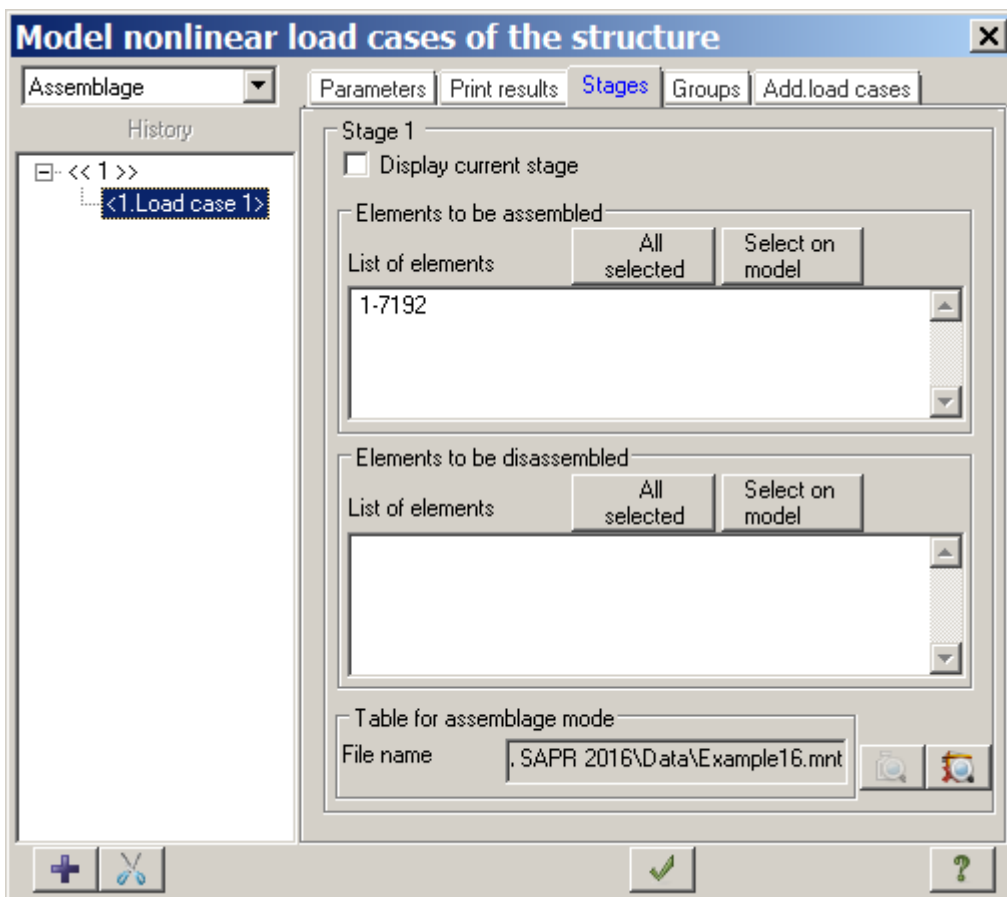







Figure 16.18 **Model nonlinear load cases** dialog box for defining assemblage stages

- ⇒ To unselect elements, on the **Select** toolbar, click **Unselect all** button .
- ⇒ In the **Model nonlinear load cases of structure** dialog box, to create the second stage of assemblage, click the **Add** button .
- ⇒ In the **PolyFilter** dialog box, select the **Filter for elements** tab (the second tab).
- ⇒ Select **By FE numbers** check box and specify number '7'.
- ⇒ Click **Apply** .
- ⇒ In the **Model nonlinear load cases of structure** dialog box, in the **Elements to be disassembled** area, click **All selected**.
- ⇒ To unselect elements, on the **Select** toolbar, click **Unselect all** button .

To model nonlinear load cases:

- ⇒ In the **Model nonlinear load cases of structure** dialog box, select the first tab **Parameters** and select the row that corresponds to the first stage of assemblage.
- ⇒ For the first load case define the following parameters (see Fig.16.19):
 - select **Step (1)** in the **Analysis method** list box;
 - select **Equal steps** option and define number of steps **10**;
 - select **Displacement and forces after every step** in the **Print results** list box;

- in the **Display intermediate results** list, select **Display all** option.
- ⇒ Select the row for the second load case and define the following parameters:
- select **Step (1)** in the **Analysis method** list box;
 - select **Equal steps** option and define number of steps **10**;
 - select **Displacement and forces after every step** in the **Print results** list box;
 - in the **Display intermediate results** list, select **Display all** option.
- ⇒ To input defined data, click **OK** .

