SCAD Office
Version 23
SCAD++

V.S. Karpilovskyy, E.Z. Kryksunov, A.A. Maliarenko,
A.V. Perelmuter, M.A. Perelmuter, S.Y. Fialko

Kyiv, 2022

SCAD Office.V.23.System SCAD++ / V.S.Karpilovskyy, E.Z.Kryksunov, A.A Maliarenko, A.V.Perelmuter, M.A.Perelmuter, S.Y. Fialko. — Electronic Edition, SCAD SOFT, 2022. — 895 p.

The book is intended for users of the software application pack. It can be useful both for beginners who do not have an experience of using structural analysis software and for those who know software of this type well enough and have an experience of working with this particular application. The book deals successively with all aspects of preparing data in the graphics interactive mode, performing an analysis, reviewing and documenting results, as well as with peculiarities of using analysis and design post-processors. A minimum amount of theoretical basics is given to help users understand better analysis methods implemented in the software.

The book may be of interest for students of appropriate specialities and developers of similar software products.

© Authors, 2022

1. SCAD Office System

SCAD Office is a set of programs for performing the strength analysis and design of different types of building structures. The system includes programs of four types:

- **Structure CAD** (**SCAD**) – an integrated system for finite element structural analysis intended for solving problems of designing rather complex buildings and structures;
- auxiliary programs which provide the "maintenance" of **SCAD** and enable to: create and calculate the geometric properties of various sections of bar elements (**Section Builder**, **Consul**, **TONUS**, **SEZAM**), determine loads and actions on the designed structure (**WeST**), calculate the subsoil parameters for the structures on the elastic subgrade (**Cross**); import data from architectural systems and generate enlarged models (the **FORUM** preprocessor integrated into **SCAD**);
- design and analysis programs **Kristall**, **ComeIn, Decor**, **ARBAT**, and **ZAPROS,** which enable to solve specific problems of checking and analyzing steel, masonry, timber, and reinforced concrete structural members, and soil and foundation members according to the requirements of codes;
- design and engineering programs **COMET** and **Monolit**, which are used for the development of the design documentation at the stage of the detailed design elaboration.

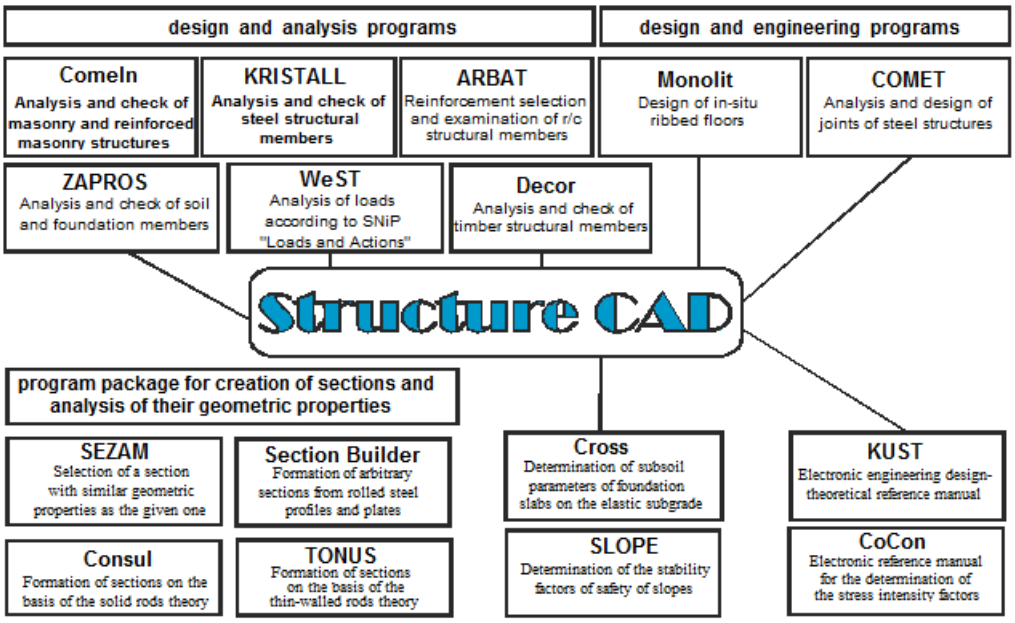


Figure 1-1. *Structure of the* **SCAD Office**

SCAD includes advanced data preparation, calculation and results analysis tools and has almost no limitations on the sizes and shape of the designed structures. However, solving "simple" problems, which can be quite time-consuming, is not less (and in many cases even more) important for the engineer. The check of elementary beam sections, arrangement of loads on the structural members, determination of the geometric properties of the compound sections – is not a full list of such routine design problems.

These circumstances stimulated the development of a series of "small" satellite programs of **SCAD** focused on solving routine design problems. They make up the **SCAD Office** package together with the main computing system itself. The satellite programs are based on the common data representation, methods of control, verifications of compliance with codes, display of results, generated documents etc. Any of the programs included in **SCAD Office** can be used separately from the others.

The design procedures were selected and implemented in the system programs on the basis of the following considerations:

- applicability — it makes no sense to develop single-use industrial software products;
- clear formulation of the problem — the software is mass-user oriented, therefore it must not implement problems with an unclear formulation or an ambiguous solution;

- relative complexity of the traditional methods — it makes no sense to program the trivial calculations, the problem must be such so that the use of the program would reduce the labor costs, and/or the probability of calculation errors;
- availability of reference materials.

Moreover, taking into account that the programs can be used not only by the experienced designers, but also by the beginner engineers, the developers tried to create such software products that could prevent the user from missing any of the numerous checks required by the codes. In particular, the set of the considered designs is based among other things on the fact that only the structures the requirements to which are completely defined by the codes are analyzed.

1.1. Structure CAD

Structure CAD (**SCAD**) is an integrated system for finite element structural analysis and design which enables to determine the stress-strain state of structures subjected to static and dynamic actions, and to perform a set of functions for designing the structural members.

Project

The software is based on a system of functional modules interconnected by a single information environment. This environment is called a **project**; and contains full information on the design model which is given in the native formats of the software. During the generation of the design model the project is filled with information and saved on the disk in a file with the **.spr** extension. The name of the project and the file are specified when creating a new model.

The initial data for performing the analysis can be prepared either with the help of the interactive graphical tools or by describing them in the text format. It should be noted that in the current version of the software all the input information necessary for performing the majority of practical calculations can be prepared graphically, and the text description may become necessary only in the case of unique designs. A text file is a convenient format for storing the data. It can be easily compressed with the help of the popular archiving software, and, if necessary, it can be converted to the native project formats of **SCAD**. Another attractive feature of the text format is its compatibility with the previous versions of the software which enables to use the software for performing the checks of the structures designed earlier.

Functional Modules

SCAD functional modules are divided into four groups. The first group includes modules that provide the input of the initial data in the interactive graphical mode (graphical preprocessor), perform static and dynamic calculations (processor), graphical check and analysis of the results of the calculation (graphical postprocessor). Modules of the second group enable to calculate the design combinations of forces, combinations of loadings, principal and equivalent stresses, reactions, loads from the structural fragment, and to perform the stability analysis, generate the response spectra and the spectra of the amplitude-frequency characteristics, and are called the design postprocessors. The documentation of the analysis results is performed with the help of the modules of the third group. The fourth group includes the design modules (design postprocessors) which enable to perform the check and selection of the reinforcement in the reinforced concrete structural members, check and selection of the sections of the steel structural members.

The modular structure enables every user to customize the **SCAD** configuration to meet his particular needs concerning the classes of the solved problems, tools for creating the design models, analyzing and documenting the results of the calculation.

All functional modules are implemented in a single graphical environment. The user interface, scenarios of the interaction of the user with the system, functions of the control of the initial data and the analysis of the results are completely unified which makes it easier to master the software and provides a logical sequence for performing the operations.

The structure of the software and the interrelation between its components are shown in Fig. 1.1-1.

Processor and Finite Element Library

A highly efficient processor enables to solve the static and dynamic problems with a large number of the degrees of freedom. The calculation is accompanied by a detailed log which can be analyzed either during the calculation or after its completion. The system of the control of the initial data performs the check of the design model and logs all the detected errors and warnings.

The finite element library contains different types of bar elements including hinged bar and frame elements, and elements of a grade beam on the elastic subgrade, and enables to take into account shear in a bar section. Plate elements which are represented by three- and four-node plate, shell, and deep beam elements can have additional nodes on their sides, and provide the solution of the problems for the materials with different properties (allowing for the orthotropy, isotropy, and anisotropy). Moreover, the library includes different types of solid elements, a set of three- and four-node multilayer and axisymmetric finite elements, as well as the special elements for modeling the constraints of finite rigidity, elastic constraints, etc.

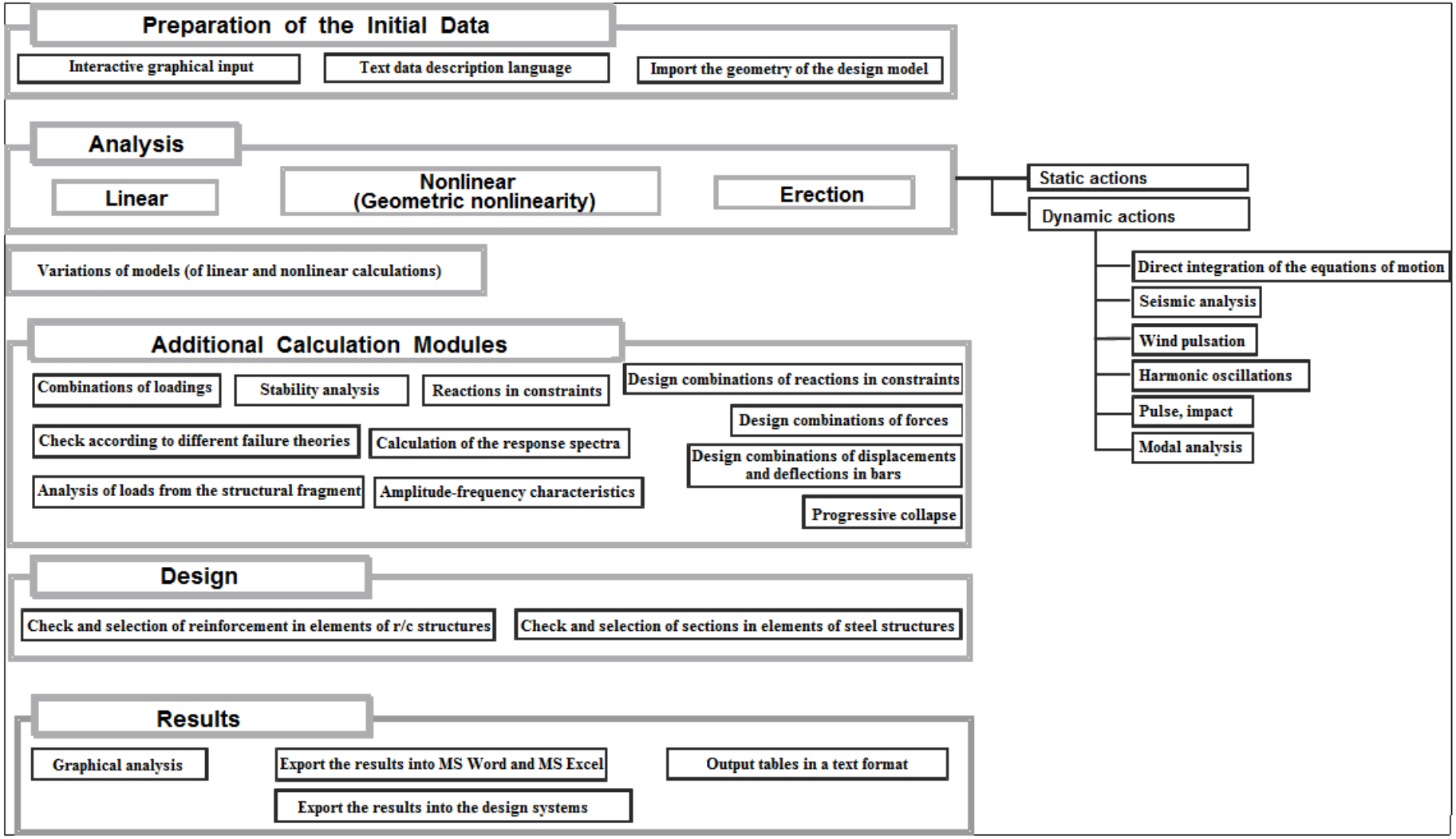


Figure 1.1-1. *Structure of* **SCAD**

Graphical Tools for Generating a Design Model

Two types of graphical modules for creating the design models are implemented in **SCAD**. They include a graphical preprocessor traditional for the finite element software where the finite elements are used as the main "building material" for creating a design model, and the **FORUM** preprocessor for creating the enlarged models.

The traditional preprocessor provides a wide range of tools for creating the models which include the functions of generating the models based on the parametric prototypes of the structures, generating the meshes of elements on a plane and in space, copying the fragments of the models, assembling from the submodels and groups, different functions of geometric transformations. All the main parameters of the models including the stiffness properties of the elements, boundary conditions, static and dynamic loads etc. are specified in the graphical interactive mode. The graphical interface has been adapted as close as possible to the technology of creating and modifying the design models and allows for the peculiarities of this type of information processing.

The preprocessor includes the parametric prototypes of multi-storey and single-storey frames, trusses with different configurations of chords and lattices, grade beams, and surfaces of revolution (cylinder, cone, sphere, and torus). The boundary conditions, types and rigidities of finite elements can be automatically assigned to them in the process of their generation.

Special tools are provided for creating design models the surface of which is described analytically. These tools enable to automatically generate a finite element mesh on the surface specified as a function of two variables. An automatic meshing is used to generate arbitrary meshes on the plane. This way a mesh can be applied to any area of the design model.

The set of functions for modifying the geometry of the design model includes various types of geometric transformations which enable to change the scale of the whole model or of the selected fragment, rotate about the given axis, mirror the model, move a part of the model by a given distance etc. Together with a wide range of tools for working with the selected nodes and elements these functions enable to create design models of almost any complexity.

Grid lines are used to provide an engineer with a familiar environment for working with the design model. These grid lines can be used for performing the operations of creating a model and its fragmentation. They can be displayed on the whole model or on any fragment.

Enlarged Design Models and Connection with Other Systems

A design model or its part can be also generated on the basis of the information obtained from:

- other design programs;
- mesh generators;
- graphics editors;
- engineering programs.

It can be imported from such popular systems as AllPLAN, ArchiCAD, Revit, Tekla, Advance Steel, files in the DXF and DWG format of AutoCAD, a series of other formats (for example, IFC, SDNF, ...).

Obviously, an architectural model can not be fully and automatically converted into a design model. This is due to "the architectural excesses" which are not always the elements of the design model and do not affect the results of the calculation, not always accurate junction of the elements of the model, lack of the data on the boundary conditions of the elements, loads, materials, etc. The complexity of turning the imported model into the design one usually depends on how consistently the architect and the design engineer have worked at the stage of creating an architectural model.

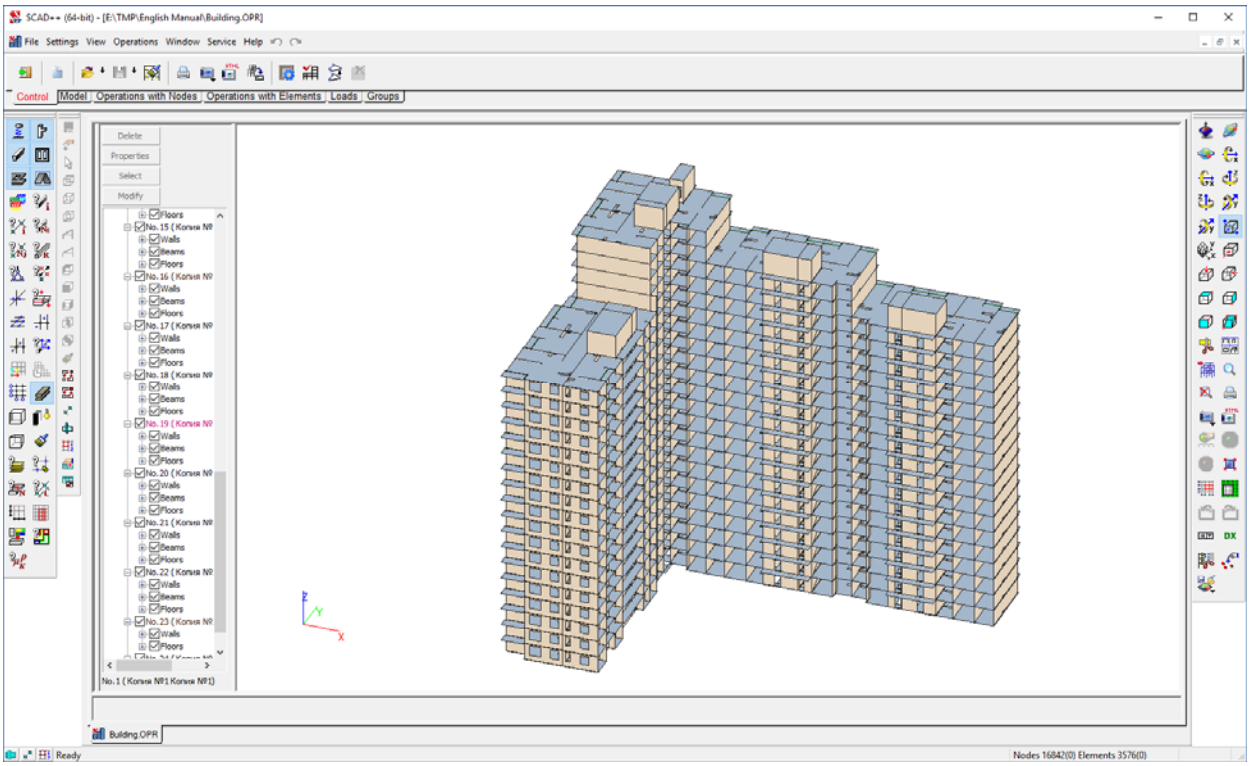


Figure 1.1-2. *Design model generated in the **FORUM** preprocessor*

SCAD includes a special preprocessor **FORUM** in order to simplify this process. It uses objects very close by purpose and name to those of the architectural model to generate a design model. They include columns, beams, floors, walls, and roofs. The representation of the geometry of the design model in the **FORUM** preprocessor (Fig. 1.1-2) enables to:

- create a structured design model from enlarged elements;
- save the structure of the object specified in the architectural model;
- transfer the structure of the object into the preprocessor of **SCAD** using an automatic transformation (meshing) of the enlarged model into the finite element model (design model).

Groups

Groups of nodes and elements play a special role in the generation of a design model and the analysis of results. Groups are named sets of nodes or elements that can be used repeatedly in various operations. Characteristic parts of the structure, for example, floors, elements of a spatial frame, walls, or other sets of objects can be combined into groups. This process is completely controlled by the user. The main thing is that groups are available at all stages of working with the model – during its generation, analysis and documentation of the results of the calculation. The structure of the graphical environment enables to localize the needed information within one or several groups of nodes and elements.

Filters

There is a recent tendency for design models to grow more and more complicated; such models contain an ever increasing number of nodes and elements. This substantial increase in the dimensionality of the problems has required to revise the main criteria of the efficiency of the process and methods of creating the design models and the analysis of the results of the calculation. When dealing with large and detailed models, it is natural to emphasize not the functions of formation (though the role played by those is by no means less important) but functions of control of the created model. A well-developed system of filters implemented in **SCAD** is particularly important. The filters are used to establish the rules for displaying the information on the model, functions of visualization and fragmentation of the model. The filters enable to select the information on the design model which has to be displayed by tens of criteria. Color display

tools are widely used for displaying the information. Together with the fragmentation they enable to "get" to any parameter regardless of the dimensions of the model.

Graphical Postprocessor

When the considered design models are large, it becomes problematic to analyze the results. The amount of the resulting information that can be obtained is usually much greater than a man can comprehend and analyze. Therefore, here together with the solution of purely technical tasks of improving the time factors (reaction of the system to the user's request or time spent for deleting lines of an invisible contour during the generation of isolines and isofields) problems of finding the objects with the values of the analyzed factor critical for this problem appear. The operations of fragmentation, system of filters, customizable color scales implemented in the software provide a quick access to all types of the resulting information. It should be noted that you can select the part of the design model where the results (forces, displacements) from the given range of values were obtained "hiding" the rest of the model.

The results of the calculation can be given in the form of the deformed models, diagrams, isolines, and isofields. The numerical values of the factors can be displayed on the model as well. The process of the deformation of a model can be animated for static and dynamic loadings, and this process can be recorded as an AVI movie. Any graphic information can be printed or saved as a Windows metafile (WMF, EMF). Along with the results of the calculation, the graphical analysis tools enable to display the results of the module of the reinforcement selection in the reinforced concrete structural members on the model in the form of diagrams (for the bar elements), or isolines and isofields (for plates); these results may include such information as the area of reinforcement in the given direction, crack opening width, percentage of reinforcement etc. It makes the representation of the results more familiar for the engineer and creates additional comfort for him.

Documenting the Results

The modules of documenting the results of the calculation enable to generate tables with the initial data and the results in the text or graphical formats, and to export them to MS Word or MS Excel. Tables are generated taking into account the groups of nodes and elements; they can be supplemented with comments and graphical data. Thus, a report document can be edited with the help of MS Word and become convenient for a particular user (for example, in accordance with the company's internal standards). When exported to MS Excel the results can be subsequently processed in whatever nonstandard way the circumstances require.

1.2. Design and Analysis Programs

Design and analysis programs **Kristall**, **ComeIn, Decor**, **ZAPROS**, and **ARBAT** enable to perform checks and analysis of steel, masonry, timber, and reinforced concrete structural members and their joints according to the requirements of codes. Their development is based on the focus on the strict and, if possible, full compliance with all the requirements given in the codes. The connection with **SCAD** is established by transferring the values of the design combinations of forces for a particular element.

Kristall

Kristall enables you to perform checks of members and joints of steel structures for compliance with requirements of SNiP II-23-81* "Steel Structures. Design Codes", SP 53-102-2004 "General Rules on the Design of Steel Structures", SP 16.13330 revised and updated edition of SNiP II-23-81* "Steel Structures", DBN B.2.6-163:2010 and DBN B.2.6-198:2014 "Steel Structures" and Eurocode-3. Moreover, state standards related to SNiP, SP and DBN, "Manual on Design of Steel Structures (to SNiP II-23-81*)/ Kucherenko CSRIBS" and some methodological positions of SNiP 53-1-96 were used when creating the software.

The following modes are implemented in the application:

- **Steel** — implements recommendations of SNiP, SP and DBN on selection of steel grades;
- **Assortment of Rolled Profiles** — enables to browse through steel profile assortments and provides profile properties;

- **Bolts** — enables to browse through the assortment of bolts;
- **Limit Slenderness** — enables to assign limit slenderness according to the recommendations of SNiP;
- **Service Factors** — is used to select values of service factors for elements (γ_c) according to the recommendations of SNiP, SP or DBN;
- **Materials for Welding** — implements requirements of SNiP, SP and DBN for selection of materials for welded joints and for selection of the design strength values for welded joints;
- **Deflection Limits —** provides tables from SNiP 2.01.07-85* "Loads and Actions" (or DSTU B.V.1.2-3:2006 when working with DBN B.2.6-163:2010 or DBN B.2.6-198:2014) with limitations on deflections of structural members;
- **Assortment of Sheet and Plate Steel —** provides information on assortments of sheet and plate steel;
- **High-Strength Bolts —** provides sizes and properties of high-strength bolts, nuts, and washers;
- **Envelopes** – enables to determine unfavorable combinations of multiple loads acting on bending members, and to plot the envelope diagrams of moments and shear forces;
- **Influence Lines** – enables to plot bending moment and shearing force influence lines for continuous beams;
- **Geometric Properties** — enables to calculate the geometric properties of a given cross-section;
- **Effective Lengths** – implements recommendations from SNiP, SP, DBN and Eurocode-3 on the determination of effective length factors;
- **Resistance of Sections** – enables to determine the utilization factors of restrictions for any cross-section type available in the application database;
- **Bolted**, **Friction** and **Welded Connections** — enable to determine the utilization factors of restrictions and to plot the interaction curves for different structural designs of the given connections;
- **Local Stability** – enables to check the local stability of webs and chord plates of bending or compression members;
- **Trusses**, **Truss Member**, **Beams**, **Continuous Beams** and **Columns** — enable to check the strength, stability and limit slenderness of the members; determining the design values of internal forces and their combinations caused by the given external loads. It is possible to find the cross-section;
- **Base Plates** — this mode deals with parts of a column base plate with various types of bearing;
- **Sheet Structures** — this mode enables to check strength and stability of sheet structures (round cylindrical, conical shells, cylindrical panels).

ComeIn

ComeIn enables you to perform checks of load-bearing capacity of structural members of masonry and reinforced masonry structures for compliance with requirements of SNiP II-22-81 "Masonry and Reinforced Masonry Structures", SP 15.13330.2012, and documents related to these codes.

The following elements can be checked by the application: axially and eccentrically loaded columns of various cross-sections in their plan; masonry lintels — coursed, Dutch, and arched; reinforced concrete lintel beams; exterior and interior building walls with and without openings; basement walls.

Along with the check of the general strength and stability of members, there is a check of the local strength at the areas of bearing of beams, purlins, and other members on walls and columns.

The checks are performed both for undamaged structural members and for those with cracks in masonry and fire damages caused by high temperature. "Recommendations for Strengthening Masonry Structural Parts of Buildings and Structures. V.A. Kucherenko CSRIBS of the State Construction Committee of the USSR" document was used in the checks of damaged structures.

Another subject of checking is the load-bearing capacity of axially and eccentrically loaded members reinforced by steel battens, and of walls weakened by additional openings.

The application also performs a detailed check of the areas of beams, slabs, and trusses bearing on masonry (and reinforced masonry in some modes) walls and columns. The strength of hanging walls is checked in the areas of the foundation beams bearing on fixed supports.

ARBAT

ARBAT enables you to check the load-bearing capacity and to select the reinforcement for concrete and reinforced concrete structural members, to calculate the deflections in reinforced concrete beams, to check the local strength of reinforced concrete structural members (including inserts) in compliance with requirements of SNiP 2.03.01-84* "Concrete and Reinforced Concrete Structures" or SNiP 52-01-2003, SP 63.13330. The analysis is performed taking into account the ultimate and serviceability limit states for the design combinations of forces (DCF) selected automatically depending on the specified design loads in compliance with SNiP 2.01.07-85* «Loads and Actions» or SP 20.13330.2011. The selection of reinforcement and the check are performed for reinforced concrete structures from heavy-weight, fine-grained and lightweight concrete with the reinforcement steel of the classes provided by the codes, and the reinforcing wire.

Reinforcement is selected for multi-span and single-span beams of a constant cross-section (rectangular section, T-section or I-section) and for columns (rectangular section, T-section, I-section, ring, or round section) using the criteria of strength and crack resistance.

The **Strength of Sections** modes (Fig. 1.2-1) enable to determine the load-bearing capacity of a section depending on the arrangement, diameter (area), and class of reinforcement, the class of concrete, the conditions of operation, and the allowable crack opening width. In the general case, the analyses are performed for the action of a longitudinal force, bending moments, torque, and shear forces acting in the principal planes of inertia. SNiP 2.03.01-84* does not regulate any checks of reinforced concrete members for the serviceability limit states under the action of moments in two planes. If SP 52-101-2003 or SP 63.13330 is selected, the analysis is performed on the basis of a non-linear deformation model.

ARBAT enables to calculate the deflections at the bending of a reinforced multi-span and single-span beam under a given load. The deflections are calculated for a rectangular section, T-section or I-section in compliance with Sec. 4.31 of SNiP 2.03.01-84* (Sec. 7.3.4 of SP 52-101-2003, Sec. 8.2.21 of SP 63.13330). The curvature is determined taking into account cracks in the tension area.

