CHAPTER 11. NONLINEAR PROBLEMS

General provisions

The non-linear processor is intended to solve physically and geometrically static non-linear problems, as well as problems with constructive non-linearity and prestressing.

In geometrically nonlinear problems, there is no linear relationship between deformations and displacements. The solution of these problems is carried out by a step method, and the step is selected automatically.

In physically nonlinear problems, there is no linear relationship between stresses and strains—nonlinear elasticity or elastoplasticity. The solution of nonlinear elastic problems is performed by the step method (formula 18.5.2). The number of steps and load factors are set by the user or calculated by the program. For elastoplasticity problems, an iterative method is used (formula 18.6.2).

The physically nonlinear problems also include creep problems, which are solved by the iterative method (formula 18.6.2).

In problems of constructive nonlinearity, the calculation scheme changes. For example, contact with a support occurs when a certain amount of displacement is reached at some point. For problems of constructive nonlinearity, with one-way constraints and with friction, the iterative method is used (formula 18.6.2).

Step and iterative methods reduce the solution of nonlinear problems to a sequence of linear ones. At each step or iteration, increments of displacements the increments of stresses (forces) are calculated, which are added then to obtained earlier. If there are restrictions, the total stresses are corrected.

In the physical sense, the iterative method searches for such additional compensating loads that impart displacements to a linearly deformable body equal to the displacements of a nonlinear body under a given load. In this regard, this method is also called the method of compensating loads.

The non-linear processor makes it possible to combine linear and non-linear finite elements. Detailed information about nonlinear finite elements is contained in Chapter 19.

The geometric interpretation of the step and iterative methods is shown in figures 11.1, 11.2.

11.1 NONLINEAR ELASTICITY

Modeling of non-linear elasticity of materials is performed by physically non-linear elements. Deformation laws ($\sigma - \varepsilon$ dependence) are used, which make it possible to take into account almost any non-linear properties of the material. Two materials (concrete and reinforcement) are available.

The stiffness matrix of a linear system is formed based on the stiffnesses calculated by numerical integration for each nonlinear finite element. The scheme of numerical integration and the set of stiffnesses used are determined by the type of finite element. The section of the finite element at the integration points is divided into a number of elementary subdomains. In the centers of these subdomains, new values of the physical and mechanical characteristics of the material are determined in accordance with the given deformation diagram.

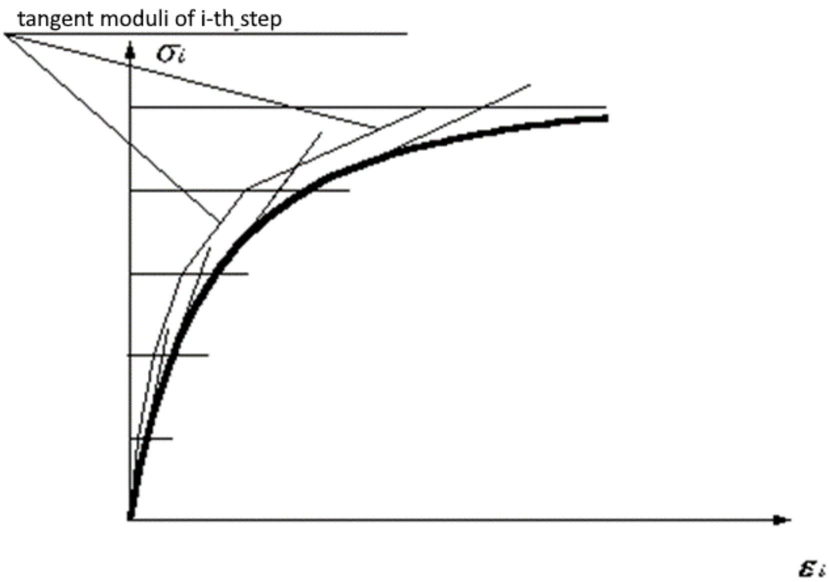

Fig. 11.1. Geometric interpretation of the step method
for the case of uniaxial tension (compression)