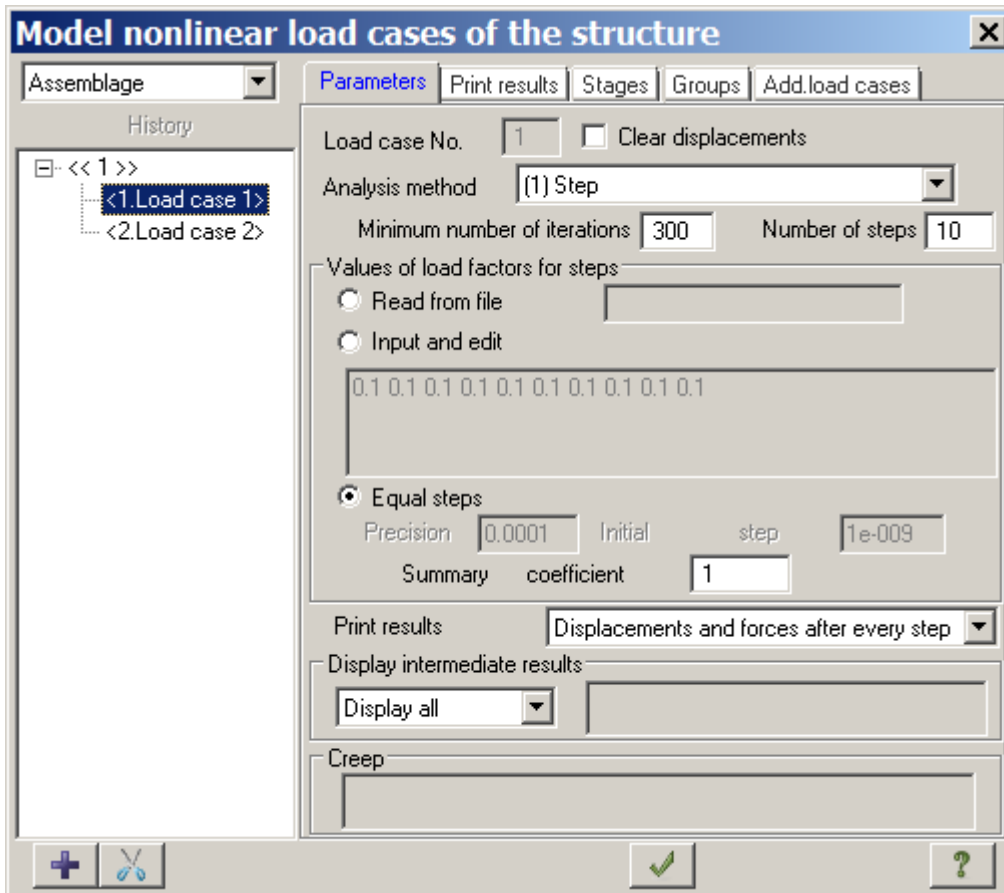



Figure 16.19 **Model nonlinear load cases** dialog box for defining parameters of nonlinear load cases

- ⇒ Close the **PolyFilter** dialog box.

Step 7. Nonlinear analysis of the model

- ⇒ 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 8. Review and evaluation of analysis results



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

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

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** .

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** (that corresponds to the fifth stage of assemblage) and click **Apply** .


To present displacement contour plots in the floor slab of the first floor:

- ⇒ 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.
- ⇒ On the **Projection** toolbar (by default, it is displayed at the bottom of the screen), click **Projection on XOZ-plane** .
- ⇒ On the **Select** toolbar, point to **Select elements** drop-down list and click **Select elements** button . Then select elements of the floor slab of the 1st storey.
- ⇒ To present on the screen only selected elements of the model, on the **Select** toolbar, click **Fragmentation** .
- ⇒ To present the model in dimetric projection, on the **Projection** toolbar, click **Dimetric projection** .


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 display intermediate results:








- ⇒ To display analysis results for 50% load application from the second load case, in the **Mode shape No. (component, time interval)** list, select the row **50%** 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 elements of the floor slab:

- ⇒ On the Status bar, in the **Mode shape No. (component, time interval)** list, select the row that corresponds to application of 100% of load (**100%**).
- ⇒ To display analysis results by cracks in plate elements, on the **Advanced results** tab, on the **Fracture pattern** panel, select the **Cracks in plates** 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 display data on crack propagation at the bottom edge of plate elements, on the **Advanced results** tab, on the **Fracture pattern** panel, select the **Cracks in bottom layer**  command.
- ⇒ To display direction of crack propagation in plate elements, on the **Advanced results** tab, on the **Fracture pattern** panel, select the **Direction of cracks (plates)** command .
- ⇒ To display data on crack propagation at the top edge of plate elements, on the **Advanced results** tab, on the **Fracture pattern** panel, select the **Cracks in top layer**  command.
- ⇒ 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.42.
- ⇒ In the **Information about element** dialog box (see Fig. 16.20), to display parameters of section with cracks, select the **Cracks** check box.