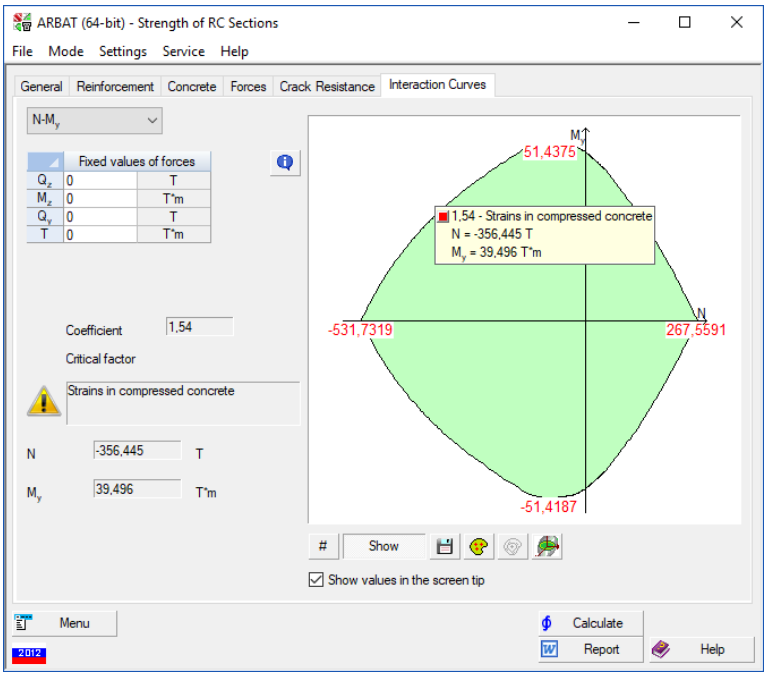


Figure 1.2-1. *The check of the load-bearing capacity of a column in* **ARBAT**

The application enables to perform a check of a given structure of a rectangular area of a solid slab bent in one and in two directions.

ARBAT enables to check the reinforced concrete structural members for the local loads (local compression, punching, tearing) in compliance with the requirements of SNiP and SP, and to perform the analysis of inserts, short cantilevers, and keys.

Decor

DECOR enables you to perform structural analysis and various checks of members and joints of timber structures for compliance with requirements of SNiP II-25-80, SP 64.13330.2011. The application also provides reference data which is often used when designing timber structures. The modes implemented in the application are classified into the reference (the **Information** group) or the design ones (the **Analysis** group).

The reference modes include:

- **Deflection and Strain Limits** — provides the limit values of deflections of timber structural members and limit values of deformations of joints;
- **Densities** — provides information on the density of wood according to Annex 3 of SNiP II-25-80 (Annex E of SP 64.13330.2011);
- **Range of Timber** — provides information on the sizes of softwood sawn timber according to GOST 24454-80;
- **Design Strength** — enables to calculate the design strength for the type of stress-strain state selected by the user;
- **Timber** — provides data on the maximum moisture content of timber, types and grades of glue;
- **Limit Slenderness** — provides information on the limit slenderness of structural members.

The design modes include:

- **Geometric Properties —** enables to calculate the geometric properties of a given cross-section;
- **Effective Lengths —** implements recommendations of Sec. 4.21 of SNiP II-25-80 (Sec. 6.23 of SP 64.13330.2011) for determination of the effective lengths;
- **Resistance of Connections** — enables to determine the utilization factors of restrictions for notched connections and connections with cylindrical dowels;
- **Resistance of Sections** — enables to determine the utilization factors of restrictions for any cross-section type available in the application database, under the action of arbitrary forces; and to plot the interaction curves for any admissible combination of the pairs of forces;
- **Continuous Purlins** — enables to check continuous purlins for the ultimate and serviceability limit states;
- **Cantilever Purlins** — enables to check cantilever purlins under uniformly distributed loads for the ultimate and serviceability limit states;
- **Beams** — enables to check single-span beams with various boundary conditions for the ultimate and serviceability limit states;
- **Columns** — this mode enables to check columns and posts;
- **Truss Member** — this mode enables to check a separate truss member;
- **Trusses** — this mode enables to perform all the necessary strength and stability checks of the truss members for the most common structural designs;
- **Arches** — this mode enables to perform strength and stability checks of arches from glued laminated timber of a rectangular cross-section;
- **Rafters** — this mode enables to check rafters;
- **Frames** — this mode enables to check frame structures.

ZAPROS

ZAPROS enables you to perform the analysis and check of soil and foundation members according to SNiP 2.02.01-83* and SP 50-101-2004, SNiP 2.02.03-85 and SP 50-102-2003, SP 22.13330.2011 and SP 24.13330.2011, or DBN B.2.1-10-2009. The application also provides reference information commonly used in geotechnical engineering. The design and reference functions implemented in the application are classified into the following groups: **Foundations**, **Piles**, **Pile Field Tests**, **Information**.

The modes of the **Foundation** group enable to perform the following operations: determine the tilt of a rectangular foundation caused by loads acting on it; perform the deformation analysis of subsoil under pad and strip foundations, rectangular in their plane, and rigid slabs; determine the stiffness parameters of the subsoil consisting of a finite number of layers, each layer being linearly deformable and having a constant thickness; calculate the ultimate pressure under the foundation base (design resistance of soil).

The **Piles** group includes two information modes — **Service Factors for Piles** and **Nomenclature for Piles**, together with the following design modes: determination of the load-bearing capacity of a pile subjected to a vertical load; determination of the stability factor of safety of the subsoil, the minimal and maximal bending moment and the shear force in a given section of the pile, and some other pile characteristics; determination of the settlement of a pile subjected to a vertical load.

The **Pile Field Tests** group includes the calculation of the load-bearing capacity of piles obtained from the dynamic testing; determination of the load-bearing capacity of a driven (production) friction pile subjected to compression, obtained from the soil testing with a sample pile; determination of the load-bearing capacity of a driven (production) friction pile subjected to compression, obtained from the soil testing with a probe pile; determination of the load-bearing capacity of a driven (production) friction pile subjected to compression, obtained from the cone penetration test.

Reference modes enable to browse limit values of the relative difference of settlements, tilt, and an average or maximum settlement for various types of structures given in Annex 4 of SNiP 2.02.01-83* (Annex F of SP 50-101-2004, Annexes E, G, L of SP 22.13330.2011 or Annex I of DBN B.2.1-10-2009); provide information on **the design resistance of various kinds of soils** given in Annex 3 of SNiP 2.02.01-83* (Annex A of SP 50-101-2004, Annex C of SP 22.13330.2011, Annex F of DBN B.2.1-10-2009); **the soil properties**, given in Annex 1 of SNiP 2.02.01-83* (Annex D of SP 50-101-2004, Annex B of SP 22.13330.2011, Annex C of DBN B.2.1-10-2009); and **the service factors** from Table 3 of SNiP 2.02.01-83* (Table. 5.2 of SP 50-101-2004, Table 5.4 of SP 22.13330.2011, Table F.7 of DBN B.2.1-10-2009).

1.3. Design and Engineering Programs

Design and engineering programs **COMET** and **Monolit** are used for the development of the design documentation at the stage of the detailed design elaboration.

Monolit

Monolit enables you to design reinforced concrete monolithic ribbed and flat floor slabs. Ribbed floors are formed by a system of slabs and beams bearing on columns and/or walls. The system has been developed in compliance with the current standards (SNiP 2.03.01-84*, GOST 21.501-93, DSTU B A.2.4-7-95, GOST 21.101-97, DSTU B A.2.4-4-99, GOST 7473-94, DSTU B V.2.7-96-2000, GOST 14098-91, SNiP 52-01-2003, SP 52-101-03, SP 63.13330).

The general floor layout is arranged on the orthogonal grid of nodes that have sequential numbering. Nodes are located at the points of intersection of the floor structural members: beams, walls, columns. Slabs are assumed to have uniform thickness over the entire floor, the top of slabs is in the same level as the top of beams. Rectangular beams are divided into secondary beams, which carry the uniformly distributed load from the floor slabs, and main beams, which carry the load from the secondary beams running perpendicular to them and from the floor slabs. Load-bearing walls and/or frame columns serve as the floor supports. The support conditions of the floor bearing on the walls are determined by the wall material: brick walls that provide for free zero-moment support of beams and slabs, and concrete walls that are monolithic with the floor and provide rigid, moment connection of the beams and slabs with the supporting structure. All load-bearing elements of the structure (walls, columns, beams) can be displaced with respect to the axes connecting the nodes of the grid lines.

The program outputs the required set of floor working drawings, which includes: formwork layout with characteristic cross-sections, layouts of the upper and lower reinforcement of a slab (arrangement of the reinforcement meshes), beam and column reinforcement drawings, drawings of welded cages and meshes used for the reinforcement of slabs and beams, specification of details, steel consumption per slab and the overall specification, specifications for slabs and the overall specification. All the necessary notes are given. Reinforcement is completely unified.

All output documents are prepared in formats standard for the majority of printers – A4 and A3. However, they can be printed out using devices of other format, as well as a Plotter. In order to complete the output documents the results can be imported into the DXF formats of the AutoCAD software file.

COMET

COMET is used to check and design the most common types of joints of steel bar structures used in civil and industrial engineering. The approach implemented in the application is based on selecting from a set of parameterized standard structural designs of joints (prototypes). The parameters of the prototypes change during the design process depending on the specified design conditions (material, internal forces etc.), and the selected codes.

The main task of the program is to obtain a design of a joint that complies with the selected codes and design specifications. **COMET** generates a drawing of the joint and provides the data on the resistance of its individual elements (structural members, welds, bolts etc.). The latter enable to assess the quality of the obtained design, and if necessary change some parameters of the structure.

The following joint prototypes are implemented in the program: rigid and pinned beam-to-column joints, nominally pinned column bases, rigid column bases without stiffeners, with stiffeners and wing plates, joints between beams which use bolts and beam splices with end-plates, truss panel points.

Joints between an I-beam and a column are implemented in the program as welded joints and joints with an end-plate with ordinary and high-strength bolts. The joints can be horizontal and inclined, with and without haunches, allowing for the reinforcement of the column and without the reinforcement.

The graphical results can be exported as a DXF (AutoCAD) file.

1.4. Auxiliary Programs

Section Builder

Section Builder enables you to create arbitrary compound sections from rolled steel profiles and plates and calculate their geometric properties necessary for performing the structural analysis. The calculations are performed according to the ordinary rules of the strength of materials, and the moment of inertia at free torsion is determined approximately as a sum of free torsional moments of inertia of profiles that make up the section. A report containing the results of the calculation of the geometric properties is generated.

The results of the calculation of the geometric properties can be exported to **SCAD** and to **Kristall**.

Consul

Consul enables you to create arbitrary sections and calculate their geometric properties on the basis of the solid rods theory.

The following main characteristics can be obtained in the result of the calculation — cross-sectional area, values of the moments of inertia, radii of gyration, section moduli, torsional and sectorial properties, coordinates of the shear center.

The graphical interactive tools enable to create complex sections of an arbitrary shape with holes and include the functions of rounding angles, editing the contour of the section and the coordinates of the vertices, moving a group of the selected vertices. The application enables to import the sections from the **DWG** or **DXF** file formats, and to work with the parametric sections specified by the user.

The calculated geometric properties can be used in **SCAD** when specifying the stiffness properties of the elements.

TONUS

Tonus enables you to create sections and calculate their geometric properties on the basis of the thin-walled rods theory.

The following main characteristics can be obtained in the result of the calculation — cross-sectional area, values of the moments of inertia, radii of gyration, section moduli, torsional and sectorial properties, coordinates of the shear center.

The graphical interactive tools enable to create arbitrary thin-walled sections (including mixed open-closed ones). The application enables to import the sections from the **DWG** or **DXF** file formats, and to work with the parametric sections specified by the user.

The calculated geometric properties can be used in **SCAD** when specifying the stiffness properties of the elements.

SEZAM

Sezam enables you to find a section (a hollow section, an I-beam, a Tee section, or a channel) which approximates an arbitrary section set by the user according to its geometric properties the best. Since all the codes deal only with a certain type of sections, the section obtained in the result of the approximation can be used in the design programs to take into account the elastoplastic stage of work, check for the web buckling, check for the out-of-plane buckling, or to make some other checks.

The initial section can be specified as a file created by **Section Builder**, **Consul** and **Tonus**, by the set of geometrical properties, or as a compound section from the set of prototypes given in the program (e.g. two channels, two I-beams, etc.).

The following properties are approximated for the given section: area, principal moments of inertia, moments of resistance.

West

WeST enables to perform the analyses dealing with the determination of loads and actions on building structures in compliance with the recommendations of SNiP 2.01.07-85* "Loads and Actions" (Fig. 1.4-1), SP 20.13330.2011, SNiP 2.01.07-85 (Belarus), or DBN B.1.2.2-2006.

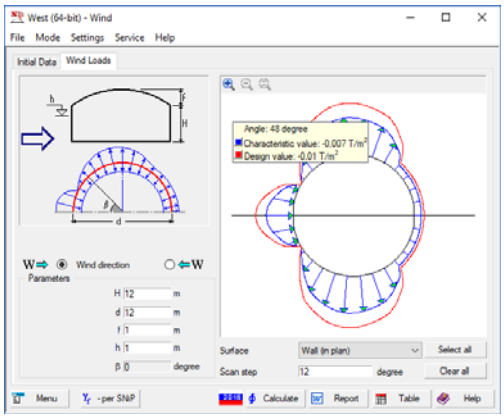


Figure 1.4-1. *Determination of the wind load in* **WeST**

Design modes include:

- **Self-weight** — determination of the load from the self-weight of a multilayer pack consisting from different materials per unit area;
- **Temporary Loads** — determination of values of uniformly distributed temporary loads (full and reduced values) in various rooms in accordance with the instructions of SNiP, SP, and DBN;
- **Wind** — calculation of the static component of wind loads for the structures of various types specified in SNiP, SP, and DBN;
- **Snow** — calculation of the snow loads for the structures of various types specified in SNiP, SP, and DBN;
- **Temperature** — determination of the thermal actions in accordance with SNiP, SP, and DBN.

Slope

Slope enables to determine the factor of safety of a slope. The failure mechanism is assumed to be a slip of a soil mass with respect to a stable part of the slope. The shear resistance along the slip surface is calculated under static conditions. The Coulomb failure criterion is satisfied along the whole surface.

The actual shear stress obtained by the calculation is compared with the ultimate shear resistance, and the result of the comparison is expressed as a factor of safety. The factor of safety of a slope is a minimum factor of safety for all potential slip surfaces which satisfy given restrictions (the restrictions are usually defined by the method of analysis).

The initial data include:

- sizes of the sliding soil mass;
- depth of the tension crack (if the checkbox indicating the presence of a tension crack is checked);
- soil properties;
- position and characteristics of the boreholes;
- loads acting on a certain part of the slope.

Cross

Cross enables to determine the subsoil parameters for the analysis of the foundation structures on the Winkler elastic subgrade based on the modeling of the behavior of a horizontally layered soil deposit. The

geological structure of the soil deposit is assumed to be arbitrary and it is restored based on the data of the geological surveys.

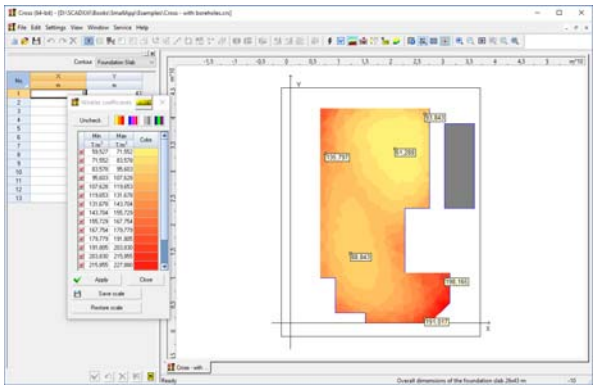


Figure 1.4-2. *Determination of the Winkler coefficient for the foundation slab*

The software considers a construction site where the designed structure is located together with other objects that affect the main structure in the sense that loads upon the soil transferred by these objects may cause settlements of the designed foundation.

The results of the analysis are the values of the Winkler coefficient in any point of the foundation slab (Fig. 1.4-2). **Cross** enables to automatically transfer the geometry of the subgrade from **SCAD** and to return the results of the calculation to the system.

1.5. Service Functions

Besides the design and analysis, design and engineering, and auxiliary programs **SCAD** has a number of service programs such as: **CoCon**, **Kust**, **Section Viewer**, **Formula Calculator**, **Converting Units of Measurement** etc., which are described in the Chapter 30 **Service Functions** of this book.

2. Theoretical Foundations

A person who uses software for the analysis of constructions and who is interested in the stress and strain state is not required to know all mathematical and computational problems in detail. However he needs to have an idea about how mathematical problems are formed and what are the numerical methods to solve them. It is hard to rationally choose an analytical model and correctly evaluate the accuracy of the final results.

L.A. ROSIN. Problems of the theory of elasticity and numerical methods. — St. Petersburg, State Technical University, 1998. p.5

This section provides some short information on the approaches and methods used by **SCAD** for the numerical analysis. The amount of this information is sufficient for the user to better understand further instructions of this manual and to deal with situations that arise as one tries to solve a particular engineering problem. This manual cannot replace the literature on the subject referenced in the text, but it can serve as an introductory guide to these books. Moreover, the considered issues are covered more extensively in the book [11], which can be used as additional documentation for **SCAD**.

2.1. Structure and Its Design Model

General Information

The design analysis of any structure begins with an attempt to determine what is important in the considered case, and what can be neglected. Such a simplification of the problem is always performed, because the real structure with the whole set of its properties can be analyzed only at the expense of certain approximations.

A real structure relieved of all its inessential details and thus represented in an idealized form is called a ***design model***. Some general methods of design modeling have gained a wide spread (a material idealized as a continuum; an assumption of the material homogeneity; reduction of a body geometry to such standard shapes as bars, plates, or shells; external load arrangement, etc.). Other methods of modeling are more particular and are used only in special cases. However, in all cases the selection of a design model is the most important component of the engineering analysis and one of the most characteristic features of the art of engineering (this is indeed the matter of art, not just science!), which defines the professional level of the designer.

As any other art, the art of design modeling can be learned only from practice. Therefore, we shall not dwell on it much longer [1]. However, once the design model has been established (if even roughly), a detailed description of it has to be made in the form appropriate for the numerical analysis. It is this stage of design on which we will give further particular recommendations.

It should be noted that the following decisions have to be made at the early stages of the design modeling: on whether the analysis should be *linear* or *nonlinear*, whether to take the forces of inertia into account and to proceed with a *dynamic* analysis, or to confine yourself to a *static* analysis.

The expected structural behavior is predicted by the designer on the basis of his accumulated experience and engineering intuition. Therefore, all decisions made must be subject to a posterior revision. If time derivatives can be neglected in the governing equations that describe the behavior of a system, then we are dealing with a static problem, and thus with an analysis of the behavior of a motionless system. In the dynamics problems, when the forces of inertia proportional to the accelerations of masses play a significant role, and in the creep problems, when the velocities are taken into account, we deal with the analysis of a moving system.

Nonlinear problems may be caused by the effects arising due to changes in the geometry of the system under the load (*geometric* nonlinearity), lack of proportionality between the stresses and strains

[1] The problem of transition from a structure to the design model and back – from a design model to the structure – is discussed extensively in a wonderful book by V.I. Feodosiev [16], which we recommend to every designer. Very informative and delicate considerations on this question can be also found in a monograph by I.I. Blekhman, A.D. Myshkis, Y.G. Panovko [1].

(*physical* nonlinearity), unilateral constraints being enabled or disabled as the system is loaded or unloaded (*structural* nonlinearity), or the effects defined by the variability of the system structure in the process of its construction (*genetic* nonlinearity).

All these peculiarities of the expected structural behavior affect the selection of the design model, for example, during the determination of the possible degrees of freedom or the arrangement of loads acting upon the system.

A question of particular importance is how to divide the system into finite elements, i.e. standard parts of which (only!) the system will consist.

If the division is too fine, the great amount of elements will increase the computation time, and a lot of computer memory resources will be requested for storage and processing of data. The instability effects of the computation process itself may also arise. If the division is too rough, the accuracy of the results may be lost, especially when a plate or a shell structure is analyzed.

Hardly any general advice can be given on how to select the optimum division of each particular system into finite elements. Existing "estimates of convergence" are of an asymptotic nature (see, for example, [6, 15]), but they are often too abstract to be practically useful for a particular engineering task. Therefore, you should rely mainly upon the accumulated experience and the results of checks performed for the same structure divided into several different sets of finite elements. The approaches when some fragments of the system are analyzed successively with ever finer finite element meshes can be also recommended.

Design Model According to the Slope Deflection Method

Since the computing algorithms implemented by **SCAD** are based on the slope deflection method[1], idealization of the structure should be performed in a way suitable for using this method, i.e.: the system should be represented as a set of bodies of standard types (bars, plates, shells etc.) called finite elements and attached to nodal points.

These finite elements must be previously studied, so that we can determine their reaction to unit nodal displacements and rotations. Finite elements used in **SCAD** are described in Chapter 3.

A node of a design model, according to the slope deflection method, is defined as a rigid body of vanishing sizes. The position of the node in space at the deformation of the system is defined by coordinates of its center and directions of three axes rigidly attached to the node. In other words, the node is considered to be an object with six degrees of freedom: three linear displacements determined as the differences between the node coordinates in the deformed and non-deformed states, and three rotation angles. Other parameters defining the deformed position of the system (additional degrees of freedom) can be also associated with the nodes.

In the slope deflection method, *elements of a system are considered to be connected only to nodes of the design model*. This peculiar feature of design models is not often mentioned in educational and reference books. This approach is approximate because when equivalent forces are reduced to nodal loads, the equilibrium of some types of finite elements (e.g., plates and shells) can be preserved only in an integral sense. Inter-element boundaries between elements other than bars can contain cuts; therefore the issue of so-called consistency (compatibility) of elements should be addressed. Consistent elements ensure that the displacements and their derivatives conform to one another in points located on the opposite edges of a cut. If the elements are inconsistent, additional conditions have to be satisfied to compensate potential possible inconsistent displacements of the edges of the cut.

It should be noted that all elements included in the **SCAD** library are either consistent or compensated against inconsistency by additional conditions.

[1] It means that the common system of governing equations that describe the mechanical behavior of a structural system uses nodal displacements and rotation angles as its unknowns. Some types of finite elements (so-called hybrid elements) internally use force unknowns, but these variables are eliminated before the formation of the common system of governing equations.

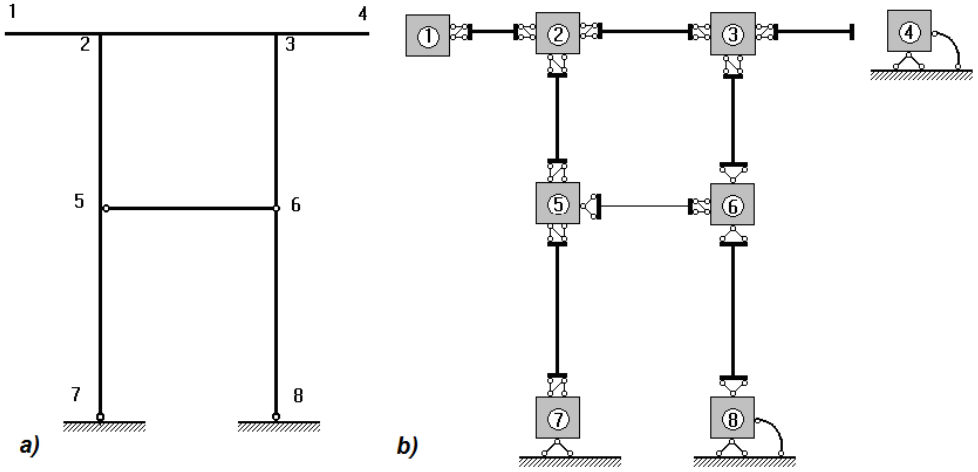


Figure 2.1-1. *Depiction of a design model*: *a* — traditional; *b* — detailed

The conditions of junction between elements and nodes are not always shown in traditional methods of visualization of design models. For example, the design model shown in Fig. 2.1-1, *a* in a traditional manner can give one an idea that its elements are directly connected to one another, while a more detailed picture of Fig. 2.1-1, *b* helps one avoid this deduction. Note also that the detailed picture shows other peculiarities of this model. In particular, one can see the possibility for specifying the same kinematic conditions with different sets of constraints (compare nodes 1 and 4, or 7 and 8). However, this technique is not always harmless. For example, from the standpoint of kinematic properties of the problem, two versions of a design model shown in Fig. 2.1-2, are equivalent (a beam clamped at its left end and simply supported at the right end).

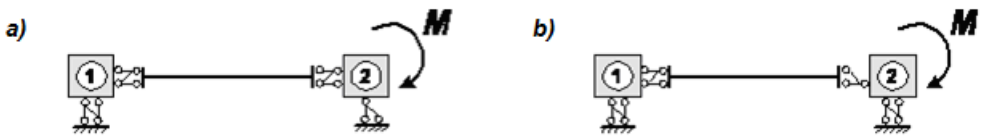


Figure 2.1-2. *Two ways of representing the same design model*

However from the standpoint of the force specification these ways are different: the model in Fig. 2.1-2, *a* transfers the moment to the bar, thus node 2 of this model will have a slope, while the model in Fig. 2.1-2, *b* transfers no moment and thus node 2 will have a zero slope. In order for the model in Fig. 2.1-2, *b* to have the bending moment in its bar, one should treat this moment as applied to the bar in a cross-section near the node rather than to the node itself. On the other hand, using the model in Fig. 2.1-2, *a* you can calculate the rotation angle at the node 2, which is equal to the slope of the rightmost section of the bar; and the model in Fig. 2.1-2, *b* does not enable to calculate the slope of the end section because **SCAD** is capable of outputting only nodal displacements and rotation angles.

The whole design model is assumed to consist solely of elements of a certain predefined type. The list of element types used by **SCAD** can be altered or supplemented, but each particular version of the system uses a certain set of elements called a finite element library (FEL), from which the “elementary particles” for creating a design model are selected.

Finally, it should be noted that all the nodes and elements of a design model are numbered. Numbers assigned to them should only be treated as names for the convenience of reference. Using the numbers we can, for instance, indicate a node where a certain load is applied, list nodes a particular element is attached to, make a list of elements joined in a certain node (which is called a “star of elements” at the node) or being in a certain specified group. The numbering does not perform any other functions, and it almost does not affect the computation time, because the system is capable of optimizing the stiffness matrix profile. However, the results of an analysis are usually output in the order of numbering of the nodes (displacements) or elements (internal forces and moments). We, therefore, can not say that nothing at all depends on the numbering.

Principal and Auxiliary Unknowns

The principal system of the slope deflection method is formed in a usual way, by imposing constraints upon each node that prevent the node from any displacements. The conditions that the forces in those constraints are equal to zero are the governing equations of equilibrium, and displacements of the said constraints are the principal unknowns of the slope deflection method.

All six displacements may take place in a node of a common spatial structure:

1 — a linear displacement along the X axis;
2 — the same along the Y axis;
3 — the same along the Z axis;
4 — a rotation angle (slope) with its vector along the X axis (a rotation about the X axis);
5 — the same along the Y axis (a rotation about the Y axis);
6 — the same along the Z axis (a rotation about the Z axis).

The numbering of the displacements (degrees of freedom) given above will be used later without further notes; the documentation also uses the symbols: X, Y, Z, U_X, U_Y and U_Z to denote the respective linear displacements and rotation angles. If one of the displacements in a node does not affect the stress state of all the elements joined in this node (e.g., the rotation angles of a node joining only bars with hinges at their ends, which is a common case in the analysis of trusses), then this displacement is not included in the set of principal unknowns.

The whole system may even happen to have this property, and the same reduced set of unknown displacements will be then present in each node of it, or, in other words, some of the displacements are absent from the degrees of freedom of all the nodes of the system. This property, or feature, of the system can be denoted in a special way so that we do not have to use some of the displacements. Here are some examples:

- a flat truss located in the XOZ plane — we can indicate that the displacements along the Y axis and all the rotation angles (U_X, U_Y, U_Z) will not be considered;
- a plate in the XOY plane — the displacements along the X and Y axes, and the rotation angle U_Z are excluded.

In some cases a node of the design model can be assigned additional degrees of freedom (auxiliary unknowns), which will no longer be equivalent to the components of linear or angular displacements of a node as an infinitesimally small rigid body in its central point. The second mixed derivative (torsion) of the deformed plate surface, or components of transversal shear and reduction in laminated piecewise-heterogeneous shallow shells can be the examples of an auxiliary unknown.