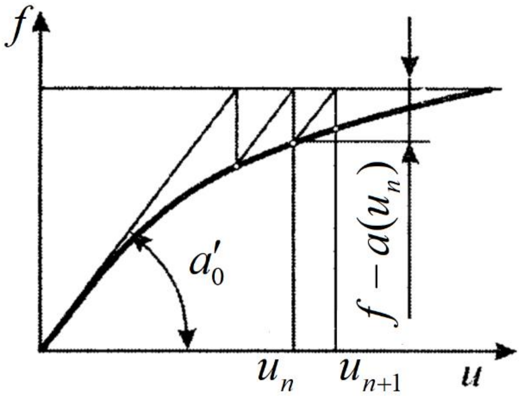

Fig. 11.2. Geometric interpretation of the iterative method
for the case of uniaxial tension (compression)

At each step, the stress-strain state is evaluated, that for bar elements is at the integration points, and for flat and solid elements - at the center of gravity. In the section of the calculation results "Information on the state of materials" the reports are given about the development or achievement of limit states, the appearance of plastic hinges or failure states. The table is formed if, in the process of solving the problem, the material of the section was partially or completely destroyed, or a plastic hinge formed in the section. Otherwise, the table remains empty. The messages indicate the percentage of destruction of the sections of the elements, both in the main and in the reinforcing material. When a plastic hinge is formed in the section, the corresponding text and the value of the limiting moment are printed.

11.2 Geometric nonlinearity

Simulation of geometric nonlinearity is performed using the corresponding linearly elastic finite elements, namely threads, membranes, bending rods and plates. A step method with automatic step selection is applied. At each step, the positive definiteness of the matrix of the linearized system is controlled, which makes it possible to solve problems of the stability of the deformed circuit. A method has also been implemented that makes it possible to solve geometrically nonlinear problems after loss of stability.

For geometrically nonlinear problems, it is recommended to use automatic step selection, or automatic step selection with the search for new forms of equilibrium.

11.3 Physical and geometric nonlinearity

It is used for the problems that require simultaneous consideration of the nonlinear elastic properties of the material and large displacements. A step method with automatic step selection is applied. The selection criterion is a change in geometry and stiffness.

11.4 Constructional nonlinearity

An iterative method is used to solve problems of constructive nonlinearity. Simulation is provided by special finite elements of unilateral links, friction, pretension.

11.5 Elastoplasticity

An iterative method is used to solve the elastoplasticity problems. The library of elastoplastic finite elements contains flexible rods and shells, plane stress and plane strain elements, solid elements, and soil elements.

11.6 Creep

Creep is taken into account for all physically non-linear elements, for which the law of decrease in the elastic modulus in time (in days) must be specified. The calculation can be made after any complete history of non-linear loadings. The results are the total displacements of the nodes and the forces in the elements.

11.7 ASSEMBLAGE

The **ASSEMBLAGE** system is intended for computer simulation of the construction process. In the graphic system, a design scheme is created that contains all the elements that are mounted and dismantled. Assembling and disassembling of an element can only be done once.

At each stage, a design scheme is calculated that contains elements assembled and not disassembled to this stage.

After the circuit is created, it is necessary to form a wiring table containing three data sets:

1. **Stages**. For each stage of construction, the elements that will be assembled and disassembled are indicated. Empty stages are allowed, without assembled and disassembled elements, which are used only to set the load. Each stage must have its own assembling load, the number of stages and assembling loads is the same.

2. **Groups**. For each group of elements of the scheme, the correction factors are set for the modulus of deformation and for the strength of concrete in accordance with the numbers of the stages of erection. Correction factors cannot decrease from stage to stage. If information about groups is not specified, then the characteristics of the material remain unchanged at all stages.

3. **Additional loads.** For each stage, numbers of additional load cases and coefficients (including zero and negative ones) with which these load cases must be taken into account during construction are specified. Additional means those present only during construction. These are, for example, loadings from the storage of building materials, from their movement within a floor or from floor to floor, etc. They are given under numbers greater than the total number of stages of construction. Operational loads on the completed structure are specified on post-stage load cases.

As a result of the operation of the calculation processor of the ASSEMBLAGE system, the forces and stresses accumulated during the construction process are calculated in the elements.

By default, the displacements of nodes during the calculation process are not accumulated, but are calculated for each stage.

When modeling structures made of reinforced concrete, climatic conditions can be taken into account under which freezing or thawing of the laid concrete mixture occurs, which is realized by the coefficients for reducing or increasing the strength and deformation modulus of concrete at various stages of construction. At the same time, the current strength and modulus of concrete deformation, as well as the presence of temporary formwork racks, are taken into account. At each stage of the calculation, the amount of reinforcement in all sections of reinforced concrete elements is determined. The results of reinforcement for each stage are presented in the form of tables, by which it is easy to assess whether the design reinforcement is sufficient.