Element 42 [X]

Nodes No.
42, 43, 67, 68

Eler 42 Block # 1 ☐ Selected

Stiffness type
2*. Plate H 20

FE type 241 Unif. angle 0 Orthotropy 0

Area, centre of gravity coordinates
S=0.933333m², Xc=6.5m, Yc=0.466667m, Zc=3

Load cases Load. # 2 Result 100%

☐ ☐ ☐ ☐

Nx	3.99787	t/m**2
Ny	8.47909	t/m**2
Txy	4.17734	t/m**2
Mx	-3.22985	(t*m)/m
My	-5.65933	(t*m)/m
Mxy	0.756459	(t*m)/m
Qx	13.8606	t/m
Qy	13.3891	t/m

Show sect. 1

☐ Diagrams ☐ Cracks

Figure 16.20 Information about element No.42 dialog box

- ⇒ To display intermediate results, use the **Results** box.
- ⇒ To switch to another nonlinear load case, use the **Load case No.** list.

In the Fig.16.21 you will see intermediate results for the state of section with cracks for element No.42 of the 2nd nonlinear load case (the second stage of assemblage) when 70% of load from the 2nd load case is applied.

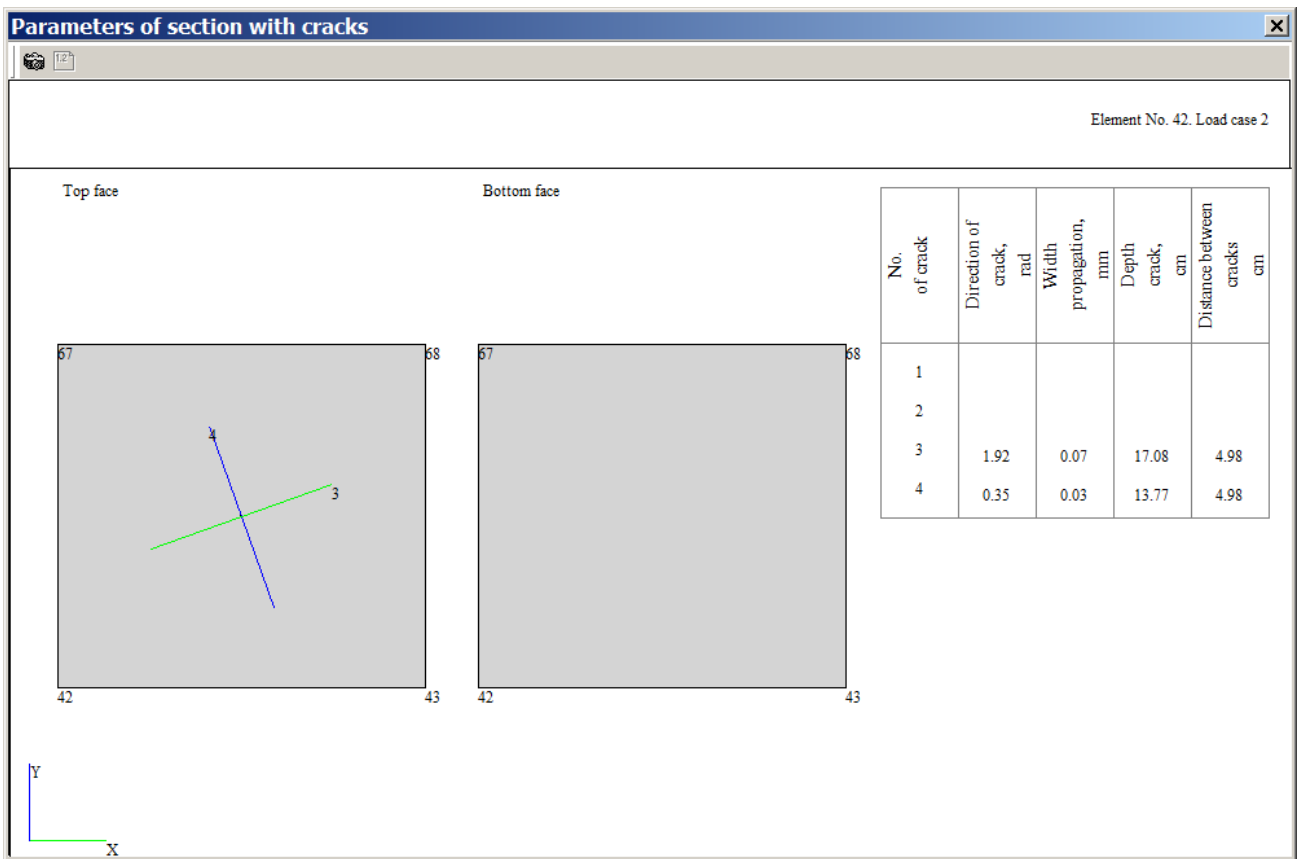



Figure 16.21 Parameters of section with cracks dialog box

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