External and Internal Constraints

Limitations (constraints) can be imposed on some of the nodal displacements due to the structural peculiarities. We will say that a *mono-constraint* is imposed upon a system if it restrains one component of the nodal displacement (e.g., $X = 0$ or $U_Y = 0$). If a constraint upon the system makes a linear combination of the nodal displacement components equal to zero, we will refer to this constraint as a *poly-constraint* [11]. A mono-constraint is always an external one, because it limits the displacements as an external action. A poly-constraint imposed on the components of nodal displacements of the same node is also an external one, but the restrained displacement is directed at a certain angle to the coordinate axes rather than being coincident with one of them (e.g., the poly-constraint $X_i\cos\varphi + Y_i\sin\varphi = 0$ is directed at the angle φ to the coordinate axis X). Other poly-constraints involving the displacement components of different nodes are internal. They are usually caused by the presence of rigid elements in the system (in its design model).

It should be noted that when selecting a design model, you should make sure that rigid elements which model the internal constraints do not form a statically indeterminate system such as a closed contour. Otherwise the system would be degenerate and the distribution of stresses in it would be ambiguous.

Another, somewhat opposite, danger is missing a constraint when creating a design model, i.e. not restraining a displacement that leads to a dimensional instability. During the analysis **SCAD** performs the check of the dimensional stability and modifies the unstable models by imposing additional constraints. However, the user must always keep in mind that this instability might be the result of his negligence, and do not rely on the "service" of correcting the detected mistakes performed by the system. In particular, this

modification is correct only if the reactions in the additional constraints automatically added to the system are equal to zero, and no load is applied in the direction of this displacement.

An analysis of a system with mono-constraints is not complicated: the respective nodal displacements are simply excluded from the set of the principal unknowns. This simplicity causes a motivation to describe those external constraints that restrain displacements or rotations not directed along the coordinate axes as mono-constraints.

Another approach is to try to restrain any kind of displacement by adding some elements with sufficiently large stiffness, but it is approximate and might lead to a noticeable loss of accuracy. The only advantage of this approach is that it enables to determine the reaction of the added constraint. Wider capabilities arise from the use of special "null elements" which, among other things, enable you to perform the analysis for a specified displacement [10].

Junction between Elements and Nodes

Nodes (located in the end sections of bars) of elements joined to the nodes of the system can be located at some distance from the center of the model node, i.e., a junction eccentricity may take place. We will assume that in all element types where this eccentricity is allowed, the node (end section) of an element will be connected to the center of the model node by a rigid insert, and this insert will be a part of the finite element (see the description of bar-type finite elements in Chapter 3 **Finite Element Library**).

Obviously, the presence of the said rigid insert will make the displacement of the element end section unequal to that of the model node (the former depends also on the nodal rotations), while the rotation angles of the node and of the end section will be equal.

The difference between the displacements (rotation angles) of a node and the element end section connected to the model node may be also caused by the peculiarities of the junction design. Assuming that an element can be connected by its nodes to the centers of the model nodes through six constraints, each one restraining one of six possible mutual displacements, we can also imagine the absence of any of these six constraints. The absence of one angular constraint corresponds to a cylindrical hinge; the absence of all three mutual angular constraints – to a spherical hinge; the absence of a linear constraint – to a "slider", etc. To be brief, we will further refer to all cases of this kind as "hinges", though it is not very accurate. It should be noted that the use of various junctions between the elements and the nodes, as well as constraints imposed on the nodal displacements, enables to diversify our design models. This fact is illustrated by Fig. 2.1-1. We recommend you to compare equal though represented in different ways pairs of nodes 1 and 4, or 7 and 8.

It should be noted that the incomplete equivalence between the element and the model nodal displacements, as well as the eccentric junction, is allowable not for every element type; one should refer to finite element definitions to find out whether those are possible. Limitations, if there are any, can be circumvented using artificial tricks (by introducing additional nodes, very compliant constraining elements etc.).

Groups, Substructures, Super-Elements

It is often convenient to represent a design model as a complex of several different fragments, each one being a meaningful entity. In the **SCAD** environment, named structural fragments are referred to as *groups*. A group can be formed from the functionally related parts (e.g., a frame of one of the storeys in a multi-storey building), or from the parts with a common structural system (a floor, a wall), or from the structural part with a different geometry (e.g., one face of an angular shell). **SCAD** allows you to create groups out of any components of a design model (nodes, elements, loads) and their combinations.

Further, when talking about a group, we will mean a part of the model extracted from the whole system by a criterion specified by the designer. However, the software itself does not perform any actions on a group; it is the user that has to define all the actions to be performed during the analysis (e.g., perform a unification, select rebars, print the results etc.).

SCAD enables to use the fragments represented as individual design models when performing certain actions (chiefly in the course of entering information). Such fragments will be called *substructures*. A substructure possesses all features of a design model, it consists of nodes and elements, includes constraints, hinges etc. When creating a model, its substructure is treated like a single entity, determining its place in a system (main model) by merging some nodes of the subsystem, which play the part of nodes of a conventional finite element, with nodes of the main model.

A substructure used not only for inputting the initial data but also for organizing the computation process is called a *super-element*. The super-element modeling should be applied in cases when the same super-element is included in the system several times. Results of calculations performed for the first super-element of the series of similar ones will then be used for the rest of this series, which will considerably reduce the number of calculations.

Loads and Actions

The interaction of a system with its environment is usually represented as a set of loads or actions applied to the nodes of the system (*nodal loads*) or to the interior points of its finite elements (*local loads*). The local loads can be forces or moments, either concentrated or distributed over lines, areas, or volumes. The loading of the system can be sometimes represented as a thermal action on its elements or given displacements in nodes, i.e. these actions are kinematic rather than force factors.

If the actions vary with time, they cause accelerations of the system masses which in turn give rise to inertial forces. In cases when the inertial forces cannot be neglected, the action is referred to as *dynamic*; keep in mind that classifying an action as static or dynamic should be based upon *inertial properties of the system* just as well as on the properties of the action.

Both *characteristic and design load values* are used in the engineering practice. They are related by the safety factors for loads. Generally speaking, the characteristic and the design values are used in different kinds of calculations: the former are used to check the serviceability limit state, and the latter – to check the ultimate limit state. **SCAD** assumes by default that the loads are given by their design values.

A set of loads and actions applied to the system simultaneously and considered together is called its *loading*. Sometimes one loading comprises only those simultaneously applied loads that have a common origin (e.g., only pressure caused by wheels of a single overhead travelling crane, or snow load on the roof of a single span in a multi-span building) or have some other common properties (e.g., the same safety factor for load set by the design codes). If afterwards a combined effect of several loadings (their sum with some coefficients) has to be considered, it is called a *combination of loadings*, and the coefficients will be respectively referred to as the coefficients of this combination.

The fact that loadings may form various combinations in different moments of time, and the number of these combinations can be rather big, leads to a non-trivial problem of finding such combinations of external actions that result in the most disadvantageous consequences for the considered structural element or its section. These combinations are called the *design combinations of forces* (**DCF**). When solving this problem, you should remember about the logical relation between loadings, defined by the nature of loads applied to the system or by the regulations of the design codes. Such logical relationships make it sometimes impossible to apply some loads simultaneously (the incompatibility of loads), such as, for instance, the wind pressure caused by winds of different directions. In other cases when considering a certain loading it might be necessary to take into account another loading (e.g., if a friction force is considered, then the normal pressure must be taken into account as well), though the inverse relation of this kind might not take place (the normal pressure can be considered independently without the friction force).

It should be noted that the use of the *combinations of loadings* and the determination of the DCF are based on the superposition principle, and therefore, they make sense only for the *linear systems*. If there is any kind of nonlinearity involved, all the loads acting on the system must be analyzed within a single loading.

2.2. Basic Relationships of the Finite Element Method

Linear Static Analysis

SCAD is based on the finite element method (**FEM**) in the form of slope deflection method. This form has been selected due to its simple algorithmic implementation and physical interpretation; possibility of creating unified methods for the construction of stiffness matrices and load vectors for various types of finite elements; possibility of taking into account any types of boundary conditions and complex geometry of the designed structure. Detailed descriptions of the method with thorough proofs can be found in a lot of books (see for example, [6, 15]). This section only briefly covers the basic design relationships.

The stress and strain state of each material point x of a finite element with the volume V and the surface area S, can be described by the vectors of stress $\boldsymbol{\sigma}(x)$ and strain $\varepsilon(x)$. In the theory of linear elasticity the latter are expressed through the vector of displacements $\boldsymbol{u}(x)$ as follows:

$$\boldsymbol{\sigma} = \boldsymbol{M}\boldsymbol{\varepsilon},\ \ \boldsymbol{\varepsilon} = \boldsymbol{B}\boldsymbol{u}, \tag{2.2.1}$$

where $\boldsymbol{B}$ is a linear matrix differential operator; $\boldsymbol{M}$ is a symmetric positive definite elasticity matrix of Hooke's law that depends only on stiffness properties of the structural material.

The full potential energy of the element is determined by the following formula:

$$\Pi^{(e)} = \frac{1}{2}\int_v \boldsymbol{\varepsilon}^T \boldsymbol{M}\boldsymbol{\varepsilon}\, dV - \int_v \boldsymbol{u}^T \boldsymbol{p} dV - \int_s \boldsymbol{u}^T \boldsymbol{q}\, dS, \tag{2.2.2}$$

where $\boldsymbol{p}$ and $\boldsymbol{q}$ are vectors of volume and surface forces, respectively.

The displacements $\boldsymbol{u}(\boldsymbol{x})$ of any point of the considered element can be approximately expressed through the unknown nodal displacements $\boldsymbol{Z}$:

$$\boldsymbol{u}(x) = \sum_i \varphi_i(x)\boldsymbol{Z}_i = \boldsymbol{\Phi}(x)\boldsymbol{Z}_e\ , \tag{2.2.3}$$

where $\varphi_i(x)$ are the interpolation functions usually referred to as shape functions, which should meet the smoothness requirements in order to ensure the convergence of the method [15]; $\boldsymbol{\Phi}$(x) is the matrix of interpolation functions; $\boldsymbol{Z}_e$ is the vector of all unknown nodal displacements of the considered element (index "e").

Substituting (2.2.1) and (2.2.3) to (2.2.2) we obtain

$$\Pi^{(e)} = 1/2\, \boldsymbol{Z}_e^T \left(\int_v (\boldsymbol{B}\boldsymbol{\Phi})^T \boldsymbol{M}\boldsymbol{B}\boldsymbol{\Phi}\ \ dV \right) \boldsymbol{Z}_e - \left(\int_v \boldsymbol{p}^T \boldsymbol{\Phi}\ \ dV + \int_s \boldsymbol{q}^T \boldsymbol{\Phi}\ \ dS \right) \boldsymbol{Z}_e \tag{2.2.4}$$

The expression (2.2.4) can be represented in the following form:

$$\Pi^{(e)} = 1/2\, \boldsymbol{Z}_e^T \boldsymbol{K}_{(e)} \boldsymbol{Z}_e - \boldsymbol{f}_e^T \boldsymbol{Z}_e, \tag{2.2.5}$$

where $\boldsymbol{K}_{(e)} = \int_v (\boldsymbol{B}\boldsymbol{\Phi})^T \boldsymbol{M}\boldsymbol{B}\boldsymbol{\Phi} dV$ is the stiffness matrix of the element;

$\boldsymbol{f}_e^T = \int_v \boldsymbol{p}^T \boldsymbol{\Phi} dV + \int_s \boldsymbol{q}^T \boldsymbol{\Phi} dS$ is the vector of reduced nodal forces.

The full potential energy of a system is a sum of the potential energies of all its elements:

$$\Pi = \sum_e \Pi^{(e)}\ , \tag{2.2.6}$$

and its minimization produces the system of governing equations of FEM:

$$\boldsymbol{K}\boldsymbol{Z} = \boldsymbol{f} \tag{2.2.7}$$

with the global stiffness matrix $\boldsymbol{K}$ and the vector of nodal forces $\boldsymbol{f}$, obtained by summing the respective members of the stiffness matrices $\boldsymbol{K}_{(e)}$ and the vectors $\boldsymbol{f}_{(e)}$ of particular finite elements, which is an advantage of the considered approach.

There are known conditions of convergence and error estimates for FEM in the form of slope deflection method. The conditions of convergence include the linear independence and the completeness of the system of basis functions, as well as their consistency (conformity) or other conditions compensating the inconsistency.

There are also some easily verified conditions for establishing the completeness of the basis functions, their consistency, or whether the conditions compensating the inconsistency are satisfied. In the case of polynomial approximations, these conditions come in the form of equalities which must be satisfied by the basis functions at each finite element. This theoretical basis enables us to check the correctness of the application of the known finite elements, to develop the principles for constructing new consistent and inconsistent elements and to obtain their error estimates as well.

SCAD finite element library contains a lot of elements for modeling the behavior of various types of structures. There are well-known bar elements, triangular and quadrangular elements for the analysis of the plane stress state, plates, shells, 3D elements such as tetrahedron, parallelepiped, and trihedral prism. The library includes the following elements: inconsistent triangular and rectangular elements of isotropic and orthotropic plates and shells, plates on the elastic subgrade, laminated plates and shells; consistent

triangular and quadrangular elements for the analysis of the plane stress state, plates and shells with optional nodes on their sides, all constructed by the sub-area method. These elements are based on the elements for the analysis of the plane stress state with two degrees of freedom in a node, and plates with three degrees of freedom.

The library contains isoparametric elements for the analysis of the plane stress state and for solving spatial elastic problems; one- and two-dimensional (quadrangular and triangular) axisymmetric elements. Moreover, there are various special elements for modeling the constraints of finite rigidity, elastic compliance between nodes, null-elements of various types, elements specified by a numerical stiffness matrix. All finite elements contained in the library are theoretically justified and have error estimates of energy and displacements. The integral force error can be estimated by a value proportional to h^{τ}, where h is the maximum size of finite elements, $\tau = 2$ for rectangular plate elements, $\tau = 1$ for all other elements. The displacement error can be estimated by a value proportional to h^{τ}, where $\tau = 4$ for consistent rectangular and quadrangular plate elements, and $\tau = 2$ for all other elements. The possibility of performing the analysis of curvilinear bars with rectilinear elements and of any shells with triangular and rectangular (for cylindrical shells) plane shell elements is also theoretically justified. The energy and displacement error in this case can be estimated by a value proportional to h.

Additional Constraints

If the displacements $\boldsymbol{Z}$ are restrained by the constraint conditions

$$\boldsymbol{CZ} = \boldsymbol{0}, \tag{2.2.8}$$

then only some components of the vector $\boldsymbol{Z}$ can be varied freely, and in order to take these constraints into account, it was suggested in paper [10] to use a modified Lagrangian function:

$$\Pi_{LM} = \tfrac{1}{2}\, \boldsymbol{Z}^T(\boldsymbol{K} + \boldsymbol{C}^T\boldsymbol{D}_0\boldsymbol{C})\boldsymbol{Z} - \boldsymbol{Z}^T\boldsymbol{f} + \boldsymbol{\rho}^T\, \boldsymbol{CZ}, \tag{2.2.9}$$

where $\boldsymbol{D}_0$ is a symmetric positive definite matrix, and $\boldsymbol{\rho}$ is a vector of Lagrange multipliers (reactions in the constraints).

Stationarity conditions of the functional (2.2.9) yield a system of equations

$$\begin{aligned} &\boldsymbol{KZ} + (\boldsymbol{C}^T\boldsymbol{D}_0\boldsymbol{C})\, \boldsymbol{Z} + \boldsymbol{C}^T\boldsymbol{\rho} = \boldsymbol{f}; \\ &\boldsymbol{CZ} = \boldsymbol{0}. \end{aligned} \tag{2.2.10}$$

It should be noted that the matrix $\boldsymbol{C}$ must contain only linearly independent rows; otherwise (if there are duplicated constraints) the system may happen to contain a statically indeterminate substructure made up from rigid elements, which results in a degenerate problem. In all other cases the system of equations (2.2.10) has a nonzero determinant, including the case of a degenerate matrix $\boldsymbol{K}$ (the structure without additional constraints is unstable) with its excess not exceeding the rank of the matrix $\boldsymbol{C}$.

Adding the product $\boldsymbol{C}^T\boldsymbol{D}_0\boldsymbol{C}$ to the stiffness matrix $\boldsymbol{K}$ corresponds to the inclusion of an elastic element with special properties ("null element") into the structure. It is inserted in parallel to an infinitely rigid constraint and therefore does not distort the results of analysis. Possible methods for constructing such "null elements", some of which have been used in the development of **SCAD**, are considered in detail in [10].

Rigid Bodies

One of the possible types of poly-constraints is the case when the displacements of some nodes of the system are connected to each other as in the case of their connection by a rigid body. In the **SCAD** software the rigid body is treated as a specific type of finite element. Its relationship to finite elements can be noticed at least from the fact that the rigid body unites nodes attached to it. Roughly, a rigid body can be considered as a limiting case of a finite element as the rigidity properties of this element approach the infinity.

Rigid body can be represented graphically as a "spider" consisting of one master node and a number of slave nodes – Fig. 2.2-1. The connection can be full or partial.

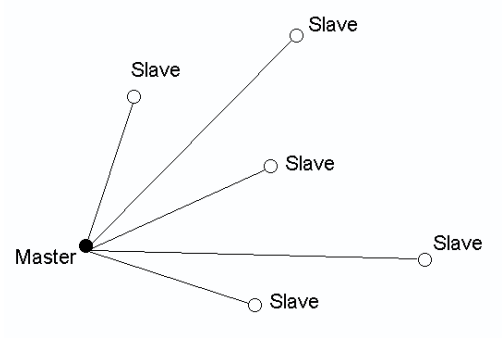


Figure 2.2-1

We have to distinguish between linear and angular components of the vector of nodal displacements $\boldsymbol{Z}$. The first ones will be denoted as $\boldsymbol{u}$, and the second ones as $\boldsymbol{\theta}$.

When the connection is full all the displacements of the slave nodes are expressed through the displacements of the master node on the basis of the kinematics theorems for a rigid body:

$$\begin{pmatrix} \boldsymbol{u}_S \\ \boldsymbol{\theta}_S \end{pmatrix} = \begin{pmatrix} \boldsymbol{u}_M + \boldsymbol{\theta}_M \times \boldsymbol{\rho}_{M-S} \\ \boldsymbol{\theta}_M \end{pmatrix}, \qquad (2.2.11)$$

where $\boldsymbol{u}, \boldsymbol{\theta}$ are vectors of linear displacements and angles of rotation of the node in the global coordinate system, and the indices M and S stand for the master and slave nodes respectively;

$\boldsymbol{\rho}_{M-S}$ is the radius vector connecting the master node to the slave one.

In the case of partial connection the expression (2.2.11) covers only those degrees of freedom which satisfy the kinematic relationships of a rigid body. Other displacements and angles of rotation of each of the slave nodes remain independent and are not expressed through the displacements of the master node. Such connection can be represented as a connection in two stages. The full connection of displacements and rotation angles of all slave nodes with displacements and angles of rotation of the master node is performed during the first stage, and then independent degrees of freedom of the slave nodes are disconnected.

Since the equations (2.2.11) include the angle of rotation of the master-node, when using the rigid bodies only the design models with the degrees of freedom including the angles of rotation (type 2, type 3 and type 5) can be used. If there are no angles of rotation in other nodes due to the selection of the respective types of finite elements, it is recommended to impose the missing constraints.

The connection codes are introduced for the convenience of specifying the connection types. The connection codes for the spatial design model are as follows X, Y, Z, U_X, U_Y, U_Z.

Each of these parameters is set to f or b. The value b (blocked) means that the connection of displacements corresponding to the given degree of freedom (in the global coordinate system) is performed, and the value f (free) means that it is not performed.

There are the following possible options of a set of nodal displacements in the connection code:

- spatial design model – connection codes X, Y, Z, U_X, U_Y, U_Z;
- two-dimensional frame – connection codes X, Z, U_Y (this type of a rigid body imposes displacements on the slave nodes in its own plane, such a rigid body can be called a *disc*);
- bendable plate located in the XOY plane – connection codes Z, U_X, U_Y. (this type of a rigid body, which can be called a *stamp*, connects the displacements of the nodes belonging to it at their exit from the plane of the rigid body).

It should be noted that, on the one hand, the rigid bodies (and all the poly-constraints) are an extremely powerful tool for creating the design models, which enables the user to express their individual creativity in the most unusual situations. On the other hand, this tool is not safe – even the experienced designers can get into a well hidden trap, which is difficult to detect. One of such traps is the condition of linear independence. This means that the design model should not contain a statically indeterminate subsystem consisting entirely of rigid elements. Otherwise, the distribution of forces in the system is not unique.

Let us consider some typical examples of using the rigid bodies. Figure 2.2-2,*a* shows the structure of a strong slab reinforced by wide ribs. Application of the traditional approaches for modeling this structure with the help of rigid inserts will make the bending of the slab in the span between the reinforcing ribs considerably greater than in the case when the reduction of the slab span due to the rib width is taken into account.

Using the rigid bodies in the model (they are limited by the dashed line in the figure) enables to join the bar with the necessary eccentricity to the nodes on the middle surface of the plate, and to provide the absence of distortion of the section at the junction with the plate, which prevents its bending in this area. It

should be noted that in this case the spatial rigid body can be used, despite the fact that all its nodes are in the same plane.

The absence of distortion of the cross-section contour in the models made from the thin-walled bars is often provided by the introduction of diaphragms or similar stiffening elements. If these elements are not loaded and their stress state is of no importance to the designer, the diaphragms can be modeled by rigid bodies which do not allow deformations in their plane.

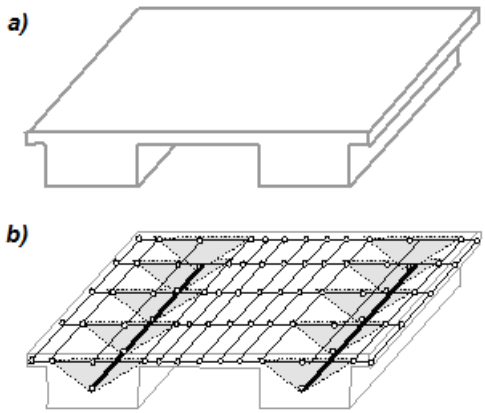


Figure 2.2-2. Modeling of a wide rib by the rigid bodies

An example connection of plates of different thicknesses with one of the surfaces aligned is shown in the Figure 2.2-3. It is advisable to use a rigid body with pairs of nodes along the line of thickness variation adjacent to different middle surfaces of the plates.

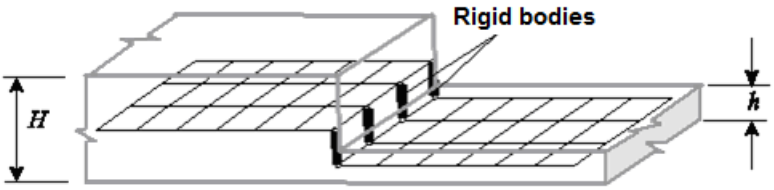


Figure 2.2-3. Modeling of a non-central thickness jump

It should be noted that simply merging the displacements of the upper and lower nodes will not provide an adequate description of the performance features of such a connection since the tangential displacement of the upper node is determined not only by the tangential displacement of the lower node, but also by its angle of rotation.

This approach can be also applied in the case when the step between the upper and lower plates is formed by the beam (see Fig. 2.2-4). Three-node rigid bodies with a bar element modeling a beam attached to their middle node are used here.

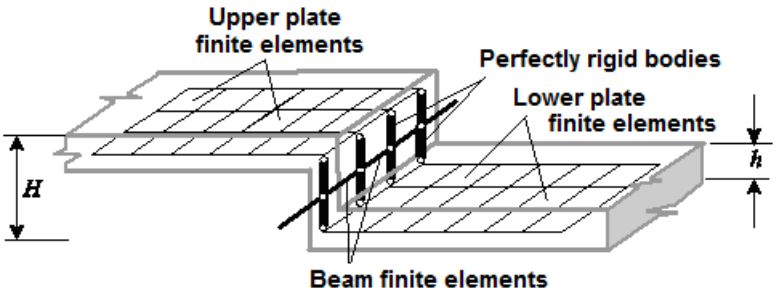


Figure 2.2-4. Modeling of a step

Returning to the connection of plates of different thicknesses with one of the surfaces aligned, it should be noted that when the rigid bodies are used, the nodes of the parts of the plate with different

thicknesses are located at different levels, which may not always be convenient. You can place all nodes at the same level and use the “Move the plate midplane” option, which will also lead to the desired effect.

A typical example of the application of the rigid bodies is modeling of the joint area of monolithic columns and floor slabs (see Fig. 2.2-5). Dimensions of a rigid body in this case correspond to the dimensions of the column cross-section. It should be noted that the rigid bodies can be automatically entered into the joint area in the **FORUM** preprocessor.

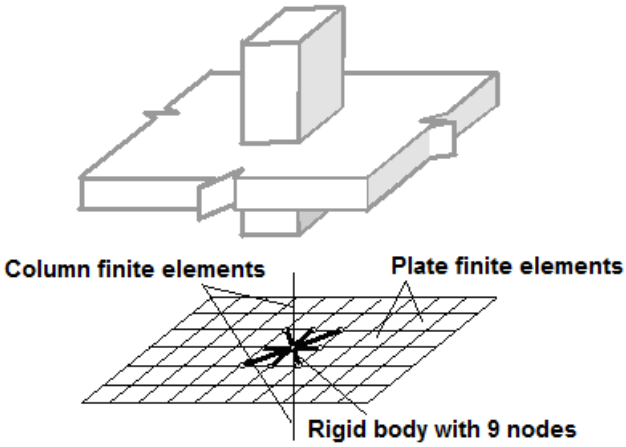


Figure 2.2-5. Modeling of the slab-to-column joint area

Dynamical Analysis

If the load applied to the system changes with time, i.e. $\boldsymbol{f} = \boldsymbol{f}(t)$, then we have to assume the stresses and displacements also to be functions of time, which may force us to introduce velocities $d\boldsymbol{Z}/dt$ and accelerations $d^2\boldsymbol{Z}/dt^2$ in the consideration. If inertia forces that arise $\boldsymbol{J}(t) = \boldsymbol{M}\, d^2\boldsymbol{Z}/dt^2$ cannot be treated as negligibly small comparing to the loads upon the system and to the elasticity forces, then they should be taken into account in the equilibrium equations which will necessarily become differential equations of motion:

$$\boldsymbol{M}\frac{d^2\boldsymbol{Z}}{dt^2} + \boldsymbol{K}\boldsymbol{Z}(t) = \boldsymbol{f}(t)\,. \tag{2.2.12}$$

The mass matrix of the system is determined as a sum of mass matrices of its elements plus masses related to the inertial properties of the structural parts not included in the design model.

In **SCAD** a mass matrix of an element can be a consistent matrix or a matrix of concentrated masses:

- **consistent mass matrix** of an element is calculated using the shape functions of an element (this matrix enables to take into account the mass moments of inertia correctly). These shape functions are the same as those used when calculating a stiffness matrix, therefore it is called "consistent".

- **in the matrix of concentrated masses** the mass of each element is distributed between its nodes. In other words, concentrated masses in the nodes of the element make up the mass of the element.

Therefore the matrix of concentrated masses is diagonal all elements of which except for the diagonal ones are equal to zero. This approach is approximate, but when the finite element mesh is dense enough it gives good results.