To use the **ASSEMBLAGE** system in the project, set the **«ASSEMBLAGE» system will be used in project** checkbox in the project settings and click the **Apply** button.

To specify the load cases that will act on the scheme, the load editor should be used (see section 2.15).

There are 4 special types of load cases for linear assembling (Fig. 11.3), for each of which it is possible to select a normative document for the formation of DCL. In addition, design combination of loads (DCL) can be calculated.

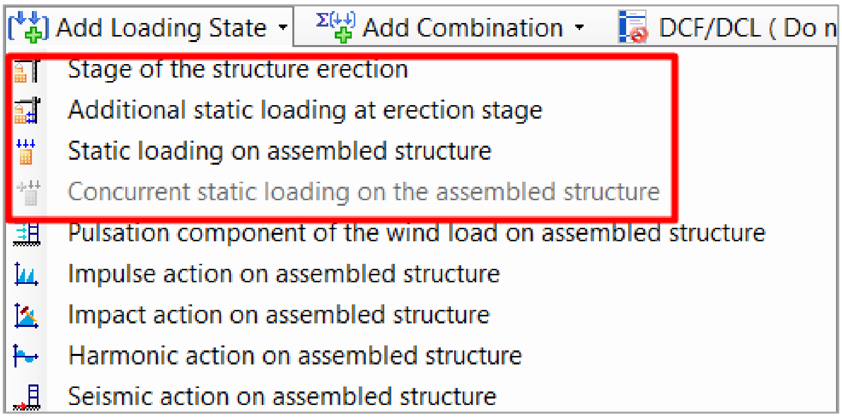

Fig. 11.3. Types of load cases for linear assembly problems

History of erection of the structure contains information about the stages of erection of the structure (Fig. 11.4). The maximum number of construction stages for a structure is 120.

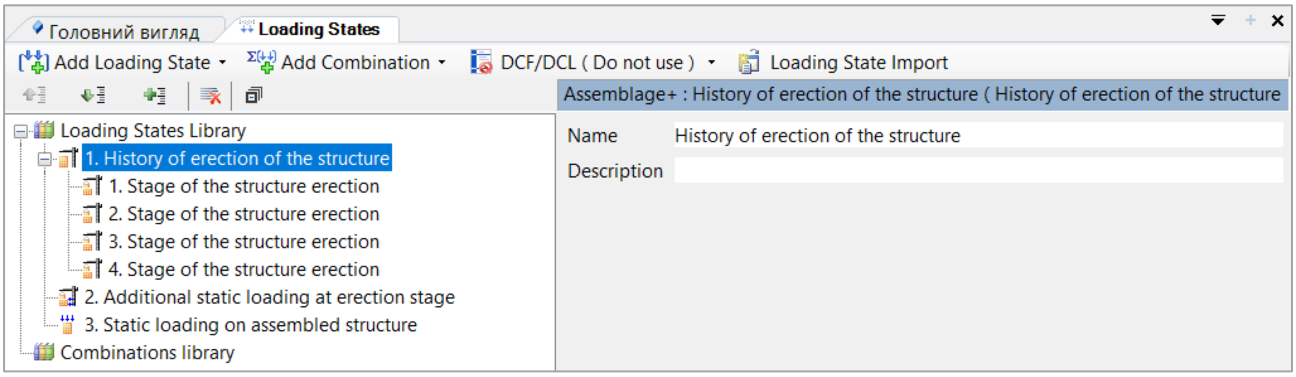

Fig. 11.4. History of erection of the structure

Assembly linear problems can additionally, depending on the selected project parameters, be combined with the problems “THERMAL CONDUCTIVITY” and “DYNAMICS+”. Non-linear assembly problems - with “THERMAL CONDUCTIVITY”, “DYNAMICS+” and FILTRATION problems.

To add a stage in the list of load cases, the history of structure erection must be selected. During the calculation, the stages will be "assembled" one after the other. Usually, a floor of a building is taken as a stage.

In the **Parameters of election stage** area (Fig. 11.5) there is the **Reset displacements** checkbox, which allows for each individual stage, regardless of its location on the diagram, to take initial displacements equal to zero. If the checkbox is not set, then the initial displacements of the current stage will be equal to the displacements of the previous stage.