The problem of determining characteristics of natural oscillations of the system (modal analysis) is to find conditions in which the unloaded system performs harmonic oscillations according to the law

$$\boldsymbol{Z}(t) = \boldsymbol{\Psi}\sin(\omega t + \varphi). \tag{2.2.13}$$

In expression (2.2.13) vector $\boldsymbol{\Psi}$ describes the mode of the natural oscillations (relations between the displacements of nodes); ω — their frequency; φ — the initial phase. Substitution of (2.2.13) to (2.2.12) seeing that $\boldsymbol{f}(t) = 0$ will give an equation for natural oscillations

$$(\boldsymbol{K} - \omega^2 \boldsymbol{M})\ \boldsymbol{\Psi} = 0, \tag{2.2.14}$$

the nontrivial solution of which exists only when values ω_i (i = 1,..., n), called natural frequencies or eigenvalues, turn the determinant of matrix $[\boldsymbol{K}-\omega^2\boldsymbol{M}]$ into zero. The respective modes of natural oscillations Ψ_i (eigenvectors) can be calculated only up to an arbitrary multiplier. This multiplier can be, for instance, assigned in such a way that the maximum component of vector $\boldsymbol{\Psi}_i$ is equal to one. The property of orthogonality of eigenvectors with respect to both the mass matrix and the stiffness matrix is also worth noticing, i.e.

$$\boldsymbol{\Psi}_i^T \mathbf{M} \boldsymbol{\Psi}_j = 0 \quad \text{and} \quad \boldsymbol{\Psi}_i^T \boldsymbol{K} \boldsymbol{\Psi}_j = 0 \quad \text{at } i \neq j. \tag{2.2.15}$$

In the dynamical analysis, the number of components of vector $\boldsymbol{Z}$ where inertia forces are involved (the number of *dynamical degrees of freedom*) can be often much less than that of the static components. The typical example is presented by nodal slopes which, usually, are much less influential dynamically than the respective linear displacements. In the **SCAD** environment inertial moments corresponding to the nodal slopes and other inertial characteristics can be neglected, but this has to be specified by the user when formulating a dynamical problem. If some inertial components of the load are not taken into consideration, then we can divide vector Ψ into sub-vector Ψ_0 for which the inertia forces are zero and sub-vector Ψ_I where the inertia forces are involved, and write system (2.2.14) in the form

$$\boldsymbol{K}_{00}\ \boldsymbol{\Psi}_0 + \boldsymbol{K}_{01}\ \boldsymbol{\Psi}_1 = \boldsymbol{0};$$

$$\boldsymbol{K}_{10}\ \boldsymbol{\Psi}_0 + \boldsymbol{K}_{11}\ \boldsymbol{\Psi}_1 = \omega^2 \boldsymbol{M}_1 \boldsymbol{\Psi}_1. \tag{2.2.16}$$

Sub-vector $\boldsymbol{\Psi}_0$, can be excluded from this system. After this procedure of "static condensation", the dimension of the modal analysis problem decreases dramatically, and the problem becomes stated as follows

$$\left[\left(\boldsymbol{K}_{11} - \boldsymbol{K}_{10}\boldsymbol{K}_{00}^{-1}\boldsymbol{K}_{01}\right)^{-1} \boldsymbol{M}_1 - \boldsymbol{\lambda}^2 \boldsymbol{I}\right]\boldsymbol{\psi}_1 = 0\,, \tag{2.2.17}$$

where $\boldsymbol{I}$ is a unit matrix, and $\lambda = 1/\omega$.

The modal analysis produces eigenvalues λ_i and eigenvectors $\boldsymbol{\Psi}_I$ of the problem (2.2.17), which can be used to determine the circular frequency $\omega = 1/\lambda$ (radians per second), cyclic frequency $\theta = \omega/2\pi$ (Hz), and period $T = 1/\theta$.

Since the natural oscillation modes are orthogonal, the solution of any dynamical problem represented as an expansion

$$\boldsymbol{Z}(\mathrm{t}) = \sum_i y_i(\mathrm{t})\ \boldsymbol{\Psi}_\mathrm{i} \tag{2.2.18}$$

leads to the splitting of the system of differential equations (2.2.12) into independent equations with respect to generalized coordinates y_i(t). These equations usually contain one more term proportional to velocity, to take into account the viscous resistance to the motion. Thus the equations become

$$\frac{d^2 y_i}{dt^2} + 2\xi_\mathrm{i}\,\omega_\mathrm{i}\,\frac{dy_i}{dt} + \omega_\mathrm{i}^2 y_i = P_\mathrm{i}(\mathrm{t})/M_\mathrm{i}. \tag{2.2.19}$$

Generalized forces

$$P_\mathrm{i}(\mathrm{t}) = \omega^2\ \boldsymbol{\Psi}^T_i \boldsymbol{f}(\mathrm{t}); \tag{2.2.20}$$

masses

$$M_i = \Psi_i^T\, M \Psi_i \tag{2.2.21}$$

and viscosity coefficients ξ_i together with initial conditions y^0_i and y'_i, derived from $\boldsymbol{Z}^0 = \boldsymbol{Z}(0)$ and $\boldsymbol{Z}' = d\boldsymbol{Z}(0)/dt$ by formulae:

$$y_\mathrm{i}^0 = \boldsymbol{\Psi}_i^T \boldsymbol{M}\boldsymbol{Z}^0\,; \quad y_i' = \boldsymbol{\Psi}_i^T \boldsymbol{M}\boldsymbol{Z}'\,, \tag{2.2.22}$$

define the complete solution of the problem. This solution can be written as the expression:

$$y_i = \exp[-\xi_i\,\omega_i\,t]\ \{[(y^0_i\,\xi_i\,\omega_i + y^1_i)/\,\omega_{Di}]\ \sin\omega_{Di}t\ +\ y^0_i\} +$$

$$+ (1/\omega_{\mathrm{Di}}M_\mathrm{i}) \int_0^t P_\mathrm{i}(\mathrm{t})\ \exp[-\,\xi_\mathrm{i}\,\omega_\mathrm{i}\,(\mathrm{t}-\tau)]\ \sin\omega_{\mathrm{Di}}(\mathrm{t}-\tau)\ \mathrm{d}\tau\,, \tag{2.2.23}$$

where the first term takes into account the initial conditions and the second one is called a Duhamel integral.

The frequency of damped oscillations included in the expression (2.2.23):

$$\omega_{Di} = \omega_i (1 - \xi_i^2)^{1/2} \tag{2.2.24}$$

is little different from ω_i at usual values of the *logarithmic decrement*, therefore

$$\delta = 2\pi\xi\omega / \omega_D \approx 2\pi\xi. \tag{2.2.25}$$

In most cases the value of the logarithmic decrement general for the whole model (not for individual elements) is used, which corresponds to the structures made from the same material and with almost the same designs of the joints. Design models with a large and/or nonuniform damping, for example, those which include a subgrade subsystem are an exception. This case is taken into account only in the variant of the direct integration of the equations of motion.

Accounting for Damping

In the literature on the dynamic calculations different authors use different damping models, this difference is applied in various codes which use different forms of the representation of the internal losses, in particular:

- energy absorption factor (dissipation factor) — ψ;
- damping parameter in fractions of the critical value — ξ;
- logarithmic decrement of oscillations — δ;
- loss factor (complex internal friction) — ζ;
- coefficient of internal friction (inelastic resistance) — γ.

There are the following relationships between these parameters (see Section 3 of [4]):

$$\psi = 2\delta; \quad \gamma = \delta / \pi; \quad \xi \approx \delta / (2\pi); \quad \delta = \pi\eta / \sqrt{1 - \gamma^2 / 4}.$$

The necessary values of these parameters for different types of structures can be obtained in **KUST** included in the SCAD Office system.

If damping is taken into account the dynamic equilibrium equation (2.2.12) takes the following form

$$\boldsymbol{M}\ddot{\boldsymbol{Z}} + \boldsymbol{C}\dot{\boldsymbol{Z}} + \boldsymbol{K}\boldsymbol{Z}(t) = \boldsymbol{f}(t) \cdot \tag{2.2.26}$$

In the general case the conditions of orthogonality of the type (2.2.15) are no longer satisfied for the damping matrix $\boldsymbol{C}$, and the decomposition of the system of differential equations (2.2.26) into the independent ones with respect to the generalized coordinates $y_i(t)$ becomes impossible. However, you can make it possible by using the Rayleigh hypothesis on proportional damping

$$\boldsymbol{C} = \alpha \cdot \boldsymbol{M} + \beta \cdot \boldsymbol{K}. \tag{2.2.27}$$

Here α, β – coefficients of proportionality, the term $\alpha\boldsymbol{M}$ stands for the damping of lower modes, and term $\beta\boldsymbol{K}$ – for the damping of upper ones.

It follows from the decomposition of the displacement vector by the natural oscillation modes that:

$$\alpha + \beta\omega_i^2 = 2\xi_i\omega_i, \quad i = 1, 2, \ldots, N_{\text{modes}} \tag{2.2.28}$$

Let's use the equation (2.2.28) at i = 1, 2 to find the unknowns α, β:

$$\alpha = \frac{2\omega_1\omega_2(\xi_1\omega_2 - \xi_2\omega_1)}{\omega_2^2 - \omega_1^2}, \quad \beta = \frac{2(\xi_2\omega_2 - \xi_1\omega_1)}{\omega_2^2 - \omega_1^2}. \tag{2.2.29}$$

where ω_1, ω_2, – first two natural cyclic frequencies [rad/sec]; ξ_1, ξ_2 – modal damping for the first and second natural oscillation modes given in the percentage of the critical damping.

It is assumed that $\omega_2 > \omega_1 > 0$ and $\xi_2 \geq \xi_1 > 0$. In the case when $\omega_2 = \omega_1$ or the first two frequencies are very close ($(\omega_2 - \omega_1)/\omega_1 < 0.0001$), it is assumed that $\alpha = 2\xi_1\omega_1$, a $\beta = 0$.

The parameter α refers to damping of low modes, and β – of high ones. It is important to take both parameters into account. The following simplification is often made β = 0. Damping of the upper modes

turns out to be considerably underestimated, therefore the dynamic reaction of the system can be considerably overestimated (up to 2 times and more).

Damping models implemented in the method of the decomposition by the natural oscillation modes and at the direct integration using the Rayleigh hypothesis are generally different. It is evident, for example, from the ambiguity of the determination of the parameters α, β from (2.2.29), since by specifying different values of the pairs of the index i, we will obtain different values of α, β. However, for most of the practical problems the method of the decomposition by the natural oscillation modes and the methods of the direct integration give close results if both of them are applied correctly.

2.3. Solving Systems of Equations

Once a given structure is represented as a finite element model, the problem of determining nodal displacements is reduced to the solution of a system of linear algebraic equations:

$$\boldsymbol{KZ} = \boldsymbol{F}, \tag{2.3.1}$$

where $\boldsymbol{K}$ is a symmetric $N \times N$ matrix. It should be noted that matrix $\boldsymbol{K}$ is usually positive definite and contains a considerable number of zero elements (i.e. it is a sparse matrix);

$\boldsymbol{F}$ is the $N \times k$ matrix of right hand side (loadings), where k is the number of loadings;

$\boldsymbol{Z}$ is the unknown $k \times N$ matrix of displacements.

There are both direct and iterative methods for solving systems of linear algebraic equations. The direct methods are a powerful tool for solving systems of linear algebraic equations with sparse matrices if there is an efficient way to order the unknowns and thus reduce the fill-in in the matrix factorization. The advantages of these methods are that they are less sensitive to the ill-conditioning of systems of equations, enable to detect modeling mistakes that lead to a dimensional instability of the design model, and that the computation time does not depend much on the right hand side if the latter is relatively small.

Iterative methods for solving systems of linear algebraic equations are widely used to study large-scale problems of solid mechanics. They are particularly useful in cases when a lot of fill-ins appear in the structure of the adjacency graph levels, which drastically slows the performance of the direct methods. The iterative methods are usually used in cases when the matrix of the system of equations is positive definite.

The most popular direct methods used in FEM software include a profile method based on Gaussian exclusions, and a frontal method. In the first case the matrix of governing equations is fully formed before solving these equations (see Fig. 2.3-1); in the second case groups of equations usually containing unknowns of a single node are formed in RAM. The stiffness matrix of the system is not assembled explicitly; the stiffness matrices of the elements with the degrees of freedom included in the given group are gradually added instead. Once the new front is assembled, i.e. all its adjoining elements are included in the assembly, unknowns the numbers of which do not coincide with those of the front equations will be at once subjected to the Gaussian exclusion procedure. As a result, the Gaussian exclusion is performed in a dense (frontal) matrix of relatively small size that consists of a fully assembled part of the equations and an incomplete one. In the course of the solution, the front moves along the nodes of the FEM model. The efficiency of the multifrontal method is usually an order of magnitude greater than that of the modified Gaussian algorithm.

Arrangement of the order of the equation exclusion (profile method) or of the inclusion of the finite elements in the stiffness matrix assemblage (frontal method) aimed at the reduction of fill-ins is usually performed taking into account the structure of the stiffness matrix fill-ins. Here and further, a *fill-in* will refer to a nonzero element that appears in the matrix during the factorization in the place of an original zero element (see Fig. 2.3-1). The number of fill-ins greatly affects the efficiency of both methods for solving linear algebraic equations.

Several renumbering methods are implemented in **SCAD**, e.g., the reverse Cuthill-McKie algorithm, the factor tree method, the method of nested sections, the algorithm of parallel sections, the METIS algorithm, the Sloane algorithm, the minimum degree algorithm, etc. These methods and their comparative characteristics are described in the literature on the subject. The user is authorized to select a renumbering method, because the efficiency of an algorithm greatly depends on the structure of a particular matrix $\boldsymbol{K}$, and the problem of determining the optimal renumbering of the degrees of freedom belongs to "difficult problems" (NP-complete) [19], [26], [27].

It is recommended to use a high-performance parallel solver PARFES for multi-core computers, which enables to calculate large problems much faster.

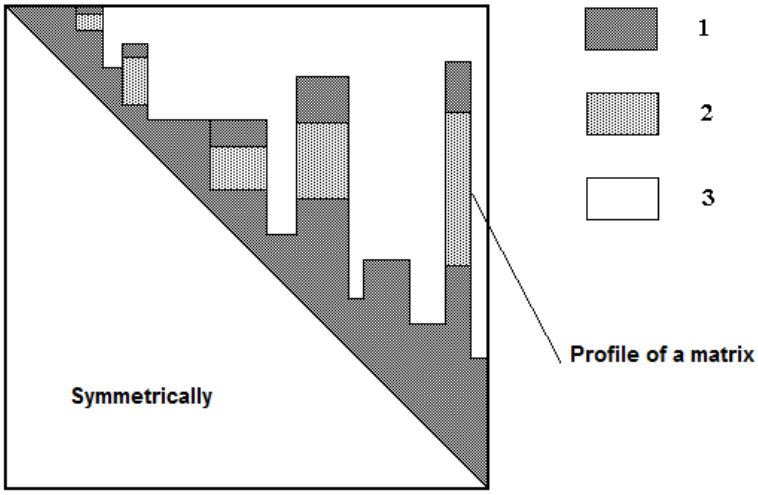


Figure 2.3-1. *The profile of a matrix.* 1 — nonzero elements, 2 — zero elements filled during the matrix decomposition, 3 — zero elements not filled during the matrix decomposition.

If any of the **SCAD** solvers is used and the process of triangular decomposition of matrix ***K*** makes one of the governing elements equal to zero, i.e. matrix ***K*** is found to be degenerate, which indicates that the structure is unstable, then an additional unstressed constraint is automatically imposed to make the system stable. The user receives information on the node numbers and the types of degrees of freedom involved in this additional constraining. It should be noted that the degenerate character of the matrix is not identified by an exact equality of the governing element to zero, but rather by a "nearly zero" number appearing on the main diagonal. The threshold for this "nearly zero" magnitude (a solution accuracy parameter) is one of the properties the user is authorized to set up.

When a message about additionally imposed constraints appears in the log file, we recommend you to review your design model carefully and find out the reason for the dimensional instability of the structure. It might turn out that the system is formally stable, but its elements have such parameters of rigidity that it is defined as "almost unstable". Then in order to perform the analysis you might have to solve the problem again with another solution accuracy parameter.

An additional service function is a system solution check (2.3.1). If there is a message that the solution error is too large (this is usually due to an ill-conditioned matrix ***K***), you should carefully analyze the nodal displacements and make sure that the obtained solution is acceptable from the engineering standpoint. The ill-conditionality is usually caused by an improper structural design (for example, the system is "almost unstable") or its incorrect idealization.

2.4. Standard Dynamic Load Cases

SCAD produces solutions of problems for dynamic loads of the following classes:

- modal analysis;
- wind pulsations;
- seismic load;
- pulse and impact loads with different time histories;
- harmonic load;
- seismic excitation in the form of an arbitrary accelerogram;
- direct integration of the equations of motion.

Further we will deal with formulations of these problems and methods of their computational implementation.

Wind Load

The method is based on the approach from Section 10 of the manual [4] where the problem of the action of turbulent wind flow pulsations is formulated as a problem of statistical dynamics.

The wind pressure on a structure in a point located at the height z from the ground level is considered as a sum of a static and a pulse components of the wind load

$$q(z,t) = q^s(z) + q^p(z,t). \tag{2.4.1}$$

The latter is a stochastic function of time determined by the random velocity of the pulsations which has a zero average value, a standard deviation $\sigma(z)$, and a dimensionless Davenport spectrum

$$S(\varepsilon) = 1200\varepsilon^{5/3} / [3v_0(1 + \varepsilon^2)^{4/3}], \tag{2.4.2}$$

where v_0 is an hourly average wind velocity at the height of 10 m;

$\varepsilon = v_0\, L^{-1}\Omega^{-1}$ is a dimensionless period of oscillations where L=1200 m is a turbulence scale, and Ω is a circular frequency (1/s).

Taking into account certain simplifications[1] based on the assumption of full correlation (the wind velocity pulsations are considered as synchronous over a space of stochastic functions of time only), we can represent the root-mean-square displacement — the j-th component of the vector Z — as

$$Z_{\partial j} = \sum_{i=1}^{s} (\eta^2_{ij}\,\mu_i^2) / \omega^4_i. \tag{2.4.3}$$

Here, s is the number of considered natural oscillation modes; η_{ij} is the reduced acceleration calculated according to the following formula

$$\eta_{ij} = (\Psi_{ij} \sum_k \sigma_k \Psi_{ik}) / \sum_r M_r \Psi_{ir}, \tag{2.4.4}$$

and the square of the dynamic amplification factor is determined as follows:

$$\mu_i^2 = (2/3) \int_0^\infty \varepsilon^{11/3}\ (1 + \varepsilon^2)^{-4/3}\, [\varepsilon^4 - 2(1 - \gamma^2/2)\varepsilon^2\varepsilon^2_i + \varepsilon^4_i]^{-1}\, d\varepsilon. \tag{2.4.5}$$

The damping parameter γ is related to the logarithmic decrement δ by the relationship $\gamma = \delta/\pi$, and ε_i denotes a dimensionless period of natural oscillations $\varepsilon_i = v_0\, L^{-1}\Omega^{-1}$.

The static design displacements are defined by the first member of (2.4.1), and the dynamic ones are determined by (2.4.3). The forces in elements of the system and the displacements of its points (which are together referred to as a response of the structure X) are determined separately from the static component of the wind load and from the inertia forces corresponding to each natural oscillation mode. The total value of the response can be determined by the formula:

$$X = X^c \pm \left[\sum_i (X_i^d)^2 \right]^{1/2}, \tag{2.4.6}$$

that shows that the oscillations occur around the shifted equilibrium state that corresponds to the static (average) component of the loading. The software outputs particular components of the dynamic response X_i^d and the overall magnitude (2.4.6), the sign before the second member being the same as that of the component X^c.

The summation of contributions of individual natural modes given by the formula (2.4.6) is called the SRSS rule ("Square Root of the Sum of the Squares"). This rule is used to sum the dispersions of statistically independent random variables in a Gaussian distribution. It is assumed that the amplitude values and phase angles of random functions which describe displacements excited by turbulent wind pulsations possess such properties.

The assumption of statistical independence is hardly valid for a group of natural modes defined by a certain multiple frequency. These oscillations are phase-dependent, because they are excited by a common frequency and therefore multiple vectors are summed algebraically.

[1] These simplifications are not used in the **SCAD** mode that performs an analysis according to the Moscow State Building Standards. The solution thus obtained is more accurate but the time needed to find it is much longer.

It should also be noted that in the case of structures symmetrical in plan multiple pairs of natural modes are usually formed, which are determined with an accuracy up to a rotation about the vertical axis. The vectors of this pair will be orthogonal to each other, but arbitrarily rotated about a vertical axis, therefore, the orientation of the algebraic sum of a pair of multiple modes can turn out to be arbitrarily oriented as well.

Therefore, a special command is provided for rotating the sum vector so as to obtain its maximum contribution to the overall dynamic response of the structure.

Seismic Actions

The regulations [14] are based on a spectral approach according to which the design spectrum curve (a law of variation of the dynamic amplification factors) defines the dynamic response of a simple pendulum to the acceleration of its suspension point under a seismic disturbance.

Generally, when the foundation of a structure experiences the accelerations d^2z_0/dt^2, caused by an earthquake, the inertia forces $\boldsymbol{J}(t) = \boldsymbol{M}\, d^2\boldsymbol{Z}/dt^2$ are defined by the absolute accelerations added to the relative ones caused by deformations of the structure

$$J_c(t) = M\,(d^2Z/dt^2 - d^2z_0/dt^2), \tag{2.4.7}$$

which gives the following fictitious load when substituted to (2.2.12)

$$f_c(t) = M\, d^2z_0/dt^2\,. \tag{2.4.8}$$

The solution under the action of (2.4.8) is sought for as an expansion by natural oscillation modes in the form of a Duhamel integral (see Section 3 of the reference manual [4]); eventually the problem is reduced to an analysis of inertia forces S_{ik} that act in the direction of k-th degree of freedom at the oscillations in i-th mode

$$S_{ik} = Q_k K_c\, \beta(T_i)\eta_{ik} \cos\varphi_{0k}. \tag{2.4.9}$$

Here Q_k is the weight of the mass related to the k-th degree of freedom, K_c is the seismic amplification factor that depends on the magnitude of the earthquake (recall that each next magnitude number corresponds to a doubling of the seismic force) and on the class of the structure; β is the dynamic amplification factor that depends on the natural oscillation period of the current mode and the category of the subsoil; η_{ik} are reduced accelerations (2.4.4) (coefficient which characterizes the distribution of the inertial forces among the natural oscillation modes); φ_{0k} is the angle between the direction of the displacement Z_k and the ground acceleration vector.

The dynamic amplification factors are determined by the processing of spectral responses, simple oscillators with natural frequencies ω_i (periods $T_i = 2\pi/\omega_i$) and coefficient of inelastic resistance γ to the accelerogram action $d^2z_0/dt^2 = a_0(t)$.

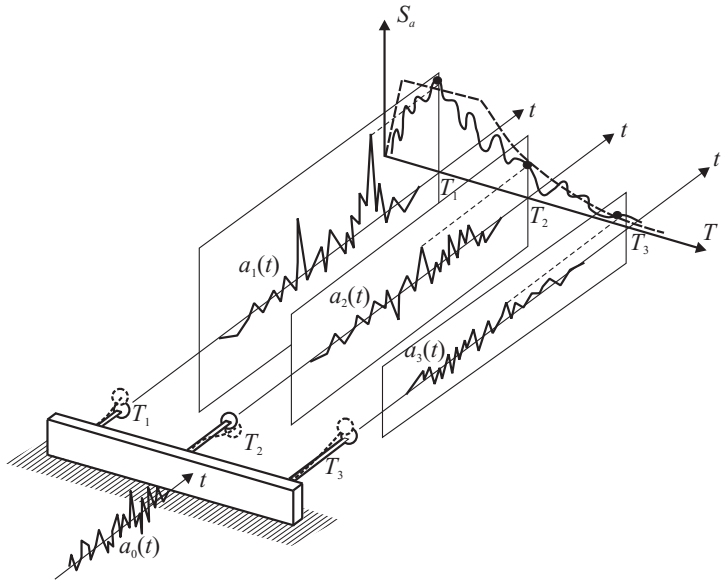


Figure 2.4-1. Plotting the dynamic amplification factor curve

The action $a_0(t)$ defines the response of each oscillator (Fig. 2.4.1)

$$a_i(\omega_i,\gamma,t)=\omega_i\int_0^t \ddot{z}_0(\tau)e^{-\gamma\omega_i(t-\tau)}\sin\omega_i(t-\tau)d\tau\,. \tag{2.4.10}$$

The maximum values of the responses given as a function of the oscillator frequency define the spectral curve of the original accelerogram,

$$\beta(\omega_i,\gamma)=\max_t\left|a(\omega_i,\gamma,t)\right|=\max_t\left|\omega_i\int_0^t \ddot{z}_0(\tau)e^{-\gamma\omega_i(t-\tau)}\sin\omega_i(t-\tau)d\tau\right|,$$

and the envelope of the set of spectral curves corresponding to different possible accelerograms, generalized accordingly, gives the dynamic amplification factor curve $\beta(T_i)$.

The coefficients η_{ik} are calculated by the formula

$$\eta_{ki}=\Psi_{ki}\left(\sum_{j=1}^{n}Q_i\Psi_{ij}\right)\Bigg/\left(\sum_{s=1}^{n}Q_s\Psi_{is}^2\right). \tag{2.4.11}$$

The responses X_i to the loads (2.4.9) are calculated for each oscillation mode taken into consideration and the design value is calculated

$$X=\pm\sqrt{\sum_{i=1}^{n}X_i^2+\sum_{i=1}^{n-1}\rho_i\left|X_iX_{i+1}\right|}\,, \tag{2.4.12}$$

where ρ_i=2, if $T_{i+1}/T_i\geq 0{,}9$ and ρ_i=0, if $T_{i+1}/T_i<0{,}9$ ($T_i>T_{i+1}$).

When performing the dynamic analysis the software enables to add the remainders. A "residual mode" is calculated for each direction (X,Y,Z). When it is added to the sum of previously determined eigenvectors, the sum of modal masses is supplemented to 100% in the respective direction.

Pulse Loads

Pulse loads are short-term forces or impacts with the duration τ not exceeding $2{,}5T_s$, where T_s is the period of the highest oscillation mode taken into consideration (see Section 5 of the reference manual [4]). There are single pulses for which the load $f(t)$ = 0 at $t > \tau$, and repeated pulses, which include the periodic ones repeated in equal periods of time T_0.

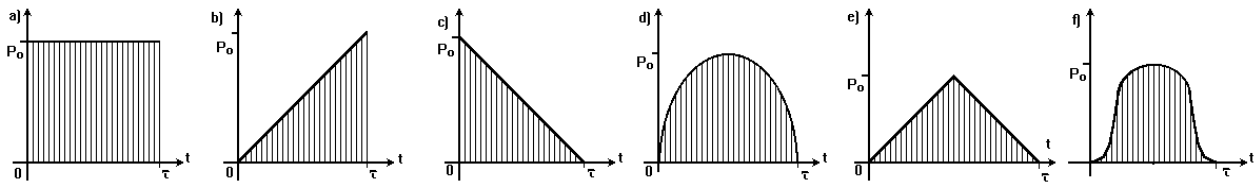


Figure 2.4-2. *Shape functions of pulse loads*

The pulse is described by its duration τ, its shape $f(t)$, and its maximum value P_0 (Fig. 2.4-2), or the value of the force momentum

$$S=P_0\int_0^{\tau}f(t)\;dt. \tag{2.4.13}$$

At $\tau < 0{,}1T_s$ the pulse can be treated as instantaneous, and its shape $f(t)$ does not matter because any of the shapes would yield about the same result.

If a body of the mass M_0 hits the structure directly, the pulse is determined by the formula:

$$S=M_0v_0(1+k_0), \tag{2.4.14}$$

where v_0 is the velocity of the hitting body at the moment of contact with the structure; k_0 is a restoration coefficient the value of which depends on the shapes and materials of the hitting bodies.

If the mass M_0 is small comparing to the sum of masses of the structure, the analysis of the impact can be performed as for a pulse of the magnitude (2.4.13); otherwise, M_0 should be treated as an associated mass and thus more accurate values of the natural frequencies should be obtained.

It should be noted that the associated mass M_0 cannot be the only mass in the system (an impact upon a mass-free structure is not considered!). The recommended shape of the $f(t)$ function in the impact is a bell (Fig. (2.4-2, *f*).

Harmonic Excitation

For the harmonic load $P_i^{(1)}\cos\theta t + P_i^{(2)}\sin\theta t$, the forces of inertia S_1 and S_2, summed over all natural oscillation modes that are taken into consideration, which correspond to a cosine (real) and sine (imaginary) components, respectively, are determined by the formulae:

$$S_1 = \sum_i a_i M \Psi_i \quad ; \qquad S_2 = \sum_i b_i M \Psi_i , \tag{2.4.15}$$

where the coefficients are

$$a_i = (P_i^{(1)}\chi - P_i^{(2)}\xi\alpha_i)/(\chi^2{}_i + \alpha^2{}_i),$$
$$b_i = (P_i^{(2)}\chi - P_i^{(1)}\xi\alpha_i)/(\chi^2{}_i + \alpha^2{}_i),$$
$$\alpha_i = \theta/\omega_i, \quad \chi_i = 1 - \alpha^2{}_i. \tag{2.4.16}$$

It is assumed here that a single dimensionless damping ratio ξ is specified for the whole model.

The maximum values are determined as follows

$$S = (S_1^2 + S_2^2)^{1/2}. \tag{2.4.17}$$

In cases when the frequency of the exciting load θ is higher than one or more natural frequencies of the system ω_i, it is necessary to perform an additional check to see whether the system experiences the state of resonance during the startup or shutdown of machines and aggregates that generate the harmonic load (see [4], p.53).

Analysis with Accelerograms

In the load configurations listed above, it was possible to perform a precise calculation of the function $y_i(t)$. In all other cases the solutions $y_i(t)$ are calculated numerically.

Particularly, when a seismic load specified by an accelerogram is under consideration, the vector $f_k = f(t_k)$ is specified at every k-th moment of time t_k. Then in (2.2.20) we have a set of generalized forces discrete in time $P_{i,k} = P_i(t_k)$. These forces are calculated by the formula similar to (2.4.9):

$$P_{ik} = Q_k K_c\, a(\omega_i, \gamma, t_k)\eta_{ik}\cos\varphi_{0k}, \tag{2.4.18}$$

where the value $a(\omega_i, \gamma, t_k)$ is the accelerogram response $\ddot{z}_0(\tau)$.

Next, the equations (2.2.19) are solved by Duhamel integrals. Since an accelerogram is given with a certain time step, the interpolation in the intermediate points is used when necessary for the high-frequency components for which the difference model condenses with time.

This will get values of the displacements $y_{i,k} = y_i(t_k)$ and forces of inertia $S_{i,k} = S_i(t_k)$, which are then used to determine an extreme value over the considered moments of time.

Direct Integration of the Equations of Motion

One of the peculiarities of this mode is the fact that displacements, velocities, and accelerations of the nodes of the finite element model are determined in the absolute reference system connected with the conventionally immobile earth. The seismic action is specified as the forced displacements of supports, therefore not the accelerograms but seismograms (relationship between displacements and time) obtained from accelerograms by the double time integration are used as the input information. This approach enables to solve the problems with asynchronous oscillations of supports in a natural way. It is recommended to use the **Accelerogram Editor** to obtain seismograms from the specified accelerograms.

A traditional problem formulation in the reference system connected with the rigid mobile base, and a problem formulation in the absolute reference system used at the direct integration of the equations of motion are shown in Fig. 2.4-3 and 2.4-4 respectively. In order to distinguish from other modes of the dynamic analysis U instead of Z is used for designating the displacements.

In the classical problem formulation:

$$\left.\begin{aligned} &\boldsymbol{u} = \boldsymbol{u}_e + \boldsymbol{u}_r; \\ &\ddot{\boldsymbol{u}} = \ddot{\boldsymbol{u}}_e + \ddot{\boldsymbol{u}}_r; \\ &\boldsymbol{M}\ddot{\boldsymbol{u}}_r + \boldsymbol{K}\boldsymbol{u}_r = -\boldsymbol{M}\ddot{\boldsymbol{u}}_e(t) + \boldsymbol{p}(t) \end{aligned}\right\} \tag{2.4.19}$$

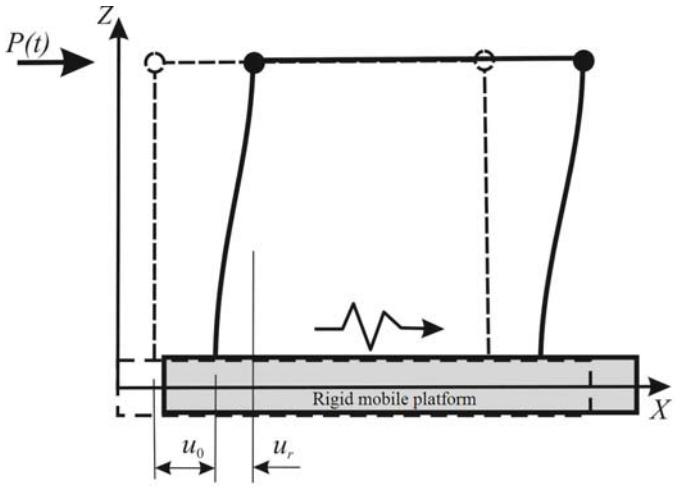


Figure 2.4-3. Solution of the problem in the relative reference system connected with the rigid mobile platform exhibiting translational motion (classical formulation)

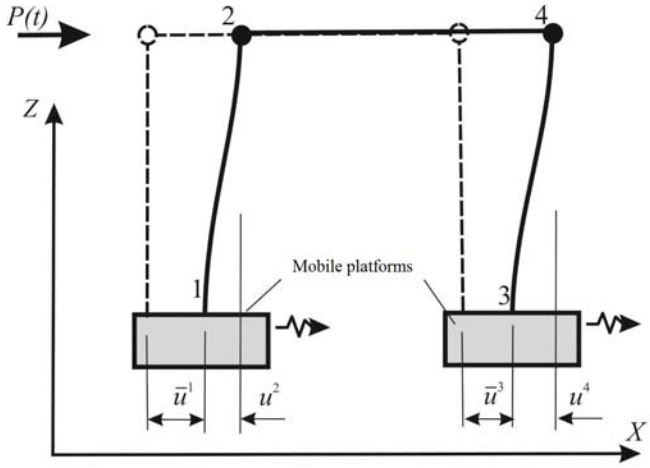


Figure 2.4-4. Solution of the problem with respect to the conventionally immobile absolute reference system

The problem formulation shown in Fig. 2.4-4 is described by the following system of equations:

$$\underbrace{\begin{pmatrix} \mathbf{M}_{11} & 0 \\ 0 & \mathbf{M} \end{pmatrix}}_{\hat{\mathbf{M}}} \cdot \begin{pmatrix} \ddot{\bar{\mathbf{u}}} \\ \ddot{\mathbf{u}} \end{pmatrix} + \underbrace{\begin{pmatrix} \mathbf{K}_{11} & \mathbf{K}_{12} \\ \mathbf{K}_{21} & \mathbf{K} \end{pmatrix}}_{\hat{\mathbf{K}}} \cdot \begin{pmatrix} \bar{\mathbf{u}} \\ \mathbf{u} \end{pmatrix} + \underbrace{\begin{pmatrix} \mathbf{C}_{11} & \mathbf{C}_{12} \\ \mathbf{C}_{21} & \mathbf{C} \end{pmatrix}}_{\hat{\mathbf{C}}} \cdot \begin{pmatrix} \dot{\bar{\mathbf{u}}} \\ \dot{\mathbf{u}} \end{pmatrix} = \begin{pmatrix} 0 \\ \mathbf{p}(t) \end{pmatrix}, \tag{2.4.20}$$

where

$$\hat{\boldsymbol{u}} = \begin{pmatrix} \bar{\boldsymbol{u}} \\ \boldsymbol{u} \end{pmatrix}; \quad \bar{\boldsymbol{u}} = \begin{pmatrix} \bar{u}^1 \\ \bar{u}^3 \end{pmatrix}; \quad \boldsymbol{u}^{\mathsf{T}} = \begin{pmatrix} u^2 & w^2 & \theta^2 & u^4 & w^4 & \theta^4 \end{pmatrix}, \tag{2.4.21}$$

$\bar{\mathbf{u}}$ – specified displacements, $\mathbf{u}$ – displacements which have to be determined, $\hat{\mathbf{M}}, \hat{\mathbf{K}}, \hat{\mathbf{C}}$ – stiffness, mass, and dissipation matrices corresponding to the full vectors of displacements, accelerations, and velocities $\hat{\mathbf{u}}, \ddot{\hat{\mathbf{u}}}, \dot{\hat{\mathbf{u}}}$; $\mathbf{M}, \mathbf{K}, \mathbf{C}$ – stiffness, mass, and dissipation matrices corresponding to the degrees of freedom which have to be determined (standard FEM matrices), matrix $\mathbf{K}_{12}$ is assembled from the

stiffness submatrices of the finite elements, the nodes of which contain the degrees of freedom of the vector of forced displacements $\bar{\mathbf{u}}$, $\mathbf{p}(t)$ – load vector.

Damping in the material is taken into account on the basis of the Rayleigh hypothesis:

$$\hat{\mathbf{C}} = \alpha \cdot \hat{\mathbf{M}} + \beta \cdot \hat{\mathbf{K}} . \tag{2.4.22}$$

Here α, β – coefficients of proportionality, the term $\alpha \cdot \mathbf{M}$ stands for the damping of lower modes, and term $\beta \cdot \mathbf{K}$ – for the damping of upper ones.

If there are local dampers, a non-Rayleigh dissipation matrix $\hat{\mathbf{C}}$ appears. If at the same time there is no damping in the material, you should specify $\alpha = \beta = 0$. Otherwise, both damping caused by local dampers and the damping in the material will be taken into account.

The following expression can be obtained from (2.4.20) taking into account (2.4.22):

$$\mathbf{M}(\ddot{\mathbf{u}} + \alpha \cdot \dot{\mathbf{u}}) + \mathbf{K}(\mathbf{u} + \beta \cdot \dot{\mathbf{u}}) = -\mathbf{K}_{21}\left(\bar{\mathbf{u}}(t) + \beta \cdot \dot{\bar{\mathbf{u}}}(t)\right) + \mathbf{p}(t) . \tag{2.4.23}$$

Whereas in the classical formulation the forced displacement of supports is usually given by an accelerogram, the displacement function $\bar{\mathbf{u}}(t)$ is specified in the proposed formulation, and the velocity function $\dot{\bar{\mathbf{u}}}(t)$ is determined by numerical differentiation of the latter.

Designate

$$\bar{\mathbf{u}}(t) = \sum_{s=1}^{M_s} \mu_s \cdot \mathbf{k}_s \cdot f_s\left(t - t_0^s\right), \quad \dot{\bar{\mathbf{u}}}(t) = \sum_{s=1}^{M_s} \mu_s \cdot \mathbf{k}_s \cdot \dot{f}_s\left(t - t_0^s\right), \tag{2.4.24}$$

where $\mathbf{k}_s$ – vectors of the spatial configuration specifying the mode of the forced displacements, $f_s(t)$ – time function, $s = 1, 2, \cdots, M_s$ – numbers of static loadings at the calculation for the respective forced displacements.

Similarly $\mathbf{p}(t) = \sum_{p=1}^{N_p} \mu_p \cdot \mathbf{k}_p \cdot f_p\left(t - t_0^p\right)$, where $p = 1, 2, \ldots, N_p$ – numbers of static loadings at the calculation for force actions, $\mathbf{k}_p$ – vectors of the spatial load configuration, $f_p(t)$ – time function.

Values $\mu_s, t_0^s, \mu_p, t_0^p$ refer to the scale factors and the delay time. We will call each term of the above sums as the p-th or s-th component of the dynamic loading respectively. Designate

$$\mathbf{b}_s = -\mathbf{K}_{21} \cdot \mathbf{k}_s, \quad s = 1, 2, \ldots, M_s . \tag{2.4.25}$$

The following Cauchy problem describes the equations of motion in the given formulation

$$\left.\begin{aligned} &\boldsymbol{M}\left(\ddot{\boldsymbol{u}} + \alpha \cdot \dot{\boldsymbol{u}}\right) + \boldsymbol{K}\left(\boldsymbol{u} + \beta \cdot \dot{\boldsymbol{u}}\right) = \\ &\quad = \sum_{s=1}^{M_s} \boldsymbol{b}_s \cdot \mu_s \cdot \left[f_s\left(t - t_0^s\right) + \beta \cdot \dot{f}_s\left(t - t_0^s\right)\right] + \sum_{p=1}^{N_p} \boldsymbol{k}_p \cdot \mu_p \cdot f_p\left(t - t_0^p\right) \\ &\boldsymbol{x}(0) = \boldsymbol{x}_0 \\ &\dot{\boldsymbol{x}}(0) = \dot{\boldsymbol{x}}_0 \end{aligned}\right\} . \tag{2.4.26}$$

Unconditionally stable Newmark method in the "predictor-corrector" form is used for solving the problem.

2.5. Stability of Equilibrium

Governing Equations

Let's consider the equilibrium of a system in the initial configuration of which the following conditions hold:

$$\boldsymbol{K}_0 \boldsymbol{Z} = \boldsymbol{f} \tag{2.5.1}$$

Let now the load acquire an increment $\delta\boldsymbol{f}$ corresponding to such values of the stress increment $\delta\boldsymbol{s}$ and the

displacement increment $\delta \boldsymbol{Z}$ that components of the vector of additional displacements $\delta \mathbf{Z}$ are relatively small but nevertheless require a geometrically nonlinear analysis to be performed. Then, as commonly known from [21], the equations in variations can be represented in the following form:

$$[\boldsymbol{K_0} + \boldsymbol{K}_1(\boldsymbol{s_0}) + \boldsymbol{K}_2(\boldsymbol{Z_0})]\delta \boldsymbol{Z} = (\boldsymbol{K_0} + \boldsymbol{K}_G)\delta \mathbf{Z} = \delta \mathbf{f}. \tag{2.5.2}$$

Here, a conventional stiffness matrix $\boldsymbol{K_0}$ acquires an increment in the form of the matrix of initial stresses $\boldsymbol{K}_1(\boldsymbol{s_0}) = \boldsymbol{T}(\boldsymbol{s_0})$, linearly dependent on stresses in the system before the load is incremented, plus the matrix of initial rotations $\boldsymbol{K}_2(\boldsymbol{Z_0})$ that includes not more than degree two of the displacements. Equations (2.5.2) do not include the matrix of initial loading by J.Oden [22], that makes corrections to the load. The non-allowance for the initial loading matrix means that we assume the load to be independent from the system's configuration, or we think that appropriate corrections have been already made when forming the load increment vector $\delta \boldsymbol{f}$. Matrix $\boldsymbol{K}_G$ is conventionally called a *geometrical stiffness matrix*.

Single-Parametric Loading

Let's consider a system equilibrated in the state for which equations (2.5.2) hold, and write equalities to determine the changes during trial loading by forces proportional to a single parameter, i.e., when the load is incremented as $\lambda\delta \boldsymbol{f}$. Denoting the respective changes in the forces and displacements as $\lambda\delta \boldsymbol{s}$ and $\lambda\delta \boldsymbol{Z}$, we have

$$[\boldsymbol{K_0} + \boldsymbol{K}_1(\boldsymbol{s_0} + \lambda\times\delta \boldsymbol{s}) + \boldsymbol{K}_2(\boldsymbol{Z_0} + \lambda\times\delta \boldsymbol{Z})]\ \delta \boldsymbol{Z} = \delta \boldsymbol{f}. \tag{2.5.3}$$

Now let us use the new stiffness matrix to find the critical value of the load $\boldsymbol{f}_{cr} = \boldsymbol{f_0} + \delta \boldsymbol{f}$. Expanding $\boldsymbol{K}_1$ and $\boldsymbol{K}_2$ from (2.5.3) into a series and keeping terms linearly dependent on λ, we obtain the equation:

$$\{[\boldsymbol{K_0} + \boldsymbol{K}_1(\boldsymbol{s_0}) + \boldsymbol{K}_2(\boldsymbol{Z_0})] + \lambda\ [\boldsymbol{K}_1(\delta \boldsymbol{s}) + \boldsymbol{K}_2(\boldsymbol{Z_0}\ ,\delta \boldsymbol{Z})]\}\delta \boldsymbol{U} = \boldsymbol{0}. \tag{2.5.4}$$

or

$$[\boldsymbol{K_0} + \lambda\Delta \boldsymbol{K}_G(\delta \boldsymbol{s},\boldsymbol{Z_0}\ ,\delta \boldsymbol{Z})]\delta \boldsymbol{U} = \boldsymbol{0}, \tag{2.5.5}$$

where $\delta \boldsymbol{U}$ is an eigenvector that describes a mode of buckling of the system, and $\Delta \boldsymbol{K}_G$ denotes the matrix multiplier that stands in the brackets to the right from λ in equation (2.5.4).

The solution of eigenvalue problem (2.5.5) is based on a linear approximation in the vicinity of the shifted state of equilibrium. Therefrom we can derive the linear buckling problem assuming all initial stresses and displacements to be zero. If further we omit the quadratic terms in the matrix of initial displacements, then we will come to the result that the left part of (2.5.3) will linearly depend on the load

$$\{\boldsymbol{K_0} + \lambda\ [\boldsymbol{K}_1(\delta \boldsymbol{s}) + \boldsymbol{K}_2(\delta \boldsymbol{Z})]\}\ \delta \boldsymbol{U} = \boldsymbol{0}. \tag{2.5.6}$$

The assumption of independence of the critical load from displacements $\lambda\delta \boldsymbol{Z}$ eliminates matrix $\boldsymbol{K}_2$, and we come to Eulerian equations for the critical load

$$[\boldsymbol{K_0} + \lambda\ \boldsymbol{K}_1(\delta \boldsymbol{s})]\ \delta \boldsymbol{U} = \boldsymbol{0}. \tag{2.5.7}$$

The minimum eigenvalue of this equation λ_1 determines the critical load as $\lambda_1\delta \boldsymbol{f}$, the value of λ_1 itself is the stability factor of safety, and the vector $\lambda_1\delta \boldsymbol{s} = \boldsymbol{S}^*$ defines the internal forces in the system at its buckling.

It is worth noticing that the use of Eulerian equations essentially assumes the unchangeable distribution of the internal forces in the system as the load intensity λ increases. This feature does not always take place, especially in nonlinearly deformable systems. Therefore some software developments [9] determine two critical values of loads: the first one is the least critical value of λ_1 in equations of the (2.5.7) type, and the second one is determined as the load intensity at which the equations of the (2.5.2) type lose their positive definiteness in the course of a loading subject to some more or less realistic law (for example, when the live load grows and the dead load remains the same) modeled by a stepwise procedure.

A nontrivial problem immediately arises of how to choose the critical value to be demonstrated as an evidence that the calculated stability factor of safety is sufficient. Unfortunately, the design codes regulating the values of the stability factors of safety never mention these critical values. Though, there are some indirect indications that suggest the use of the Eulerian value.

Special attention should be paid to the case of the stability problem when two loadings $\boldsymbol{f}_1$ and $\boldsymbol{f}_2$ act on the system, but only one of them changes, while the other remains stable. Then we will have the following expression instead of (2.5.3)

$$[\boldsymbol{K_0} + \boldsymbol{K}_1(\boldsymbol{s_0} + \lambda\times\delta\boldsymbol{s}) + \boldsymbol{K}_2(\boldsymbol{Z_0} + \lambda\times\delta\boldsymbol{Z})]\ \delta\boldsymbol{Z} = \delta\boldsymbol{f}_1+\boldsymbol{f}_2, \tag{2.5.8}$$

and the factor of safety λ_1 will relate to the possible value of the load increment f_1. The respective mode is implemented in **SCAD** as well.

Analysis of a Role Played by Particular Subsystems

When analyzing peculiarities of possible buckling phenomena, one often uses the concepts of a *constricted and forced* state of separate parts of a system when it buckles [13]. It is assumed that a part of a structure is in the constricted state when the structure itself buckles if this part, when separated from the rest of the system, buckles at a lower load transferred to this part than that under which it actually does. If this "isolated buckling" analysis of the selected part yields a higher load intensity that should be applied to the system so that the latter buckles, or if the system does not buckle at all, this situation is referred to as a forced buckling of the part of the system. Naturally these concepts affect the strategy of the system reinforcement if an insufficiency of the total stability factor of safety has been stated. One commonly begins with reinforcing that part of the structure which is in the constricted state. Though, it should be remembered that this measure is effective only to a certain extent. The criterion is that the subsystem being reinforced acquires the same "power" as the rest of the structure. To perform further reinforcing efficiently, one should reinforce all compressed members together to keep the achieved state of a uniform stability of the system members [13, p. 254].

The concepts of the constricted and forced states with respect to selected fragments of a structure introduced into the engineering practice over half a century ago enable one to form a purposeful strategy of reinforcing the structure the load-bearing capacity of which is determined by its stability. Meanwhile, as far as the authors can judge, there has been developed no formal computational tool for this, one being especially needed in the computer-aided design. This is even more astonishing seeing that the algorithms for such tools are surprisingly simple and self-evident.

Indeed, the role of particular subsystems can be checked by calculating the energy accumulated in different parts of the system as it deforms according to a buckling mode. For the whole system this energy is equal to zero:

$$\mathrm{E} = 0{,}5(\boldsymbol{Z}^*)^{\mathrm{T}}[\boldsymbol{K}_0 - \boldsymbol{K}_1(\boldsymbol{S}^*)]\boldsymbol{Z}^* = 0. \tag{2.5.9}$$

If the system has been divided imaginarily into two parts, then a stiffness matrix can be composed for each of the parts, $\boldsymbol{K}_{0,i} - \boldsymbol{K}_{1,i}(\boldsymbol{S}^*)$ (i = 1, 2), which we fill up with zeros to the dimension of the full stiffness matrix of the system. Calculating the following values

$$\mathrm{E}_1 = 0{,}5(\boldsymbol{Z}^*)^{\mathrm{T}}[\boldsymbol{K}_{0,1} - \boldsymbol{K}_{1,1}(\boldsymbol{S}^*)]\boldsymbol{Z}^* \tag{2.5.10}$$

$$\mathrm{E}_2 = 0{,}5(\boldsymbol{Z}^*)^{\mathrm{T}}[\boldsymbol{K}_{0,2} - \boldsymbol{K}_{1,2}(\boldsymbol{S}^*)]\boldsymbol{Z}^*, \tag{2.5.11}$$

will always give us values of opposite signs because their sum must be zero. This determines a subsystem (corresponding to $\mathrm{E}_i < 0$), which is reasonable to stiffen or brace additionally. Comparing these quantities with the concepts of constricted and forced states, it is not hard to guess that if $\mathrm{E}_i < 0$, then the respective **i**-th subsystem is a pushing one, otherwise at $\mathrm{E}_i > 0$ it is a restraining one. Roughly speaking, the pushing elements are the ones to "blame" for the buckling of the system.

Naturally the structure can be divided into more than two parts, even down to separate elements. The contributions of each element to the positive or negative parts of the total energy are then compared, and recommendations on strengthening/weakening particular elements are obtained. This helps to find out the role played by particular elements in the general buckling behavior in more detail.

We would like to note that one should be careful enough when using results of the above stated analysis to make a decision of modifying elements or subsystem. The matter is that the energy calculation is performed with the buckling mode **Z*** which has been determined for the system with an original stiffness and stress distribution. However, this mode also changes when these parameters are modified. Therefore the result of such modification must necessarily be validated. Also, which is even more

important, the effect of reinforcing particular elements is bounded by a certain limit above which this operation does not affect the result anymore. One can easily understand this statement by noticing that even an infinite growth of the stiffness of one of the parts of a system does not result in the same growth of the stability factor of safety because the system starts buckling in a "local" mode without deforming the over-stiffened part. Apparently, the first one who noticed this peculiarity of the buckling analysis of elastic systems was I.G. Bubnov who stated that there is a limit stiffness of supports in a problem of buckling of a beam on two such supports beyond which the further growth of it does not increase the critical force, and the supports with that limit stiffness behave like perfectly rigid ones [25].

2.6. Design Combinations of Forces (DCF)

Unfavorable design combinations of forces are selected in **SCAD** on the basis of the superposition principle. In order to limit the number of design combinations of forces (DCF) to be taken into consideration, a particular approach is used for every type of the stress state. Out of 2^n combinations (where n is the number of loadings), only those DCF are selected that correspond to the maximum value of a certain quantity which is assigned to be a criterion and depends on all components of the stress state.

Bars

Extreme values of normal and tangential stresses in checkpoints of the bar cross-section (Fig. 2.6-1) serve as the criterion for determining DCF.

The following formula is used for the normal stresses:

$$\sigma_k = \frac{N}{F} \pm \frac{M_y z}{I_y} \pm \frac{M_z y}{I_z}, \tag{2.6.1}$$

where k is a point in the bar cross-section (k = 1,…, 9). This formula transforms as follows at $y = \pm\frac{b}{2}$ and $z = \pm\frac{h}{2}$;

$$\sigma_k F = N \pm \frac{M_y}{l_{z,i}} \pm \frac{M_z}{l_{y,i}}, \tag{2.6.2}$$

where $l_{y,i}$ and $l_{z,i}$ are core sizes in the bar cross-section (i = 1, 2).

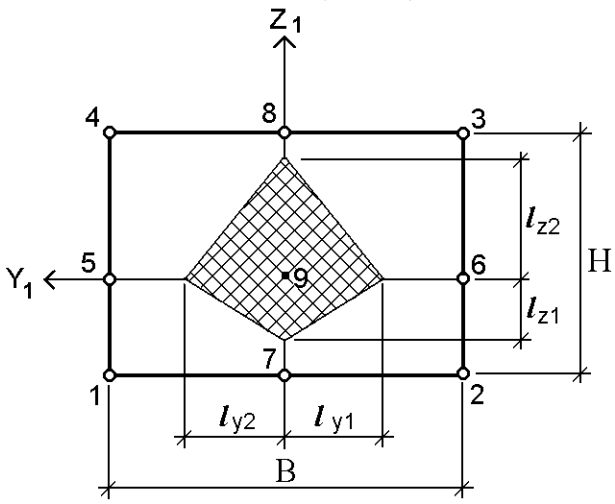


Figure 2.6-1. Numbering of the checkpoints

The following criterion is used to find the unfavorable value

$$\sigma_k F = N \pm \frac{M_y}{l_{z,i}} \sin\frac{n\pi}{8} \pm \frac{M_z}{l_{y,i}} \sin\frac{m\pi}{8}, \tag{2.6.3}$$

where m, n=0, 1, 2, 3, 4.

The given criterion is important for the reinforced-concrete structures.
This approach enables to determine the extreme normal stresses in a section of any shape.
The following approximate formula is used for the tangential stresses:

$$\tau_{y,z} F = \frac{Q_{y,z}}{2} \pm \frac{M_{кр}}{2(l_{y1,z1} + l_{y1,z2})} \tag{2.6.4}$$

Besides stresses, extreme values of the longitudinal and shear forces are calculated as well.

Membranes (Plane Stress State)

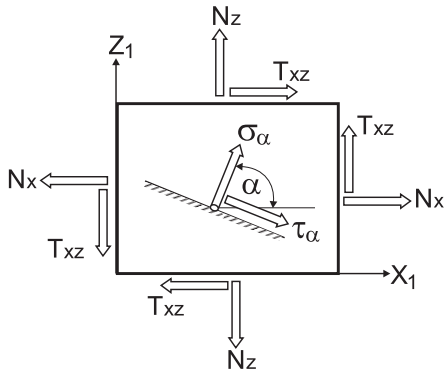


Figure 2.6-2. Stresses on inclined planes

Generally, the principal stresses at the same point of a structure are oriented differently under different loadings. Therefore, in this case the design combinations of forces are determined by envelope extreme curves of normal and tangential stresses according to the following formulae:

$$\sigma(\alpha) = N_x \cos^2 \alpha + N_z \sin^2 \alpha + T_{xz} \sin 2\alpha; \tag{2.6.5}$$

$$\tau(\alpha) = \frac{1}{2}\left(N_z - N_x\right)\sin 2\alpha + T_{xz} \cos 2\alpha \cdot \tag{2.6.6}$$

The normal stresses are calculated in the range from 90° to -90°, and the tangential ones in the range from 90° to 0°.

Plates

The approach used here is similar to that described previously for membranes (plane stress state). The bending and torsional moments in a plate enable to determine the normal and tangential stresses on the upper and lower surfaces of the plate. These stresses are equal in absolute value, therefore the formulae (2.6.4) and (2.6.5) become:

$$M(\alpha) = M_x \cos^2 \alpha + M_y \sin^2 \alpha + M_{xy} \sin 2\alpha; \tag{2.6.7}$$

$$M_k(\alpha) = \frac{1}{2}\left(M_y - M_x\right)\sin 2\alpha + M_{xy} \cos 2\alpha. \tag{2.6.8}$$

Besides, the extreme values of the shear forces are determined.