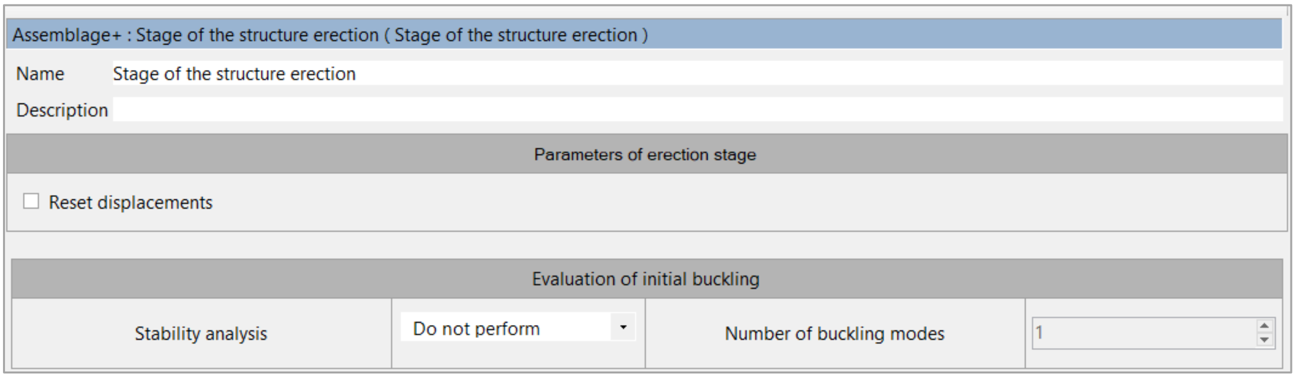

Fig. 11.5. Structure erection stage

In the **Evaluation of initial buckling** area the next parameters can be applied:

- **Stability analysis**— A drop-down list that controls how stability analysis is performed.
- **Number of buckling modes**— When performing a stability analysis, this field sets the number of modes from 1 to 10.

If to select **Additional static loading at erection stage** in the **Add Loading State** drop-down list (Fig. 11.3), then a load will be added that can be combined with erection stages. These can be any loads associated with the construction (storage of materials, formwork racks, etc.).

*The table **Loading combinations** for the **Additional static loading at erection stage** type of loading can not be adjusted. At the same time, the **Loading type** is accepted as **Constant** in the program.*

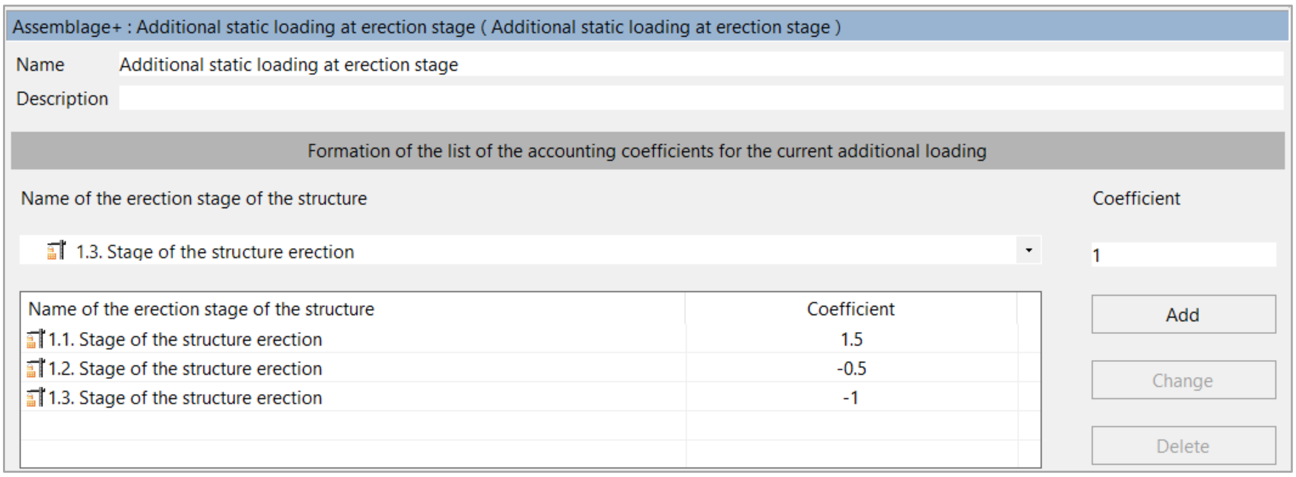

Рис. 11.6. Additional static loading at erection stage

In the **Formation of the list of the accounting coefficients for the current loading** (Fig. 11.6) there is a table that specifies the behavior of the load of this loading: at what stage and with what coefficient it participates.