Shells

The approach used here is similar to that described previously for membranes (plane stress state). However, the stresses on the upper and lower surfaces of a shell are calculated with the account of the membrane stresses and bending forces by the following relationships:

$$\sigma_x^{L/U} = N_x \pm \frac{6M_x}{h^2};\ \sigma_y^{L/U} = N_y \pm \frac{6M_y}{h^2};\ \tau^{L/U} = T_{xy} \pm \frac{6M_{xy}}{h^2}, \tag{2.6.9}$$

where h is the thickness of the shell; the indexes U and L indicate whether a value refers to the upper or lower surface of the shell.

Solid Elements

The extreme values of: *a)* the average stress (hydrostatic pressure); *b)* the principal deviator stresses serve as a criterion for determining dangerous combinations of stresses in the general case of the stress and strain state. The angles of inclination of the principal stresses in each element are determined for every loading.

The selection process is organized in the following way. The direction cosines of the principal planes for all loadings are calculated for the given element. Thus, if the model includes n loadings, then $3n$ planes will be found. Next, the stresses S_f on these planes caused by all loadings are calculated, and both negative and positive stress values are accumulated. In this regard, the criteria are designated by three-digit numbers. The first two digits denote the number of the loading the planes of which are used to calculate the

stresses caused by all loadings. The third digit can have a value from 1 to 6 with the following meaning: the overall stress on the 1*st* principal plane: 1 stands for positive; 2 – for negative;

3 and 4 — the same on the 2nd principal plane;

5 and 6 — the same on the 3rd principal plane.

For example, the criterion 143 means that the maximal positive stress appears on the 2nd principal plane of the 14th loading. The criterion 076 means that the maximal negative stress appears on the 3rd principal plane of the 7th loading.

The criteria corresponding to the maximal and the minimal values of the average stress are designated by the digits 7 and 8, respectively.

Loadings

The design combinations of forces are determined taking into account the logical relationships between loadings that reflect the physical meaning of loadings and requirements of various design codes.

There are three types of loadings:

1) *independent* (dead weight, weight of equipment etc.);

2) *mutually exclusive* (wind from the left and wind from the right, seismic actions along different axes etc.);

3) *accompanying* (braking loads provided there are vertical crane loads etc.).

It is also possible to specify an alternating loading while keeping its vector absolute value constant.

2.7. Erection

The process of creating a complex structure is, generally, a multi-phase procedure involving a certain sequence of structural assembling operations. The latter may include successive works on installation or removal of some members of the structure, installation/removal of ballast weights, regulation of the length of certain elements, modification of constraints etc.

Most actions performed during the erection modify the design model and/or the stress-strain distribution in the system.

Elementary Actions

PC SCAD has a certain set of elementary actions the sequence of which constitutes all needed transformations of a design model to determine the stress/strain distribution in the course of the erection and prestressing of the structure.

So, our set of elementary actions includes the following items:

(**A**) *Loading the system by a known action* which is a set of given loads, dislocations and thermal effects

(**B**) *Installing an external constraint* in a node of the system to restrain it from a particular incremental displacement or rotation. It should be noted that here we require the increment of a displacement rather than the displacement/rotation itself (i.e. making it zero) to be prohibited.

(**C**) *Adding an internal constraint* between the nodes of the system prohibits the change of a certain mutual displacement or rotation of these nodes (including the action of combining the displacements). We also consider the prohibition of further (at subsequent stages) changes of mutual displacements and this contributes to the closure in the system of deformations inconsistent in the previous stages.

(**D**) *Removing an external constraint*. In this case the stress/strain distribution changes together with the model. The reason for this is that generally the constraint can be stressed. Therefore, it is necessary to zero the stress in the constraint before removing it.

(**E**) *Removing an internal constraint (rejection of combining the displacements)*. In this case the stress/strain distribution changes together with the model. The reason for this is that generally the constraint can be stressed. Therefore, it is necessary to zero the stress in the constraint before removing it.

(**F**) *Mounting an element* of any type.

(**G**) *Removing an element from the system* both modifies the design model and makes it necessary to recalculate the stress/strain distribution. The transformed system must not experience any influence of its removed part, therefore this elementary action consists of two essential steps:

- making the interaction forces between the removed and remaining parts of the system equal to zero;

- modifying the design model.

The considered elementary action is a kind of generalization of the “removing an external constraint” action. Another approach can be used to implement it: imagine that the elements (constraints) are removed from the system, and their influence on the remaining part of the structure is replaced by forces N_j which existed in the removed elements (constraints), and these forces are then suppressed by applying the loads $P_j = -N_j$.

Finally, it should be noted that the removal of an element (removal of a constraint) is treated not as a failure, but as an active intervention in the structure during its construction. So the temporary restraints and props installed during the intermediate erection stages are removed. They are first destressed (wedges supporting these props are driven apart, or similar actions are performed).

(**H**) Modification of the elastic modulus of the element for the current and all subsequent erection stages, which can be used to simulate the process of concrete hardening, wear of the structure and other phenomena, leading to a change in the element stiffness. It should also be noted that this is not about modeling the loading process when the changes in the stress state affect the tangent modulus of elasticity, but about accounting for some of the non-mechanical effects, including conditional "destruction of rigidity" in the process of "death" of the element.

While the concrete hardening usually does not require the recalculation of the SSS, the stiffness reduction requires a different approach. This option is implemented in **SCAD** in the following way:

- a new unstressed element with modified rigidities is installed next to the old stressed one;
- the old stressed element is removed and its number is assigned to the new one.

I) Specification or modification of the subsoil parameters for the current and all subsequent erection stages. While the recalculation of the SSS is usually not required when the stiffness increases, a different approach is required when it decreases. Having “supposedly” divided the elements into two part: the shell and the elastic subgrade, independently determined the reactions of each of them from the general reactions, we follow the same procedure as in the previous case.

Agreement on the Loads

Actions on the system defining the nature of its stress-strain state are divided into two classes:

- *basic* (accumulated) actions for which the stress-strain state is transferred to the next erection stages;
- *independent* loadings acting only in the current erection stage and not related to other stages.

The dead weight and the prestress are typical examples of the basic action, and the wind load on the building being erected may serve as an example of an independent action.

SCAD system implements an agreement that all basic actions have to be included in a single, predetermined (e.g., the first) loading. The composition of this loading may vary at different erection stages due to the inclusion or exclusion of certain groups of loads.

Variants of the Model Modification

Two methods of removal and inclusion of elements into the system ("birth" and "death") can be considered. These variants are illustrated in Fig. 2.7.1.

In the first case (step F) a structure different from the previous one without the removed elements, consisting only of the included elements is considered. Here, addition of a new element into the design model differs in that a part of its nodes is attached to the structure assembled under the load, and other nodes are assumed to be located in their design position, which leads to the broken design model.

In the second case (step H) "dead" elements are not removed. Instead, it is inactivated by multiplying their stiffness by the significant reduction factor, for example, 10^{-6}.

Calculation according to the first method is particularly relevant for the design of multi-storey buildings with a reinforced concrete frame when during the construction of each floor the formwork is set in such a way that the top surface of the concreted slab is horizontal. Essentially, it is the design column length increased by the subsidence value of the part of the building that has already been erected that is corrected.

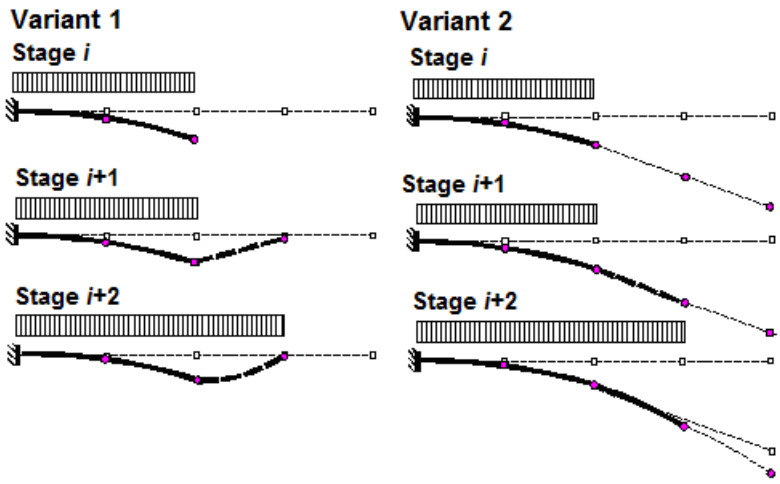


Figure 2.7-1. Two variants of building up the structure

Calculation according to the second method enables to model the "junction along the tangent line" (see Fig. 2.7-1, variant 2), which is appropriate for the analysis of structures, the construction of which is performed by the cantilever construction method.

Finite Element Procedures

At each step of the erection, any classical structural analysis method can be involved. Seeing the specific nature of the multi-step analysis, we think it is useful to represent these methods in such form as to reflect the variability of the system. For example, the governing equations of the slope deflection method will be

$$\boldsymbol{K}^{(r)}\,\Delta\boldsymbol{Z}^{(r)} = \Delta\boldsymbol{f}^{(r)}, \tag{2.7.1}$$

where $\boldsymbol{K}^{(r)}$ is the system's stiffness matrix at r^{th} step, $\Delta\boldsymbol{Z}^{(r)}$ and $\Delta\mathbf{f}^{(r)}$ are respective vectors of incremental displacements and incremental nodal loads related to the rth step. Knowing $\Delta\boldsymbol{Z}^{(r)}$, we can determine the stress increments $\Delta\boldsymbol{s}^{(r)}$ and obtain values of the displacements $\boldsymbol{Z}^{(r)}$ and stresses $\boldsymbol{s}^{(r)}$ accumulated at all r steps

$$\boldsymbol{Z}^{(r)} = \boldsymbol{Z}^{(r-1)} + \Delta\boldsymbol{Z}^{(r)}, \tag{2.7.2}$$

$$\boldsymbol{s}^{(r)} = \boldsymbol{s}^{(r-1)} + \Delta\boldsymbol{s}^{(r)}. \tag{2.7.3}$$

This step-by-step summation of the stress/strain components should be performed because of the system's memory mentioned above. The relevant entitlement for the relations (2.7.2) and (2.7.3) could be *laws of inheritance of erection states* of a structure. The genetic nonlinearity of the problem is caused by the requirement that the linear relations (2.7.1) and the laws of inheritance (2.7.2) — (2.7.3) must hold simultaneously.

The transition to the next (r + 1)-th stage of the calculation changes the stiffness matrix $\boldsymbol{K}^{(r)}$, which receives an increment $\Delta\boldsymbol{K}^{(r)}$, positive – when elements are added and negative – when they are removed, i.e.

$$\boldsymbol{K}^{(r+1)} = \boldsymbol{K}^{(r)} + \Delta\boldsymbol{K}^{(r)}, \tag{2.7.4}$$

or more extensively:

$$\begin{bmatrix} K_{11}^{(r)} & K_{12}^{(r)} & 0 & 0 \\ K_{21}^{(r)} & K_{22}^{(r+1)} & K_{23}^{(r+1)} & 0 \\ 0 & K_{32}^{(r+1)} & K_{33}^{(r+1)} & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} = \begin{bmatrix} K_{11}^{(r)} & K_{12}^{(r)} & 0 & 0 \\ K_{21}^{(r)} & K_{22}^{(r)} & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} + \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & \Delta K_{22}^{(r)} & \Delta K_{23}^{(r)} & 0 \\ 0 & \Delta K_{32}^{(r)} & \Delta K_{33}^{(r)} & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

The system of governing equations of the (r + 1)-th stage is as follows:

$$\begin{bmatrix} K_{11}^{(r)} & K_{12}^{(r)} & 0 & 0 \\ K_{21}^{(r)} & K_{22}^{(r+1)} & K_{23}^{(r+1)} & 0 \\ 0 & K_{32}^{(r+1)} & K_{33}^{(r+1)} & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \cdot \begin{bmatrix} \Delta Z_1^{(r+1)} \\ \Delta Z_2^{(r+1)} \\ \Delta Z_3^{(r+1)} \\ 0 \end{bmatrix} = \begin{bmatrix} 0 \\ \Delta f_2^{(r+1)} \\ \Delta f_2^{(r+1)} \\ 0 \end{bmatrix} \tag{2.7.5}$$

In some cases, the changes of the design model deal with the adjustment of the parameters of some of the previously assembled elements (change in the modulus of elasticity, the subsoil parameters, etc.). In this case we can assume that a previously entered, for example, at the s-th stage stiffness matrix $\Delta\boldsymbol{K}^{(s)}$ is replaced by a new matrix $\Delta\boldsymbol{K}^{(r)}$:

$$\boldsymbol{K}^{(r+1)} = \boldsymbol{K}^{(r)} - \Delta\boldsymbol{K}^{(s)} + \Delta\boldsymbol{K}^{(r)}, \tag{2.7.6}$$

It is obvious that influences related to different steps of the same erection phase are described by conventional linear mechanical laws, and the design model of a structure changes only from one erection phase to another. In this connection, we imply that the displacements $\Delta\boldsymbol{Z}^{(r)}$ and stresses $\Delta\boldsymbol{s}^{(r)}$ are those accumulated since the completion of the last step of the previous erection phase.

It should be noted that in certain cases some loads work only at r^{th} step of the erection and are removed at the steps that follow. For example, this situation is typical for a cantilever erection of a structure where the weight of the crane equipment is taken into account when forming the vector $\Delta\boldsymbol{f}^{(r)}$ the cranes being arranged exactly as this step of erection requires. As the new (r+1)th erection step begins, the nodal load vector is formed for a new arrangement of the crane equipment, and one should remember to apply also negative loads at the step (r+1) to cancel the crane load upon the system applied at the previous step. If one does not do it, the inheritance laws (2.7.2) and (2.7.3) will fail to hold.

Thus, the following operations are performed at each erection stage: installation (removal) of individual structural elements, or their groups, regulation of the actual sizes of the elements of the bearing structures, introduction (removal) of temporary constraints, modification of the parameters of the system constraints with the external environment, etc. The calculation technology that takes into account the above-mentioned circumstances is based on the principle of the stepwise tracking of the changes in the basic parameters of the design model (geometry, stiffness properties of the model elements and constraints, loading and deformation) with the closure of the system at each (predefined) stage of the building erection. A clear distinction is made between the total stress strain state (SSS) of the system that occurs at each erection stage with the account of all the previous stages, and the SSS increment caused by the additional effects on the system related solely to the considered erection stage.

All the calculations are performed under the conventional assumptions of linear structural mechanics for each erection stage. However, in general the problem becomes nonlinear due to the modification of the design model during the transition from one erection stage to the other.

2.8. Finite Element Models

Convergence of FEM

The theory of the finite element method deals very much with the problem of convergence, i.e. an asymptotic behavior of estimates of accuracy of an obtained approximate solution as the finite element mesh becomes infinitely dense. A number of important convergence theorems have been stated. For example, for consistent elements it has been determined [15, pp. 195–196], that if (k–1) is the degree of a polynomial used within a finite element for approximating its displacements and for obtaining an approximate solution of an elliptic boundary-value problem of order 2m expressed through the displacements u_{h}, then the error of energy in comparison with the exact solution u is

$$U(u - u_{\text{h}}, u - u_{\text{h}}) \le Ch^{2(\text{k-m})}\|\text{u}\|^2_{\text{k}}, \tag{2.8.1}$$

where h is the maximum value of the relative size of the element (the spacing of the mesh).

The following error estimates take place for s-th derivatives of z:

$$\|\text{u} - u_{\text{h}}\|_{\text{s}} \le Ch^{\text{k-s}}\|\text{u}\|_{\text{k}}, \quad \text{if } s > 2m - k; \tag{2.8.2, a}$$

$$\|\text{u} - u_{\text{h}}\|_{\text{s}} \le Ch^{2(\text{k-s})}\|\text{u}\|_{\text{k}}, \quad \text{if } s \le 2m - k. \tag{2.8.2, б}$$

Similar estimates for inconsistent elements have been obtained in a series of papers by I.D. Evzerov and V.S. Karpilovsky (see, for example, [5, 8]).

The data on the values of the convergence parameter enable to assign an approximate density of the finite element mesh on the basis of the following peculiar considerations [2, p.55]: "…let us just note that the convergence takes place assuming some natural limitations of initial data and the meshing of the area are satisfied, and the error of the determination of stresses/strains is of the order *Ch/L* where *C* denotes a constant depending on the shape of the area; *h* is the mesh spacing; *L* is a characteristic size of the area. This estimate should guide the definition of the mesh spacing, depending on the desired accuracy (*average*). For example, if you want the accuracy of the approximate solution to be 5%, choose the spacing of the mesh of around 1/20 of its characteristic size…", i.e. a characteristic two-dimensional spot would include about 400 nodes, and in the case of a three-dimensional problem there would be about 8,000 nodes.

It should be remembered that the above consideration is true *only in average*, and the accuracy depends much not only on the finite element type (constants *C* and *k*) but also on the position of the finite elements and their orientation with respect to the flows of principal stresses. The latter fact can be seen from a characteristic example taken from the paper [7] and shown in Fig. 2.8-1.

One should remember that the afore-said estimates of the convergence rate are oriented at the determination of the asymptotic properties of the solution, while a practical designer is interested in the accuracy of an approximate solution obtained on a particular finite element mesh. Of course, in most cases the asymptotic convergence is accompanied by an appropriate "practical convergence". The latter term will mean the possibility of obtaining an acceptable accuracy with a comparatively rough meshing. However, this rule has its exceptions too. In this regard, we would like to cite from a great mathematician and physicist A. Poincare (quoted from [1, p.52]):

"...of two series where their general terms are $1000^n/n!$ and $n!/1000^n$, mathematicians would name the first one to be convergent… because the millionth term is much less than $999{,}999^{th}$, whilst the second term would be treated as divergent for its general term may grow unlimitedly. Astronomers, on the contrary, would take the first series for a divergent one because its first 1,000 terms go increasing; and the second series would be called by them convergent because its first 1,000 terms go decreasing and this decrease is quick enough at the beginning." And then follows a breathtaking conclusion: "Both views are valid: the first one in theoretical investigations, and the second one in computational applications".

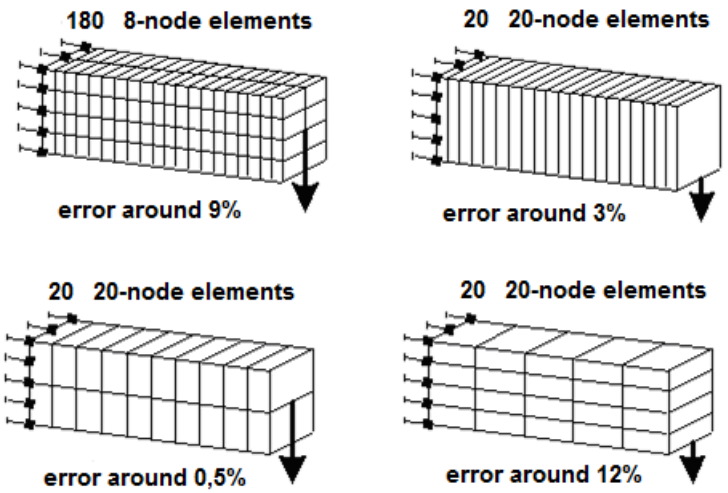


Figure 2.8-1. Different finite element meshes and the accuracy of the analysis

Apparently, when solving any responsible problem, an analysis of the solution quality must be performed. The solution can be checked by solving the problem repeatedly with a different finite element mesh. Certainly, it is hardly reasonable to solve a whole big problem on gradually thickening meshes, but this analysis obviously makes sense for characteristic fragments of a design model. The empirical fact of the solution stability at the mesh thickening is a rather convincing argument "pro" this approach.

All that has been said must not be treated as an appeal to the pure empiricism. Theoretical investigations of convergence are very important, and their results can be used in practice. However, there are some serious problems the designer should take into account. One of them is that a satisfactory

practical convergence of displacements does not necessarily imply a good convergence of internal forces and stresses the designer is interested in. Those are determined by the differentiation of the displacements, and the differentiation is an incorrect operation in the sense that a slight variation of a function value may cause a substantial variation of the function derivative.

Thus, validation of practical convergence must be oriented at investigating results important for a particular problem to be solved. Here is, for example, a characteristic excerpt from a well-known monograph by O. Zenkevich: "The spacing of a mesh needed to obtain an acceptable accuracy in shell problems may depend on a number of reasons. It is often the case that a small thickness of the shell limits the area of bending moments to an edge zone where the moments change substantially. The membrane forces can be calculated with a high accuracy even on a rough mesh, while the variation of the bending moments require an extensively fine mesh for being described properly." [6, p.257].

There is also a difficulty of comparing the stresses obtained on meshes of different densities: the stresses are often determined in centers of finite elements, and a certain effort is needed to compare stresses at the same points.

Moreover, when using some types of finite elements (such as triangular elements with the linear displacement approximation for plane elasticity), the stress fields are piecewise constant functions, and the areas of their constancy coincide with the mesh triangles. Values of the stresses determined with these elements vary intensively from one element to another, therefore usually one applies an averaging of the stresses by elements of a star and refers those to a nodal point. The comparison of such stress fields is encumbered with the operation of averaging.

Organizing the validation of the practical convergence, one should remember that the problem being solved may have unpleasant peculiarities due to an incorrect idealization of the structure. A typical example is the idealization of a load as a concentrated force (practically unfeasible situation) which may cause such features of the problem solution as infinitely big values (a logarithmic singularity in the expressions of bending moments in a plate under a concentrated force) or high gradients of the stress field.

Thus, the practical convergence check should be performed with examples similar to the problems of interest but having accurate solutions. In this case the interpretation of the testing results becomes more informative. Some problems of this kind are discussed in the next section.

Convergence Validation for Some Models

Below, a comparative analysis of a simply supported square plate, loaded by a uniformly distributed load throughout its area, is given. The analyses were performed with four finite element meshes: 4×4, 8×8, 16×16 and 24×24 (Fig. 2.8-2).

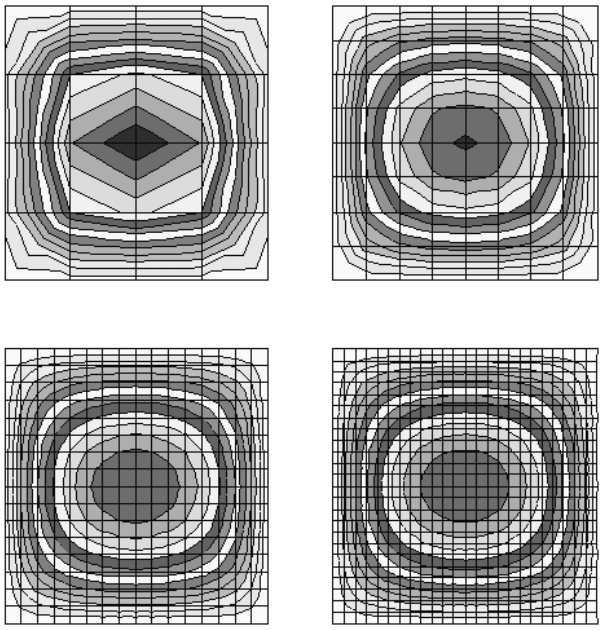


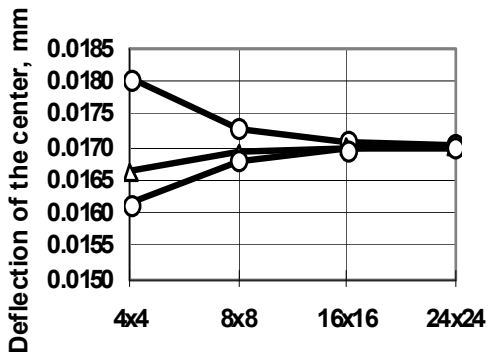
Figure 2.8-2. Isofields of bending moments obtained with different design models and finite element meshes

Table 2.8-1 contains results regarding displacements, bending moments, and shear forces obtained for the finite elements of different types on the aforesaid meshes. The same data are shown as plots in Fig. 2.8-3.

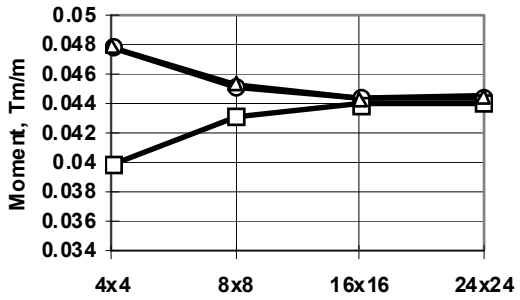
Table 2.8-1

	Displacements of the plate center for the mesh			
FE type	**4x4**	**8x8**	**16x16**	**24x24**
11/41	0,0180329	0,0172754	0,0170823	0,0170453
20/50	0,0166168	0,0169137	0,0169918	0,0170051
12/42	0,0161403	0,0168034	0,0169658	0,0169938
	Moment at the center of the plate			
M -11/41	0,04781	0,04509	0,04443	0,04443
M -20/50	0,03991	0,04313	0,04393	0,04408
M -12/42	0,04787	0,04528	0,04432	0,04448
	Shear force at the edge			
Q -11/41	0,22	0,28	0,31	0,32
Q -20/50	0,37	0,4	0,43	0,44
Q -12/42	0,24	0,31	0,33	0,34

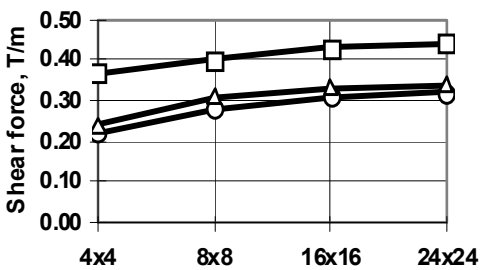
a)



b)



c)



Legend:

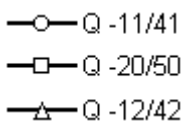


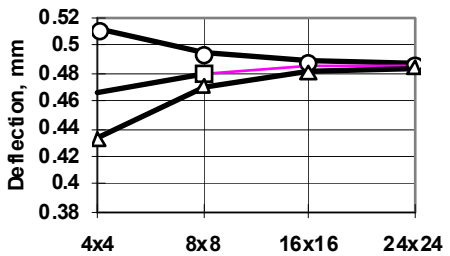
Figure 2.8-3. The convergence of results for a distributed loading:
a — by deflections; *b* — by moments;
c — by shear forces

As one can see in Table 2.8.1 and Fig. 2.8-3, the practical convergence takes place for the deflections and bending moments with the use of finite elements of different types. For the shear forces, elements of the 11th type give values noticeably different from those obtained by the use of other finite elements. It should be noted that the element of the 20/50 type was connected to four nodes only, though it was possible to introduce more nodes on its sides (up to 8 nodes in total). Checkup calculations of this system showed that the accuracy of the results improved significantly and became close to that obtained with meshes twice as dense. For example, for a mesh of 8×8 elements the deflection was 0.01701, the bending moment was 0.0442, and the shear force was 0.278.

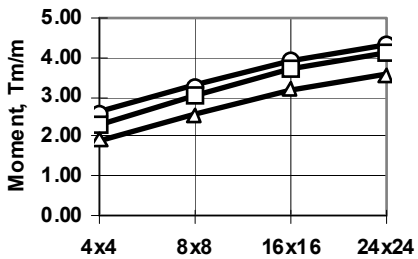
Table 2.8-2

	Displacements of the plate center for the mesh			
FE type	**4x4**	**8x8**	**16x16**	**24x24**
11/41	0.511522	0.494164	0.488470	0.487183
20/50	0.466266	0.480460	0.484425	0.485222
12/42	0.432918	0.470046	0.481375	0.483493
	Moment at the center of the plate			
M -11/41	2.61566	3.27276	3.93364	4.32066
M -20/50	2.31761	3.04494	3.72290	4.11309
M -12/42	1.89259	2.52465	3.17713	3.56252
	Shear force near the center			
Q -11/41	7.26	14.58	29.18	43.77
Q -20/50	6.50	13.31	26.81	40.26
Q -12/42	11.37	25.59	53.21	80.42

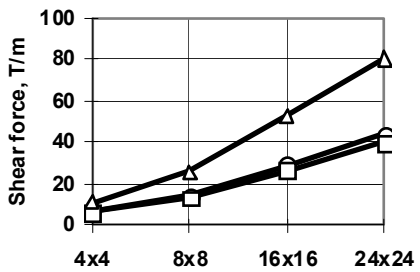
a)



b)



c)



Legend:

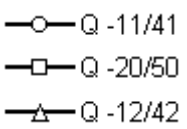


Figure 2.8-4. The convergence of results for a concentrated force:
a — by deflections; *b* — by moments;
c — by shear forces

In another series of numerical experiments, when the same plate was loaded by a concentrated force, results shown in Table 2.8-2 and in Fig. 2.8-4 prove to be less optimistic. Here the practical convergence rate of the moments slows down, and that of the shear forces does even more significantly. The values of the latter were taken in a point located at a quarter of the plate thickness from its center. Apparently, values of the shear forces in points so close to the location of the concentrated force should not be taken into consideration at all. This issue is discussed in more detail in sections that follow.

It should be noted that the quicker convergence of results for some finite element types is paid for by a noticeably longer computation time. Therefore, the loss of time caused by the use of those elements and the computation time spent for the solution of the same problem on a finer mesh with elements of another type should be compared.

Circumventing Singularities

In the vicinity of singular points — where a drastic stress concentration occurs — the use of finite elements (as well as other discretization techniques) is usually encumbered, especially in the representation of the

stress field. The finite element mesh has to be thickened dramatically, thus considerably increasing the dimension of the analysis.

However, the thickening of the mesh may not lead to a proper result, and this fact makes us perform a deeper analysis of the situation (see, for example, Fig. 2.8-4, *c*). One of the most popular judgments is the following: a concentrated force is an abstraction that does not exist in reality. If it had been actually implemented, then it would pass through a structure of any strength and run away to infinity without being resisted. So it turns out that this idealization creates an artificial difficulty which may be fought heroically only to find that this heroism is of very little practical sense. It should be remembered how the force represented as a concentrated one in the model is implemented in the actual structure, then all issues of how to provide the convergence of the finite-element solution to an exact one might be resolved (see, for example, "Coupling Elements of Different Dimensionality" section).

Fragmentation

If the behavior of the solution near singular points is of interest nonetheless, then the local refinement of the design model has to be performed. A typical example is a concentrated force applied to a plate, when the stress state in a small vicinity of this force is essentially spatial, and the usual assumptions of the plate theory are not applicable. It is possible to switch to a 3D model, but if we replace all plate finite elements by solid ones, the dimensionality of the model will increase dramatically. Thus we need to combine a two-dimensional idealization of the object with some refinements in three dimensions. The simplest way to do it is to use a fragmentation and a global-local analysis. Generally speaking, there are three approaches to this analysis [23]: (1) a force method in which an extracted fragment is subjected to forces transferred from the rest of the system which have been found by a global analysis; (2) a displacement method in which the fragment boundary is displaced in the same way as in the global analysis; (3) a mixed method. We will demonstrate the procedure of the first approach.

In this case and similar ones, it is natural and convenient enough to use the following two-step procedure:

(a) neglecting local peculiarities of the design, build a rough first-iteration design model of a full structure to enable us to estimate the stress and strain state of the object as a whole, and perform the analysis;

(b) extract a fragment from the structure that contains the peculiarity of our interest. Apply reactions to this fragment obtained by removing the rest of the structure, and forces directly imposed on the extracted fragment. Use a more detailed design model to analyze the fragment. Of the solution thus obtained, use only a part that describes points at a certain distance from the boundaries of the fragment.

This approach is concordant with the practice of selecting a series of design models for the analysis of various peculiarities of the structural behavior. However, it requires a bit of intuition and experience in order to exclude the danger of an irreducible error that appears in the analysis of a rough model.

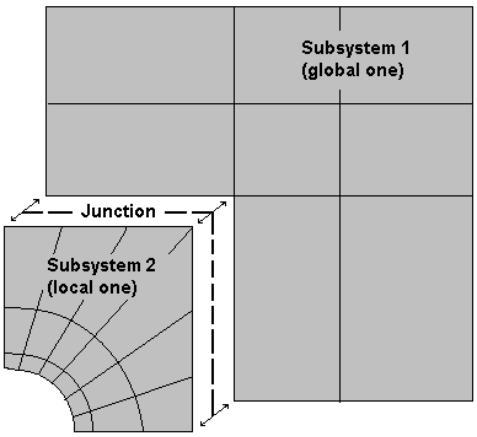


Figure 2.8-5. A joint design

In the course of the fragmentation technique application, it may turn out that the fragment nodes at a joint of the fragment with the main structure do not have their counterparts on the other side of the joint (Fig. 2.8-5). The situation of inconsistency between the meshes on two- and three-dimensional finite elements located at different sides of a joint may also arise. Thus there arises a problem of creating special finite elements to match joints at the boundary between a fragment and the rest of the system, or using other special joining techniques [24]. **SCAD** has elements that can handle this: types 20, 30, and 50 which are capable of adding more nodes at sides of rectangles.

An important role can be played, seemingly, by a mesh smoothing optimization (a gradual change of its elements' sizes).

Counteracting the Dimensional Instability

The following mistake often occurs in the design practice: the engineer understands perfectly well that the model behaves in a certain way (for example, it deforms in one plane only), but fails to remember that he has assigned a different index of type to it (for example, a general spatial system). The structure becomes unstable, and some unfavorable consequences arise. A typical example of this mistake is a design model in Fig. 2.8-6 — a foundation slab with adjoining walls. The model was geometrically unstable, and displacements of the whole system as a rigid body along X and Y axes were possible.

The results of the analysis of this unstable system showed the accuracy check error of 99%. It can be explained by the fact that though **SCAD** eliminates the geometrical instability in the course of the computation, this operation can lead to the loss of accuracy due to the incorrect strategy of the instability elimination. It is better to eliminate the instability carefully. In the considered case the displacement of the system as a rigid body should be prohibited, and an external statically determinate system of constraints should be used for this purpose.

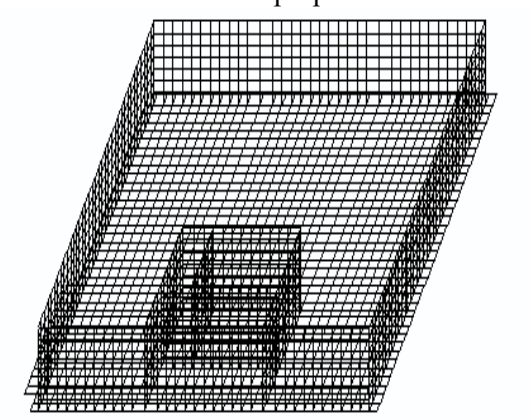


Figure 2.8-6. Design model

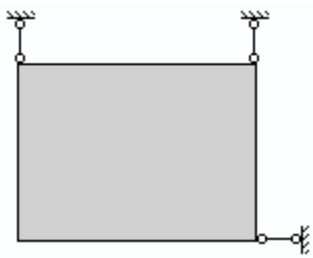
After these changes had been made to the design model, no further problems with the accuracy appeared.

The procedure of imposing nodal constraints is not difficult technically. What may be a problem is how to select a reasonable set of constraints that affect the geometrical stability of the considered design model or its fragment. A typical example is a plate on elastic subgrade sufficiently restrained out of the bending plane but capable of moving in this plane as a rigid body. The engineer understands that the plate must be restrained in the plane, but does not always select the best type of fixation.

Firstly, the fixation should be designed in such a way that the system becomes statically determinate externally (three constraints must be placed in the plate plane). If this is achieved, the reactions in the additional constraints must be zero because there are no loads in the plane of the plate. This consideration is correct only in theory, because it does not take into account the inevitable error of the numerical solutions. The rounding errors are accumulated and make the reactions nonzero, though very small in magnitude. This means that the actual numerical representation can be interpreted in such a way as if a small nonzero force were applied in the plane of the plate.

Since the said force is resisted by the conditional constraints, the values of the obtained reactions happen to depend on the way these constraints have been imposed. If the constraints are placed as far as possible from one another (Fig. 2.8-7, *a*), then the reactions will be small and will not affect the stress state much. However, if the plate is restrained in one point (Fig. 2.8-7, *b*) against two displacements and a rotation, which is theoretically possible, the effect on the stress state can be noticeable, and the results of the analysis in the vicinity of this node will be distorted.

a)



b)

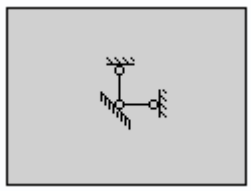


Figure 2.8-7. Imposing constraints to counteract the geometrical instability:
(*a*) recommended design; (*b*) unfavorable fixation

All that was said above applies to any fixation of a free (capable of moving as a rigid body) fragment of a system.

Coupling Elements of Different Dimensionality

It is only a rare case that a whole structure can be represented by elements of the same type (such as plate elements). More frequently the same design model may include bars and shells, and other types of elements.

SCAD allows almost any combination of finite elements to be employed, the elements being of various types, dimensionalities, sizes and geometric shapes, and with various numerical stiffness properties. However, this software flexibility contains a lot of dangers, sometimes explicit, sometimes hidden thus especially insidious. Such situations may arise, for instance, when a design model includes both one-dimensional (bars) and two-dimensional (plates and/or shells) members. As will be shown below, this merging of a variety of different elements in their common design model requires careful attention, especially in places where the said elements of different dimensionality are joined.

Bars + plates. Let's take as an example of combined plate and bar elements the analysis of a spatial framing together with a slab foundation.

So, let us consider a discrete design model combining plate finite elements and bars rigidly attached to the slab. The fact that the slab lies on elastic subgrade is of no importance for the discussed problems. The finite element mesh is formed in such a way that the bars of the structure's frame hit the nodes of the meshed slab. If no additional measures are taken, the design model described above will provide the consistency of both vertical displacements of the slab and frame (in the direction perpendicular to the plate) and the respective slopes in nodes where the plate and bar elements join one another. However, bending moments in cross-sections of the columns adjoining the slab, calculated with this model, have nothing to do with the real distribution of internal forces. This being the case, the distribution of the internal forces in other elements of the structure's frame will be affected too.

Indeed, let us imagine how the mesh of the slab thickens, and the user expects further refining of the accuracy of the analysis. However, starting from a certain density of the mesh, the further division of the mesh results in the decrease of absolute values of the bending moments in the bars in places of their attachment to the slab. When passing to a limit, as the maximum size of the mesh cell tends to zero, these bending moments should tend to zero too. This means that this design model provides a hinged rather than rigid attachment of the frame elements to the slab. The fact that the user obtains some formal nonzero numerical values of the bending moments on a specific mesh shows only a discretization error of the finite element method, but there is no reason at all to take the discretization error for a credible numerical result!

As can be seen, the bending moment in a bar of this model, independent of the mesh spacing, is transferred to the slab as a moment concentrated in a node of the mesh. On the other hand, the slab under the concentrated bending moment acquires an infinite slope at the point of its application. Thus the slab does not resist the concentrated slope and does not restrain the frame elements.

In order to improve the design model, it is necessary to consider the design of the joint between the frame and the slab. If the columns of the frame are attached to the slab by pedestals, then the latter can be treated as rigid bodies not changing their sizes whatever modifications are made to the finite element mesh.

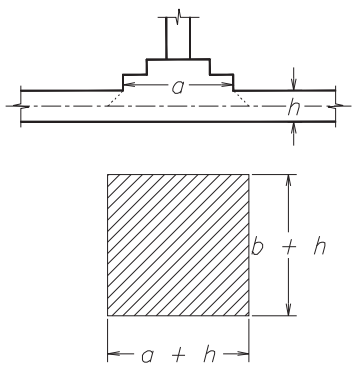


Figure 2.8-8. An attachment of a column by a pedestal: *a* and *b* are plane sizes of the pedestal foot

The size of the rigid body itself can be assigned on the basis of the sizes of the pedestal ($a \times b$), allowing for the distribution of its pressure on the slab at the angle of 45° (Fig. 2.8-8) down to the middle surface of the plate which conforms to the common engineering practice. This situation can be also simulated by introducing a cross-shaped support from bars of sufficiently high rigidity.

In this model the concentrated moment from the frame columns is transferred to the slab through a stiff washer the sizes of which $a+h$ and $b+h$ do not depend on the spacing of the finite element mesh, and therefore they do not depend on the discretization error as well. Though the discretization error will affect the numerical values of the bending moments in the columns, it will not cause the design model to become so fatally different from the real structure as it used to be in the initial problem formulation when a rigid body was not added

to the design model. In case there are no pedestals, the sizes of the column cross-section can be used as those of the rigid body, taking into account the distribution of the pressure on the slab at 45 degrees as shown in Fig. 2.8-8.

Finally, we would like to draw the reader's attention to the circumstance related to a joint analysis of the spatial frame and the foundation slab. Assuming columns of the frame to be orthogonal to the slab plane, we find that the slab does not resist the column torsion as well as displacements v, w of the columns bottom in the direction of Y and Z axes respectively because the plate finite elements just do not have appropriate degrees of freedom to restrain those linear displacements and torsions.

If there are no other external horizontal constraints in the model (those in the plane Y, Z), then the mechanical system turns out to be unsecured and geometrically unstable, and the software will have to respond to this circumstance somehow.

It should be noted that in a spatial frame no geometrical instability would arise due to it, and the software might lack mechanisms for detecting the errors of this kind. This circumstance should also be kept in mind when building a design model of a complex combined structure.

Bars + deep beams. To analyze a high-rise building including both a frame and stiffening diaphragms under horizontal loads, heterogeneous elements are naturally included in the design model.

If one takes an utterly formal attitude towards the formation of a discrete design model describing the plate by a set of plane stress finite elements and the bar frame – by one-dimensional elements without any additional measures taken, then both the bending moments and longitudinal/lateral forces in girders that adjoin nodes at the boundary between the frame and the diaphragm will be totally unrelated to the reality.

Let the model of the diaphragm use simplest finite elements with two degrees of freedom in each node (linear displacements u and v in two mutually orthogonal directions). Such finite elements do not resist the rotation of an adjoining node because the elements just lack the appropriate degrees of freedom. Therefore these finite elements do not transfer any moments to the adjoining nodes. Since the software will require all equilibrium conditions to be satisfied, including the equilibrium of each node by moments, the girders that join these nodes rigidly will have to transfer zero moments to the nodes. Thus the bending moments in the girders in these nodes will be equal to zero. It corresponds to a design model with the hinged attachment of the girders to the diaphragm, and it does not satisfy the requirement of a user oriented at the analysis of girders clamped in the diaphragm.

What corrections should be made to the original mechanical model or the mathematical formulation of the problem (which is the same thing) in order to describe the real structural behavior adequately? As it was stated before, to do that it is necessary to consider the design of a joint between a bar and a diaphragm in sufficient detail.

For example, let a steel I-beam be embedded partly in a brick wall as shown in Fig. 2.8-9, *a*. Then it suffices to add a one-dimensional bar element that penetrates into the plane stressed wall by an appropriate part of its length, to the respective design model and to its discrete counterpart, as shown in Fig. 2.8-9, *b*.

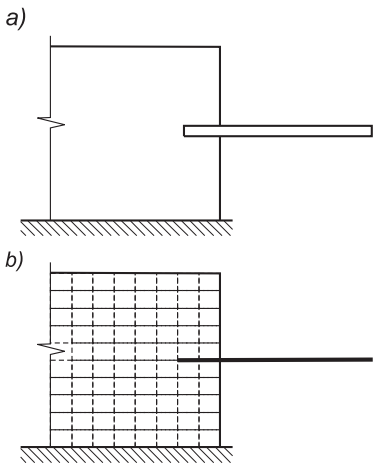


Figure 2.8-9. A cantilever clamped in a wall

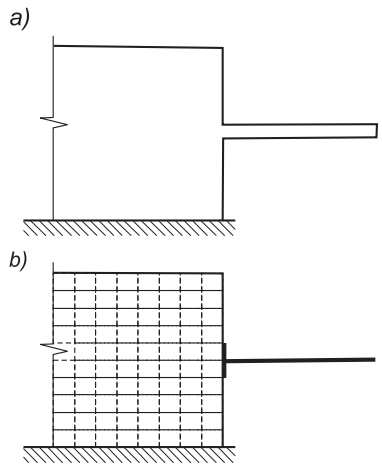


Figure 2.8-10. A monolithic joint

Another design can be suggested for a monolithic joint between a reinforced concrete wall panel and a girder of the building frame — see Fig. 2.8-10, *a*. The actual dimensions of the cross-section can be taken into account here: the height of the girder, with a rigid body placed along the wall border as shown in Fig. 2.8-10, *b*. This rigid body complies with the plane section assumption for the girder itself. The assumption states that the girder section remains flat and does not change its size after its deformation.

Certainly the two ways of building design models given here by no means comprise the wide scope of possible situations. In every particular case the designer should take into account the structural peculiarities of the problem rather than simply using some fancied and questionable general-case models.

The "joint area extension" technique as in Fig. 2.8-9 can be used to match other types of elements of different dimensionality. Fig. 2.8-11 shows such an example wherein a cylindrical shell of a wall and a spherical shell of a cover of a nuclear reactor containment are matched with a massive ring. It should be noted that in places of interpenetration of the structures some perturbation of rigidity properties arises due to the summation of the rigidities of a shell and of the solid body in parts that occupy the same region of space. This fact should be taken into account when assigning the rigidity constants.

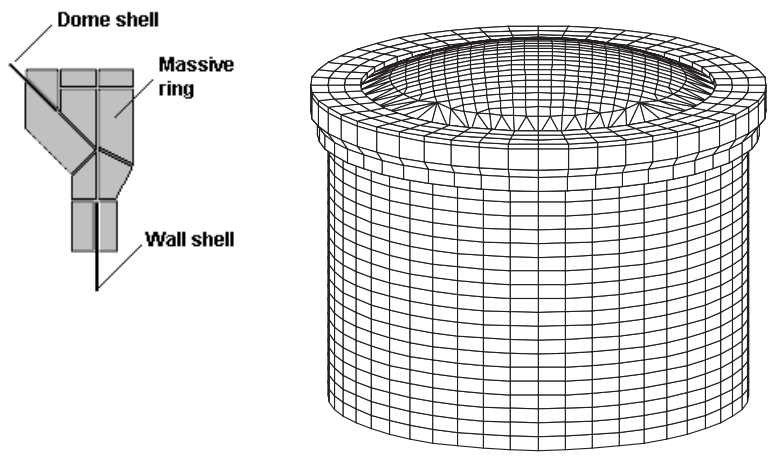


Figure 2.8-11. Coupling a wall and a cover of a reactor containment shell with a massive ring

Effects of Merged Displacements

Restraining of the freedom of nodal displacements in the form of **merged displacements**, i.e. forcing the displacements of one node to be equal to those of the other, is usually used to neglect some deformations of the system. For example, if you merge horizontal displacements of nodes in junctions of girders to columns of a single-storey frame, this action will be equivalent to the use of the hypothesis of perfect rigidity of the girder with respect to tension/compression.

The compliance of floor slabs in a multi-storey frame can be often neglected, treating them as rigid disks. If all horizontal displacement components of all nodes of each floor are merged in such a model, then the idea of a rigid disk will be implemented. However, it should be noted that the torsion of the floor slabs will become impossible, and only their translational displacements will be allowed. This effect may come as an unpleasant surprise for the designer, especially if the considered structure is asymmetric.

It is useful to pay attention to another popular fallacy, best illustrated by a practical example (Fig. 2.8-12). The results of the analysis of a slab showed that there were no lateral forces in the direction of the axis of symmetry (Y) in the stiffening core, while they were present in the other direction (along the X axis). The role of the stiffening core in the behavior of a slab was allowed for by merging displacements in nodes of abutment of the stiffening core to the plate. Linear displacements along the *Z* axis (deflections of the plate) and slope angles U_X, U_Y were merged.

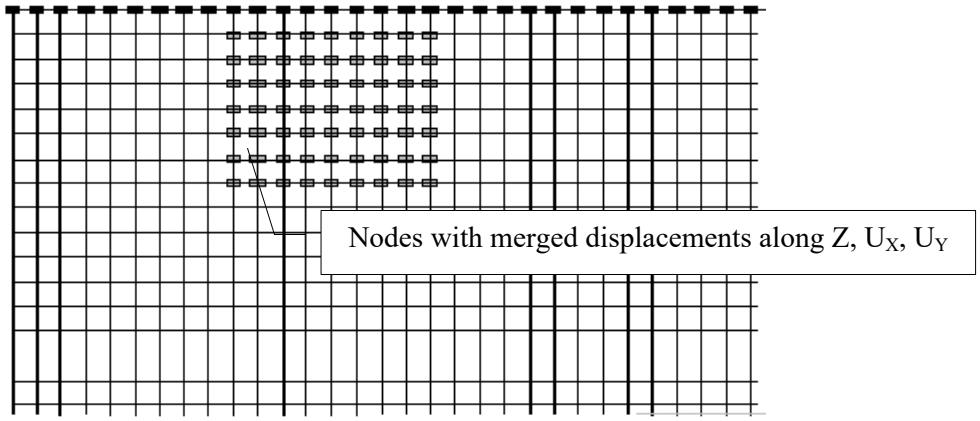


Figure 2.8-12. A fragment of the design model of a slab

The explanation for this result is that no difference was taken into account between restraining a displacement (imposing a constraint) and merging displacements of different nodes. After the linear displacements along the *Z* axis had been merged, the respective part of the plate was able to displace only horizontally, and it seemed that the slopes would be zero too.

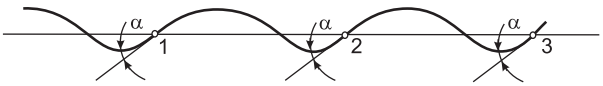


Figure 2.8-13. Merging of slope angles

However, even merging thereof may not assure their equality to zero. This can be illustrated by a simple model (Fig. 2.8-13), where nodes 1, 2, 3 … have the same displacements and **equal but nonzero** slope angles.

In the problem in Fig. 2.8-12 it would be more correct to restrain the nodes of the stiffening core against slopes **UX** and **UY** and merge the linear displacements along the *Z* axis.

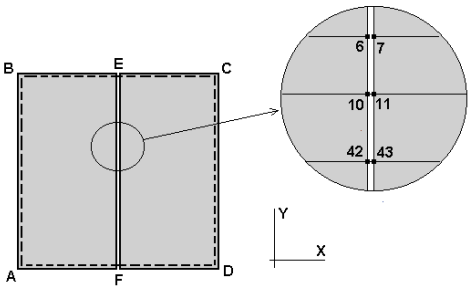


Figure 2.8-14. A plate with a cylindrical hinge

The capability of a partial rather than full merging of displacements can be used to form some special boundary conditions in non-framed structures where "incomplete" attachments to nodes (hinges) cannot be introduced in the way they can be in bar systems.

For example, if you need to describe the structure of a plate ABCD supported along its contour with its two halves connected by hinges along line EF (Fig. 2.8-14), it is sufficient to introduce two systems of nodes (…, 6, 10, 42, … and …, 7, 11, 43, …) attached to the left (ABEF) and right (FECD) halves of the structure, respectively, and merge all displacements but the slope around Y in the adjacent pairs of nodes.

Allowing for a Distributive Ability of Soil

The analysis of structures on elastic subgrade includes the problems of allowing for distributive properties of the subsoil ignored in the simplest Winkler simulation (a piano-key model). Most real soils possess a distributive ability, when unlike the Winkler design model the parts directly subjected to the load are not the only ones resisting it. Therefore, in order to allow for the distributive ability of the subsoil, it is necessary to, first, use the subsoil models other than the Winkler one, and second, introduce the subsoil beyond the foundation area into the respective design model.

In the **SCAD** environment, the subgrade located beyond area Ω occupied by the structure itself can be taken into account by using "infinite" finite elements [16], wedge-shaped or strip-shaped. These elements enable to simulate the entire surrounding of the Ω area if it is convex and polygonal (Fig. 2.8-15).

The specified polygonal area is accurate only to a certain extent. If the Ω area is non-convex or not single connected, it should be complemented to a convex area by finite elements of limited size. The thickness of the slab in the added parts should be assumed zero.

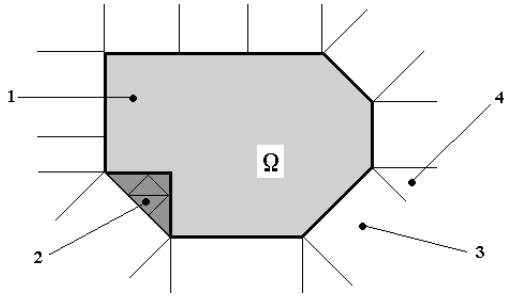


Figure. 2.8-15. Arrangement of the peripheral wedge-shaped and strip-shaped finite elements
1) the slab; (2) a complement of the Ω area to a convex one; (3) a strip element; (4) a wedge element

The use of available finite elements on elastic subgrade (bars, plates, shells) and special peripheral elements only does not enable to create an arbitrary design model of a structure on elastic subgrade. In particular, some difficulties may arise when trying to build a design model of a dam subject to plane strain because **SCAD** cannot use elements like a deep beam on elastic subgrade.

The issue can be resolved in a simple way by including bar elements on elastic subgrade between the dam boundary and the soil. The stiffness of such bars can be zero. In the same way a zero-stiffness slab on elastic subgrade can be put under a massive part of the design model.

In the case when the subsoil parameters are calculated using **CROSS**, peripheral plate elements are not added to the design model. This is due to the fact that **CROSS** solves a problem for an area unlimited in its plan, so the result already allows for the distribution of the subsoil parameters.

Using Rigid Inserts

It is generally known that poorly conditioned stiffness matrices appear often when elements of drastically different rigidities join in a single node of a design model. Let's demonstrate by a simple example how this situation can be interpreted in terms of mechanics. As it happens, poor conditioning is shown by "*almost unstable*" structures.

Fig. 2.8-16 shows a formally stable system with the following stiffness matrix:

$$\mathbf{K} = k\begin{pmatrix} 1+\alpha & -1 \\ -1 & 1+\alpha \end{pmatrix}. \qquad (2.8.1)$$

Eigenvalues of this matrix are $\lambda_1 = \alpha$; $\lambda_2 = 2 + \alpha$, and the condition number is

$$H = \lambda_2/\lambda_1 = 1 + 2/\alpha\ .$$

If the stiffness of the middle spring is big comparing to the stiffness of the end springs, then parameter α is small, and number H is great, which shows the poor conditioning and possible loss of accuracy during the solution of the system of equations with such a matrix.

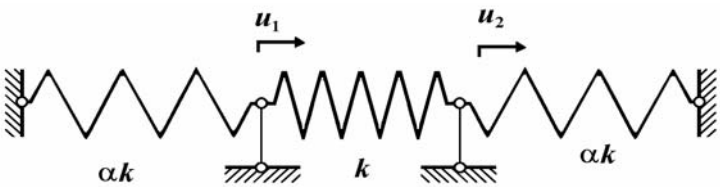


Figure 2.8-16. A system with a poorly conditioned stiffness matrix

It is easy to notice that the mechanical behavior of the considered structure approaches the behavior of an unstable system. Indeed, the displacement of the middle spring as a rigid body may occur if the resistance of the end springs is negligibly small. The reactions of the latter, due to the described relation between the rigidities, cause insignificant deformation of the middle component. However, if we change the ratio of the rigidities to the opposite, then the stiffness matrix will have the conditioning number $H \approx 1$.

Elements of drastically different rigidities meet in the same node again, but the stiffness matrix is well-conditioned and corresponds to an elastic structure (the middle spring) attached to the ground by nearly non-deformable constraints. It should be noted that detection of "near instability" based on the comparison of orders of smallness of possible deformations may show both a lack in the structure and a fault in its modeling.

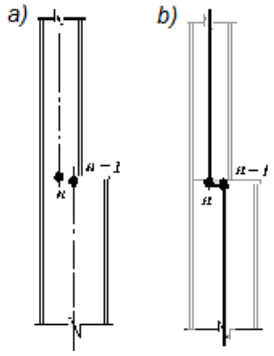


Figure 2.8-17. A stepped joint

Without considering the first case where the designer should obviously modify the structure, we will note that in the second case it is often possible to find a simple solution: declare an element with a very high rigidity to be rigid, and take this feature into account when creating the system of governing equations by introducing the appropriate constraints.

During the analysis of bar systems, an eccentricity of elements joining in nodes often has to be taken account (Fig. 2.8-17, *a*).

Inserting a bar of very big but finite rigidity between nodes *n* and *n*+1, as it seems intuitively possible, will cause, as we have demonstrated above, a drastic deterioration of the calculation accuracy due to a poorly conditioned stiffness matrix.