The erection stage of the structure is selected from the **Name of the erection stage of the structure** drop-down list. The corresponding field indicates the **Coefficient** to the load.

Using the **Add** button, the selected stage with the specified coefficient is entered into the table. If this stage is already in the table, then re-adding will not occur, instead the new coefficient will be added to the existing one.

In order to change the coefficient, it is necessary to select the corresponding line in the table, replace the current value with a new one in the **Coefficient** input field and click the **Change** button, after which the changes will be entered into the table.

To delete a row from a table, select the row and click the **Delete** button.

Load presence on stages is regulated by coefficient, which is aggregating. Thus, if you want to assign load in halves to two stages, you have to set coefficient 0.5 for each of them. If you need to reduce or remove load, use negative coefficient.

If to select the element **Static loading on assembled structure** in the **Add Loading State** drop-down list of the load editor, a load case will be added where the loads that will be applied to the structure after installation are specified, for example, the load from equipment, etc.

And if to select the **Concurrent static loading on the assembled structure** element, a load case will be added, in which loads are specified that accompany loads on the assembled structure (for example, a brake load accompanies a crane load).

Assembling stages tables

To add circuit elements to assembly stages, use the **Assignment ⇨ Assemblage** command or click the button on the toolbar.

The **Assemblage** active mode panel is represented by the following tabs (Fig. 11.7):

- Editor for Assembling Stages;
- Editor for Coefficient Groups;
- Disassembling in Dynamics+;
- Assignment Policy.

At the top of the panel there is an Assembling stage drop-down list, from which it is necessary to select the desired assembly stage.

The Editor for Assembling Stages tab (Fig. 11.7) contains two tabs - Assemblage and Disassembling, where the Elements of assembling / disassembling input fields are located. These are text entry areas that contain a list of elements assembled or disassembled at the selected stage. The list of elements is filled in automatically (see below). It is also possible to enter element numbers manually.

*The **Elements of assembling / disassembling** input area is divided into two parts: the numbers of finite elements are entered at the top, and the numbers of architectural elements are entered at the bottom.*

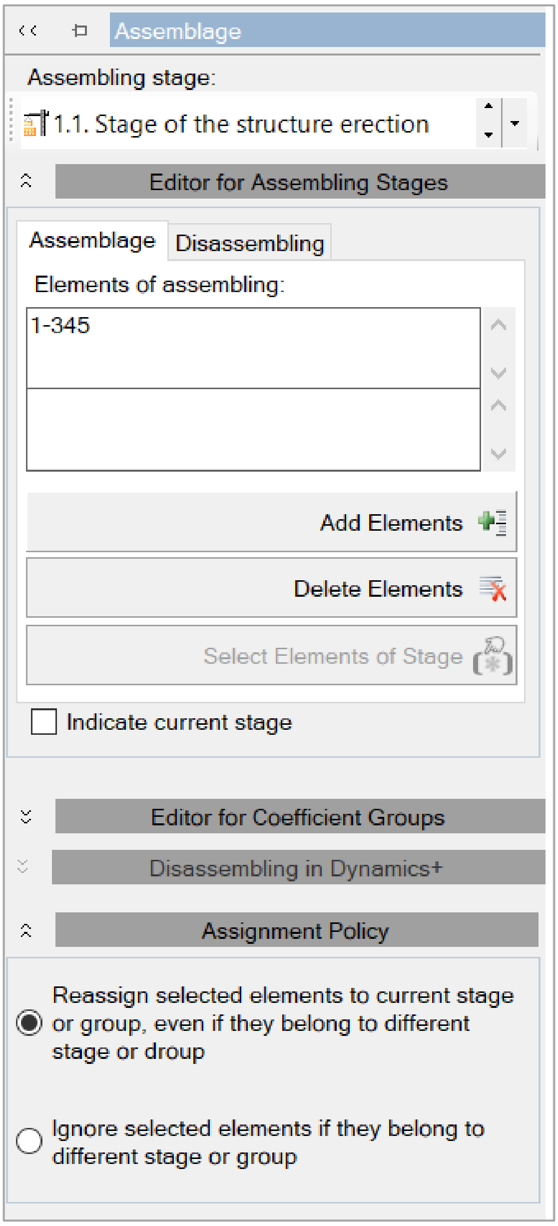

Fig. 11.7. **Assemblage** mode panel

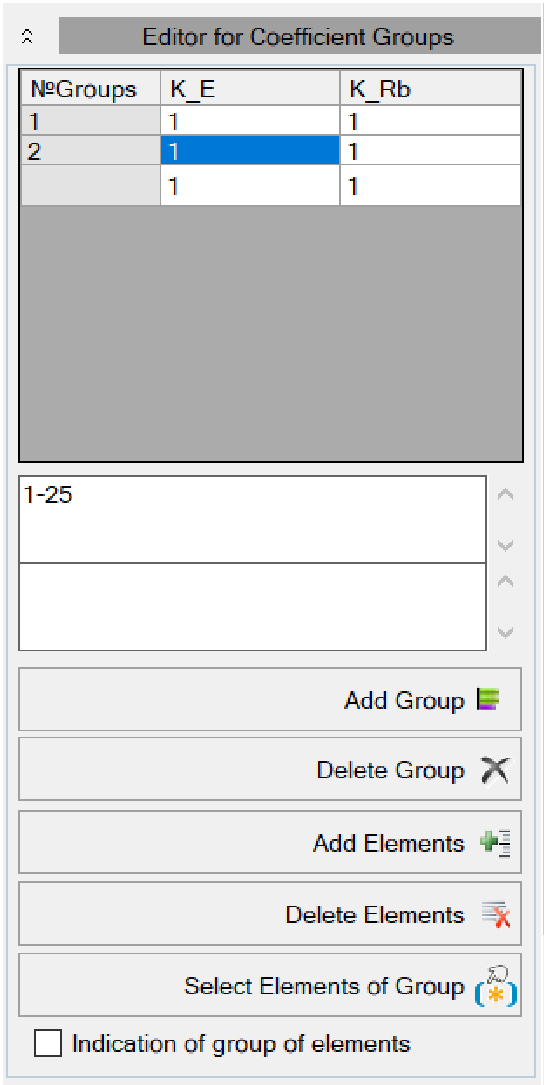

Fig. 11.8. **Editor for Coefficient Groups** tab

The following buttons are also located here:

- **Add Elements** — allows to add a list of elements selected in the diagram to the **Elements of assembling** input area.
- **Delete Elements** — allows to delete the list of elements selected in the diagram from the **Elements of assembling** input area.
- **Select Elements of Stage** — allows to select elements on the diagram from the **Elements of assembling** input area.

Only previously assembled elements are available for disassembling. Thus, in the Elements of disassembling input area, only those elements that were entered in the input area of the assembly elements at the previous stages can be entered. Accordingly, at the first stage, no element can be entered into the Elements of disassembling input area.

When the Indicate current stage checkbox is set, the selection of elements of this stage blinks on the diagram.

In the **Editor for Coefficient Groups** tab (Fig. 11.8), a table of groups is available with the ability to set the coefficient to the modulus of elasticity (**K_E**) and the coefficient to the strength of concrete (K_Rb), which is used in the **ASSEMBLAGE +** subsystem.

Below the table there is a text input area where a list of elements of this group is displayed. The list of elements is filled in automatically (see below), it is also possible to enter the element numbers manually

Control buttons for managing groups:

- **Add Group** — adds a new group to the end of the list.
- **Delete Group** — removes the selected group from the list.
- **Add Elements** — allows to add a list of elements selected in the diagram to the input area for group elements.
- **Delete Elements** — allows to delete the list of elements selected on the diagram from the input area of the group elements.
- **Select Elements of Group** — selects the elements on the diagram from the input area of the group elements.

When the **Indication of group of elements** checkbox is set, the selection of elements of this stage blinks on the diagram.

The **Disassembling in Dynamics+** tab (Fig. 11.9) is used to calculate the stability against progressive collapse in case of local destruction of load-bearing structural elements. For an assembly dynamic problem, both linear and non-linear, for the last stage of assembly, elements are indicated that will be disassembled in a dynamic setting.

For dynamic loading, it is necessary to define a graph of changes in the reactions of disassembled elements applied with the opposite sign (see Chapter 13).

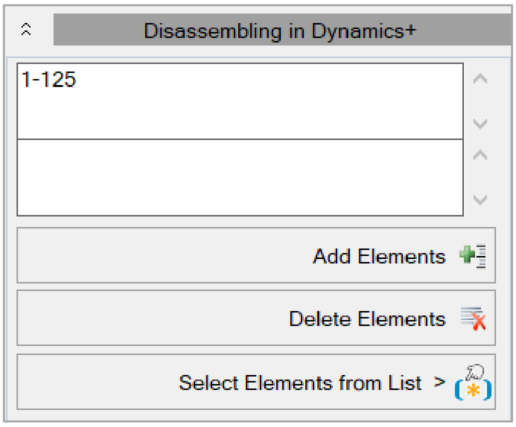

Fig. 11.9. **Disassembling in Dynamics+** tab

In the **Assignment Policy** tab, using the corresponding switches, you can control the logic of working with elements belonging to other stages.

11.8 Some remarks about Fractures and plasticity hinges in the nonlinear elements

SP LIRA10 has the ability to identify fractures for monomaterial and reinforced non-linear elements. The first opportunity to identify destruction can be realized during the calculation by turning off the calculating processor in running of the background (Figure 11.10). By the color of the elements, you can see the fracture and cracks in the bar, shell and solid elements.

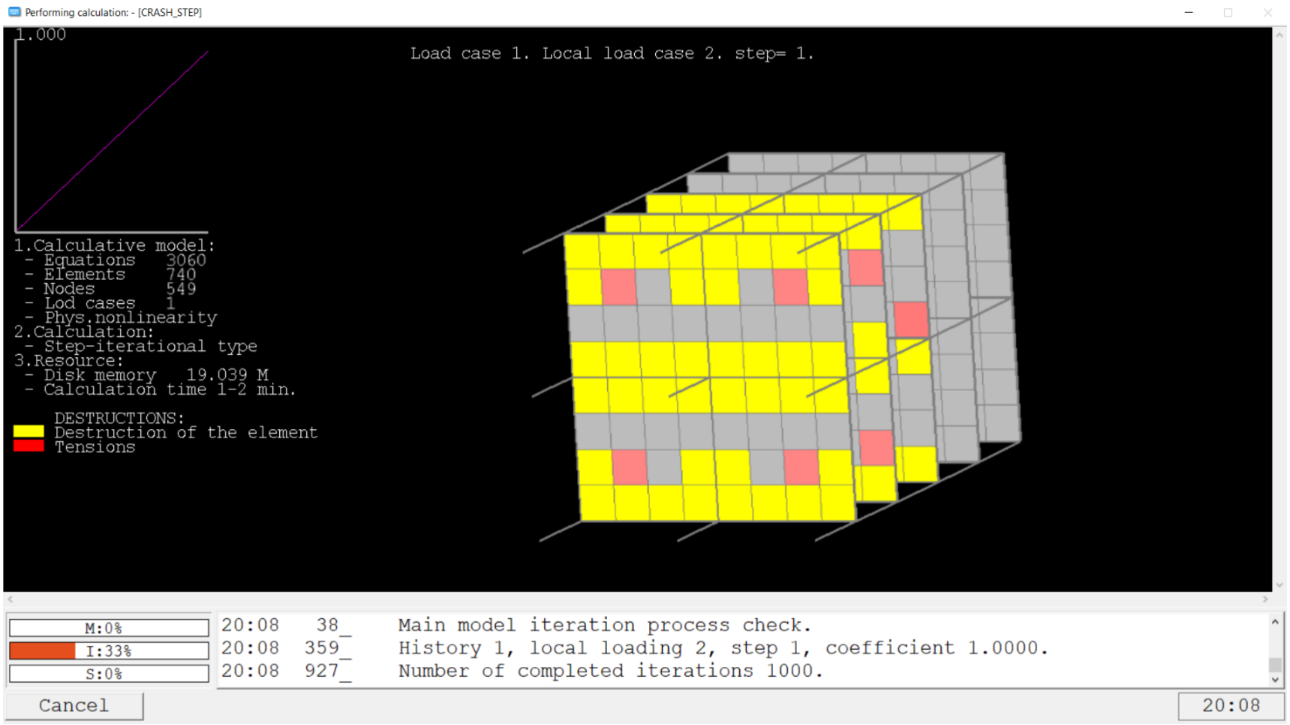

Fig. 11.10 Indication of destructions in elements during analysis

The second opportunity to identify fractures is through the **Fractures** mode in the calculation results (see section 3.7). However, this feature is only available for non-linear plates and only for static non-linear problems and the last stage of non-linear editing.

SP LIRA10 provides several options for partial or complete failure of plates:

- **Tensile Cracking.** If the maximum stresses in the concrete exceed the ultimate tensile strength of the concrete, but the element is not fractured or plastically hinged, then a tensile crack will be indicated on the element.
- **Compression Cracking.** If the minimum stresses in the concrete exceed the compressive strength of the concrete, but the element is not fractured or plastically hinged, then a compression crack will be indicated on the element.
- **Destruction in step plate elements when stresses are exceeded according to a given strength theory** — the strength theory is set in the material editor. If the equivalent tensile or compressive stress exceeds a predetermined critical value, then this will be identified as failure of the element. If the element is completely destroyed, then it does not participate in the further calculation, but it does not release energy to the neighboring elements, which caused the destruction.
- **Destruction due to the formation of a plastic hinge according to the selected normative document in plates -** the norms for a plastic hinge are set in the editor of a non-linear material. There are 4 possible options for plastic hinges:

- According to SNiP-2.03.01-84*;
- According to Eurocode 2;
- According to SP 63.13330.2012(2018);
- According to DBN V.2.6-156:2010.

Plastic hinges according to Eurocode 2, SP 63.13330.2012(2018), DBN V.2.6-156:2010 work identically. For reinforced concrete materials, the minimum negative deformations in concrete are checked for compression, and the maximum positive deformations in reinforcement are checked for tension. They are then compared with those set in the Nonlinear Material Editor to indicate that a hinge is forming. In this case, the deformations are taken from those obtained in the finite element calculation. For monolithic materials, the maximum and minimum deformations in the base material are checked. If, according to any of these criteria, the deformations go beyond the established boundaries, an indication of the hinge appears in the elements.

Plastic hinges according to SNiP-2.03.01-84* are calculated based on the forces received. Then, according to the SNiP methods, in plates with reinforcement, it is checked whether there is a formation of a plastic hinge under the assumption of a zero tensile strength of concrete and a bilinear diagram of concrete for compression. And for monolithic plates, the formation of a plastic hinge is simply checked from given forces with actual, user-defined material properties.

In SP LIRA10, there is a criterion by which a material can be considered cast-in-place, or reinforced concrete. When the thickness of the distributed reinforcement strip is less than 0.0005 of the plate`s height, then the section is considered to be concrete, not reinforced concrete. This criterion is taken from the norms that determine whether the section is concrete or reinforced concrete. Therefore, if the percentage of reinforcement in the plate is less than 0.05%, then it will be checked not as for reinforced concrete with zero tensile strength of concrete, but as for concrete, in which the tensile strength is taken into account everywhere, according to a given material diagram.

- **Destruction in rod and plate elements according to special tests for the membrane group of forces**.

When taking into account plastic hinges in plate elements, additional simplified internal checks for the membrane force group are carried out, considering the load-bearing capacity of the structure in the event of a sudden crack formation. The obtained membrane forces from the FE calculation in plates are checked by the criterion of reinforcement tension (excluding concrete), and for compression (both concrete and reinforcement are taken into account).

Fracture condition per membrane group for reinforced concrete:

$$\frac{N}{A_s} > R_s$$

or

$$-N > R_s \cdot A_s + R_b \cdot A_b$$

where N is the membrane force in the most dangerous projection;
A_s - the area of the reinforcement in the section in the direction;
A_b — concrete section area;
R_s — ultimate tensile strength of reinforcement;
R_b — ultimate compressive strength of concrete.

Fracture condition per membrane group for concrete:

$$\frac{N}{A_b} > R_{bt}$$

or

$$\frac{-N}{A_b} > R_b$$

where N — the membrane force in the most dangerous projection;
A_b — section area;
R_{bt} — ultimate tensile strength;
R_b — ultimate compressive strength.