Rigid inserts at the ends of bar elements can be used to circumvent this computational difficulty. Then the design model has only one node intermediately located anywhere between node *n* and node *n*+1, and the end sections of the adjoining bars are attached to this node by the rigid inserts. No loss of accuracy occurs in this case.

The simplest way is to merge this single node N with either node *n* or *n*+1, then the rigid insert appears at one element only. The payment for this simplification is that internal forces will be determined only in the elastic part of the bar.

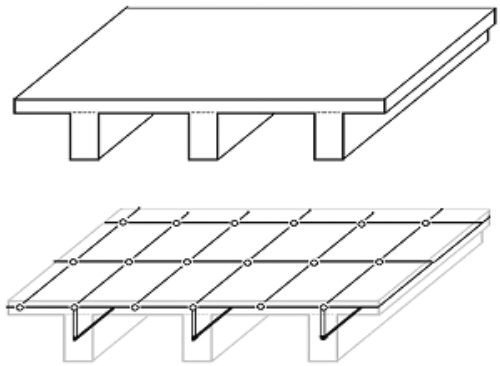


Figure 2.8-18. A ribbed plate

The use of rigid inserts is a common practice when we have to consider a plate or shell reinforced by ribs eccentric with respect to the middle surface. If the ribs are simulated by bar elements, then the eccentricity can be easily and conveniently allowed for by rigid inserts (Fig. 2.8-19). The design nodes are arranged at the plate midplane level. However, you can arrange the nodes at the level of the centers of gravity of the ribs and use the "Move the plate midplane" option, which will lead to the same effect. This method is convenient, for example, in those cases when a stub girder system a part of which is covered by a floor is designed. It should be noted that the eccentricity of the ribs position affects the membrane group of forces, therefore the allowance for eccentricity in a purely flexural structure (constructed from the respective finite elements) gives no results.

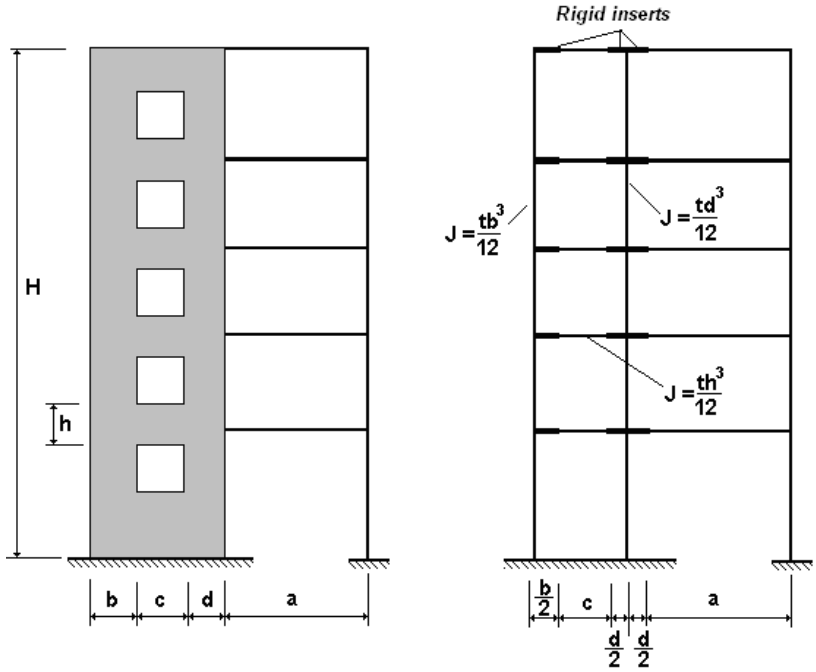


Figure 2.8-19. Using rigid inserts to account for the sizes of nodes

Commonly, the analysis of bar systems involves structures with the cross-section depth of not more than 1/8…1/10 of the distance between nodes. However, there are structures with this ratio of up to 1/5 or even 1/3 (some types of foundations under turbine generators, diaphragms of buildings, hydraulic structures etc.). In this case a bar model with point nodes located at the intersections of element axes is incorrect. To amend the situation, there is a popular approach of taking into account actual sizes of "nodes" by the use of bar elements with rigid inserts. An example of this type of model created in accordance with the recommendations [3], is shown in Fig. 2.8-19.

This technique has been known and used for such a long time that designers rarely doubt whether the hypothesis of a "non-deformable node" is applicable in every particular case. Meanwhile, this question is not trivial, as can be seen by the consideration of results of a sample problem (Fig. 2.8-20).

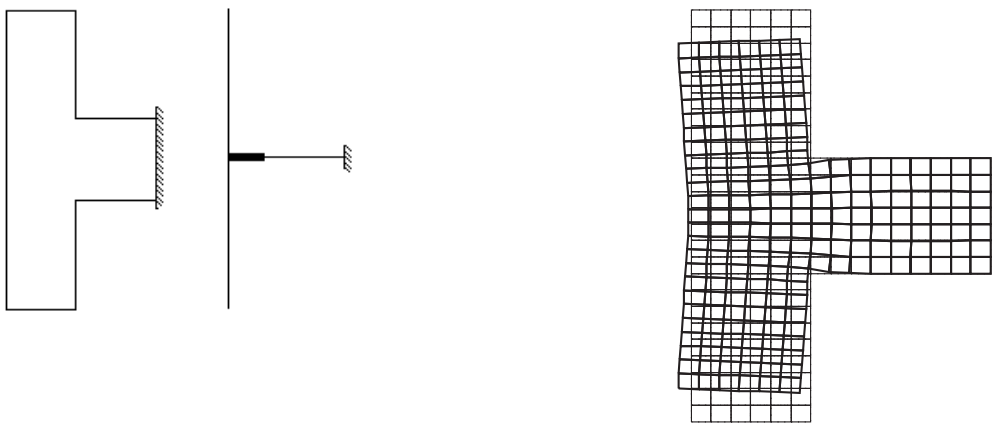


Figure 2.8-20. To the analysis of a node of finite size

Its bar model does not allow horizontal displacements, and the vertical bar does not bend. A more detailed model, however, indicates the presence of horizontal displacements which arise due to a restrained compression along the junction line AB. Since there is no such restraint on the opposite side of the "post", a non-uniform distribution of stresses appears which is equivalent to bending.

References to Chapter 2

1. I.I. Blekhman, A.D. Myshkis, Y.G. Panovko, Mechanics and applied mathematics: Logic and peculiarities of mathematical applications. Moscow, “Nauka”, 1983, 328 p. (In Russian)
2. A.V. Vovkushevsky, B.A. Shoikhet, Analysis of massive hydraulic structures taking into account seam opening. Moscow, “Energiya”, 1981, 136 p. (In Russian)
3. A.S. Gorodetsky, I.D. Evzerov, E.B. Strelets-Streletsky, V.E. Bogovis, Y.V. Genzersky, D.A. Gorodetsky, Finite element method: theory and computing implementation. Software system "Lira-Windows". Kiev, “FACT”, 1997. (In Russian)
4. Dynamic structural analysis. (Designer’s manual.) Moscow, “Stroyizdat”, 1984, 303 p. (In Russian)
5. I.D. Evzerov, Error estimates for inconsistent plate finite elements. Deposited at Ukrainian Res. Inst. for Sci.-Tech. Information, Reg. No. 1467, Kiev, 1979, 9 p. (In Russian)
6. O. Zenkevich, Finite element method in engineering. Moscow, “Mir”, 1975, 542 p. (In Russian)
7. A.B. Kaplun, Morozov E.M., Olferyeva M.A., ANSYS in the engineer’s hands. Application manual. Moscow, “URSS”, 2003, 272 p. (In Russian)
8. Karpilovsky V.S., Methods for constructing finite elements. Deposited at Ukrainian Res. Inst. for Sci.-Tech. Information, Reg. No. 3153, Kiev, 1980, 20 p. (In Russian)
9. N.M. Medvedeva, M.A. Mikitarenko, A.V. Perelmuter, Static and dynamic computer-aided analysis of masts. // Strength of materials and structural theory. Issue 48, Kiev, “Budivelnyk”, 1986, pp. 79–82. (In Russian)
10. A.V. Perelmuter, V.I. Slivker, Peculiarities of slope-deflection method’s algorithmic implementations when additional constraints are taken into account // Finite element method and structural mechanics: Proc. Leningrad Polytechnic Institute, No.349, 1976, pp. 28–36. (In Russian)
11. A.V. Perelmuter, V.I. Slivker, Numerical Structural Analysis: Methods, Models and Pitfalls, Springer, 2003, 501 p.
12. L.A. Rozin, Bar systems treated as systems of finite elements. Leningrad, University, 1975, 237 p. (In Russian)
13. A.F. Smirnov, Static and dynamic stability of structures. Moscow, “Transzheldorizdat”, 1947. (In Russian)
14. SNiP II-7-81*. Construction in seismic regions / Ministry of Construction of Russia. Moscow, 1996, 52 p.
15. G. Strang, J. Fix, Theory of the finite element method. Moscow, Mir, 1977, 349 p. (In Russian)
16. V.I. Feodosiev, Ten talks on strength of materials. Moscow, Nauka, 1975, 174 p. (In Russian)
17. S.A. Fialko, Direct methods for solving systems of linear equations in modern FEM-complexes – M.: SCAD SOFT, 2009. — 160 p.
18. S.A. Fialko, Realization of block Lanczos method with shifts in SCAD software applying to seismic analysis of structures // CADmaster, 2007, №40/5
19. A. Billionnet, On interval graphs and matrice profiles, Revue française d'automatique, d'informatique et de recherche opérationnelle. Recherche opérationnelle, 1986, 20, No. 3, 245–256.
20. S. Fialko, PARFES: A method for solving finite element linear equations on multi-core computers // Advances in Engineering Software, 2010, Vol. 41, Issue 12 — P.1256-1265.
21. R.N. Makiett, P.V. Marcal, Finite Element Analysis of Nonlinear Structures // Journal Structural Division. Proc./ASCE, 1968, Vol. 94, No ST9. P.2081–2105.
22. J.T. Oden, Finite elements of nonlinear continua. New York: McGraw Hill Book Company, 1972.
23. D. Sabahi, T. Rose, Special Applications of Global-Local Analysis, MSC/NASTRAN 1992 World Users` Conference Proceedings.
24. J.E. Schiermeier, J.M. Housner, J.B. Ranson, M.A. Aminpour, W.J. Stroud, The Application of Interface Element to Dissimilar Meshes in Global/Local Analysis, MSC 1996 World Users` Conference Proceedings
25. S.P. Timoshenko, Theory of elastic stability. New York: McGraw Hill Book Company, 1961
26. M. Yannanakis, Computing the minimum fill-in is NP-complete. SIAM Journal of Algebraic and Discrete Methods, 1981., T2. № 1, 77–79
27. C.H. Papadimitriou, The NP-completeness of the bandwidth minimization problem, Computing, 1976, 16, No. 3, 263–270.

3. Finite Element Library

3.1. General Issues

When the finite element method is used, a continuous area Ω occupied by a structure and possessing an infinite number of the degrees of freedom is approximated by a discrete model consisting of a set of subareas (finite elements) that have a finite number of the degrees of freedom and interact with each other only at the nodal points.

Each finite element (**FE**) Ω_e ($e = 1,\ldots, m$) is characterized by the following properties:

- a dimensionality of the used space (one-dimensional, two-dimensional, three-dimensional);
- a geometric shape, which is usually one of the simplest geometrical figures (line segment, triangle, rectangle, quadrangle, tetrahedron, etc.);
- a set of nodes placed (as a rule, though not always) on lines (surfaces) that divide the elements and are common for the adjacent ones;
- a set of the used degrees of freedom usually related to the nodes (though not necessarily to the nodes) — displacements, rotations, etc.;
- rules that define the relationship between the displacements of the nodes of the finite element and the nodes of the system. Nodes of the element, for example, can be attached to the nodes of the system rigidly or hingedly;
- a system of approximating functions which are defined within the area Ω_e and which enable to approximately express the components of the displacements in any point of the element in terms of its degrees of freedom;
- a physical law that defines a relationship between the internal forces and displacements;
- a class of problems to which this finite element type can be applied (plane stress state plate FE, Kirchhoff–Love plate FE, FE of the Reissner plate resting on the bi-parametric elastic subgrade, Timoshenko bar for a spatial problem, etc.);
- a set of allowed loads and actions which can be applied directly to the finite element and the method of their specification;
- and last but not least — a list of limitations and recommendations on the application.

One of the most important properties of a finite-element model is the maximum diameter of the elements

$$h = \max_e \left(\sup_{x,y\in\Omega_e} |x-y| \right),$$

depending on which the errors of the method are often estimated. In other words, h is a minimum diameter of a sphere which can be described around any finite element of the design model. Moreover, it is usually assumed that if a diameter is infinitely decreased, i. e. when $h \to 0$, the following regularity conditions are met — a sphere with a radius $\rho \geq Ch$ can be embedded into every finite element, where the constant C does not depend on h. This prevents the use of the so-called "needle" elements (too elongated rectangles, triangles with very small angles, etc.).

The approximating functions in the finite element method (FEM) are usually polynomial or piecewise-polynomial (subareas method), although there are elements with rational (the so-called isoparametric elements), trigonometric, logarithmic and other approximations of the displacement field. The selection of the degrees of freedom of the element and the corresponding approximating functions completely defines the rate of convergence and the FEM error estimate.

If we fix all parameters of a finite element design model, except for the size of the finite elements h, we can imagine that by changing this size, we will obtain a sequence of the approximate solutions of the problem $\boldsymbol{u}_h$. When talking about the convergence of FEM, we mean that this sequence tends to an exact solution of the problem $\boldsymbol{u}^*$, when $h \to 0$.

The above concept of the finite element modeling of a problem is based on what is known in the literature as the application of *h-elements*. The relative simplicity of this approach enables to solve the engineering problems directly and effectively. In recent years some software systems (for example, NASTRAN or StressCheck) have been using another finite element type, a so-called *p-element*. Unlike the

h-elements, the *p*-elements are capable of representing the curvature of surfaces and peculiarities of stress fields directly on simpler meshes containing a smaller number of elements. The accuracy of the analysis is controlled by a *p*-level (power of the polynomial) specified for each element: the higher the *p*-level, the better the accuracy. When the *h*-elements are used, the accuracy of the result can be improved by increasing the number of the elements; the computation time will grow accordingly. When the *p*-elements are used, the same phenomenon will take place when the "*p*-power" grows.

Despite the many positive characteristics of *p*-elements, the *h*-elements have a number of advantages. A global behavior of a structure can be analyzed better with the *h*-elements. They are also better in the representation of solutions of problems where a stress discontinuity may occur, such as ribbed plates, and in the solution of nonlinear problems. Finally, the *h*-elements are used in the time-proven computational technology elaborated in every detail. These are the reasons why the **SCAD** finite element library contains only the *h*-elements.

3.2. Composition of the Finite Element Library

When the software documentation refers to a finite element type, it usually means a number or another identification (such as name) of the respective section from the *finite element library* used in the software. Table 3.2-1 presents a classification of the FE types included in the finite element library, their respective possible indexes of the model type[1], identification of the calculated forces (stresses).

Table 3.2-1

FE type No.	Name	Allowable indices of the model type	Calculated stresses and forces	Purpose and capabilities
1-10	**Euler-Bernoulli bars**			Any kinds of bar systems taking into account: • rigid inserts; • elastic subgrade in one (types 1, 3, 7 and 103), two (types 2 and 102) or three directions of the principal axes of inertia (types 5, 6, 10, 105 and 110); • shear and prestressing (types 2, 5, 6, 10) [29]; • shear and prestressing according to Timoshenko hypotheses (types 102, 103, 105, 110). ***Note:*** elements of the types 6 and 7 are kept to maintain the continuity of versions.
1	plane truss bar	1, 2, 4, 5, 8, 9	N, $M(M_Y)$, $Q(Q_Z)$, r_X	
2	plane frame bar	2, 5, 8, 9	N, $M(M_Y)$, $Q(Q_Z)$, r_X, r_Z	
3,7	grade beam bar	3, 5, 8, 9	$M_K(M_X)$, M_Y, Q_Z, r_Z	
4	space truss bar	4, 5, 8, 9	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X, r_Y, r_Z	
5,6	space bar	5, 8, 9		
10	universal bar	1–5, 8, 9	depending on the model type	
101-110	**Timoshenko bars**			
102	plane frame bar	2, 5, 8, 9	N, $M(M_Y)$, $Q(Q_Z)$, r_X, r_Z	
103	grade beam bar	3, 5, 8, 9	$M_K(M_X)$, M_Y, Q_Z, r_Z	
105	space bar	5, 8, 9	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X, r_Y, r_Z	
110	universal bar	1–5, 8, 9	depending on the model type	
11-20	**Thin plates (Kirchhoff-Love theory)**	3, 5, 8, 9	M_X, M_Y, M_{XY}, Q_X, Q_Y, R_Z – when there is an elastic subgrade	Flexural plates lying in the XOY plane. Material: • isotropic; • orthotropic; • anisotropic material. There can be an elastic subgrade ***Note:*** elements of the types 13 and 14 are kept to maintain the continuity of versions.
111-120 511-520	Medium thickness plates **(Reissner-Mindlin theory)**			

[1] The main characteristics of the models depending on the value of their index are given in Chapter 28, Table 28.3-2

FE type No.	Name	Allowable indices of the model type	Calculated stresses and forces	Purpose and capabilities
21-30 121-130 521-530	**Plane problem of the theory of elasticity**	1, 2, 4, 5, 8, 9 2, 5, 8, 9	N_X, N_Z, N_{XZ} N_Y – only in the case of plane deformation; R_X, R_Z — when there is an elastic subgrade	Enable to analyze two types of structures: a) plane stress state; b) plane deformation. Material: • isotropic; • orthotropic; • anisotropic; • transverse isotropic (plane deformation) There can be an elastic subgrade.
31-40	**Solid finite elements**	4, 5, 8, 9	N_X, N_Y, N_Z, T_{XY}, T_{XZ}, T_{YZ}	For solving a spatial problem of the theory of elasticity, for the isotropic, transverse isotropic, orthotropic, and anisotropic material
41-50 91-99 591-599	**Thin shells (Kirchhoff-Love theory)**	5, 8, 9	N_X, N_Y, T_{XY}, M_X, M_Y, M_{XY}, Q_X and Q_Y, R_X, R_Y, R_Z – when there is an elastic subgrade	Analysis of thin and medium-thickness shells. Elements of zero curvature (flat). Material: • isotropic; • orthotropic; • anisotropic. There can be an elastic subgrade. The geometric features of the shell are taken into account by the geometry of the inscribed polyhedron
141-150 191-199 541-550	Medium-thickness shells (the Mindlin–Reissner theory)			
51-55	**Resilient constraints**	any	**reactions** in the directions of the specified constraints	To allow for: * the action of the elastic subgrade on the nodes of the structure including its attached (peripheral) part; * the elastic constraint between the two nodes of the structure
56-57	**Dampers**	any	**no**	Allowing for non-uniform damping at the direct integration of the equations of motion
61-70	**Axisymmetric**	11	N_X, N_Y, N_Z, T_{YZ}	For the analysis of the axisymmetric structures. All elements lie in the section of the body made with the ROZ plane
161-170 561-570		12		
71-80	**Multilayer shells,** accounting for the transverse shear, compression of layers and curvature	8	N_X, N_Y, N_Z, T_{XY}, T_{XZ}, T_{YZ} Calculated at the upper and lower surfaces of each layer	Strength analysis of thin, medium-thickness, and thick single-layer and multilayer isotropic, transverse isotropic, and orthotropic shells. Elastic properties of layers can be drastically different, in which case the hypothesis of straight normals becomes inapplicable because it may yield distorted results
81-90	**Multilayer shells,** accounting for the interlayer shear and curvature	9		
100	**Rigid body**	any		
151-160	**Null elements**	any		To impose constraints (specify displacements) in the directions different from those of the axes of the global coordinate system

FE type No.	Name	Allowable indices of the model type	Calculated stresses and forces	Purpose and capabilities
200	**Empty element**			Used in the early versions to save the numbering of the elements

A number of important convergence theorems are established in the theory of the finite elements method[1], for example, it is determined for the consistent elements [61], that if (k–1) is a degree of the polynomial with the help of which the displacements are approximated inside the finite elements, and an elliptic boundary value problem of order $2m$ is solved for which an approximate solution in displacements u_h is obtained, then the error in the energy, compared with the exact solution u, is

$$U(u-u_h, u-u_h) \le Ch^{2(k-m)} \|u\|^2_k,$$

where h is a minimum diameter of a sphere which can be described around any FE of the design model. This estimate involves a norm $\|u\|_k$ on the right, which is a root-mean-square value of k-th derivative of the sought-for function u.

Similar estimates for the inconsistent FE are obtained in the papers by I.D. Evzerov [12] and V.S. Karpilovsky [22, 23].

Mathematically proved conditions of convergence of the finite element method can be beneficially supplemented by some considerations of mechanical nature which enable to perform an informal analysis. The approximation of the field of displacements and forces by a certain finite set of predefined functions reduces the possibility of the arbitrary deformation, i.e. it can be interpreted as an imposition of certain constraints. If the elements are inconsistent, certain displacements which do not exist in the continual design model may occur along their borders (for example, mutual rotation angles of plates), which correspond to the absence of certain constraints.

As the number of the finite elements increases while their size decreases, the total number of the degrees of freedom of the structure increases, and, hence, the effect of the imposed constraints decreases. This process, provided that certain conditions are met, ensures the convergence of the method for the consistent finite elements. On the other hand, this process leads to the reduction of the mutual displacements at the boundaries between inconsistent elements, and this fact can be treated as the re-imposition of the previously removed constraints. Consequently, the convergence of inconsistent elements can take place only in cases when a positive tendency of overcoming imposed constraints prevails over a negative tendency of imposing constraints on the boundaries between elements.

Forces and stresses in the finite elements are calculated by default in the beginning, middle and end of the bar, and for other types of FEs – in the center of gravity. Forces can be calculated in the intermediate sections of bars, and in the nodes of other types of FEs. Forces and stresses in FEs are usually calculated in the local coordinate system. For example, the principal axes of inertia of the cross-sections of the flexible part are used for bars. A special coordinate system for calculating forces can be specified for all flat and solid FEs.

This chapter provides the composition of the finite element library for the linear analysis. The description of the results of the calculation and the sign convention for each type of the finite element are given in Chapter 13. The composition of the finite element library for the nonlinear analysis is given in Chapter 26.

The positive directions of the displacements along the X, Y and Z axes coincide with the positive directions of the respective axes, the positive directions of the rotation angles U_X, U_Y and U_Z are counter-clockwise when viewed from the end of the respective axis.

The following sign conventions are used in the system for different types of loads specified for the finite elements:

- a positive sign of the force means that it is directed opposite to the direction of the axes of the local or global coordinate systems;

[1] See also Sec. 2.7

- a positive sign of the moment means that its direction is clockwise when viewed from the end of the respective axis;
- a positive sign of the temperature difference means that the upper (when viewed from the end of the respective axis) fiber has a higher temperature than the lower one.

When the initial data are specified with the help of the graphical dialogs, there are usually no problems with the specification of the information for the certain types of finite elements. In addition to this chapter, see Chapter 29 that gives essential information regarding the data preparation in the text format.

The "basic" system of units is used in the chapter to describe the data: T, m and sizes of sections of bar elements are given in cm.

3.3. Bar Elements

Bar elements of different types are described in this section.

Local Coordinate Systems of the Bar

A universal bar finite element is shown in Fig. 3.3-1. There can be rigid inserts BD and CE on its ends. The B node is the beginning of the bar, and the C node is its end. The following designations of the end sections are used for the flexible part of the bar DE: number 1 designates the section at the beginning of the bar, and number 2 designates the one at its end.

In the general case it is assumed that a bar can be arbitrarily located with respect to the assumed global coordinate system XYZ. However, it is more convenient to obtain the characteristics of an individual bar finite element in the coordinate system related to the given element in a certain way. For this purpose, the local coordinate system $X_1Y_1Z_1$[1] is determined at the flexible part shown in Fig. 3.3-2, with respect to which the forces are calculated and some initial data are specified. In this system the Y_1 and Z_1 axes are the principal axes of inertia of the cross-section.

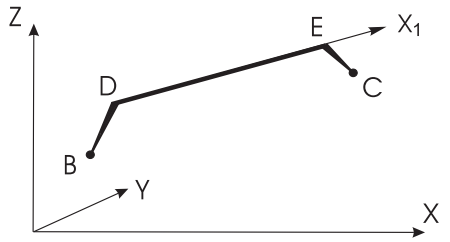


Figure 3.3-1

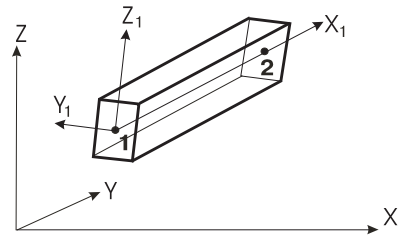


Figure 3.3-2

However, even before entering the information on the cross-sectional shape **SCAD** generates the axes ("preliminary project") of the local coordinate system $X_1^0Y_1^0Z_1^0$ (Fig. 3.3-3, *a*) using the following considerations:

- the X_1^0 axis is directed along the geometric axis of the flexible part of the section from the beginning of the bar to its end;
- the Z_1^0 axis is oriented towards the upper half-space (in the direction of the increase of the Z coordinate of the global coordinate system XYZ, in which the design model is given);
- the Y_1^0 axis complements the local coordinates in such a way so that the system forms a right-hand system $X_1^0Y_1^0Z_1^0$.

[1] Only the right-hand coordinate systems are used in **SCAD**. A coordinate system is considered to be right-hand if the Y axis is formed by rotating the X axis by an angle of 90° counter-clockwise when viewed from the end of the Z axis.

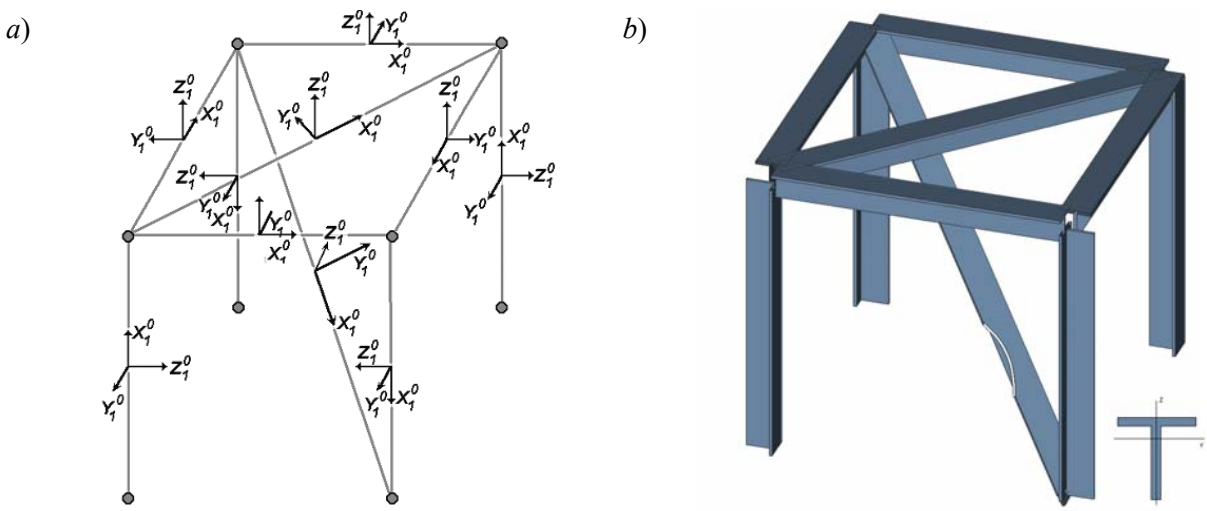


Figure 3.3-3

A particular cross-section is located in such a way so that its structural axes *y* and *z* coincide with the Y_1^0 and Z_1^0 axes. A system of the structural axes of the section *y*O*z* is given in the drawing of the cross-section in the assortment of rolled profiles, and in the dialog boxes of the **Section Builder**, **TONUS** and **Consul**. If the parametric sections are used, it is assumed that the structural axis *y* is horizontal, and the *z* axis is vertical.

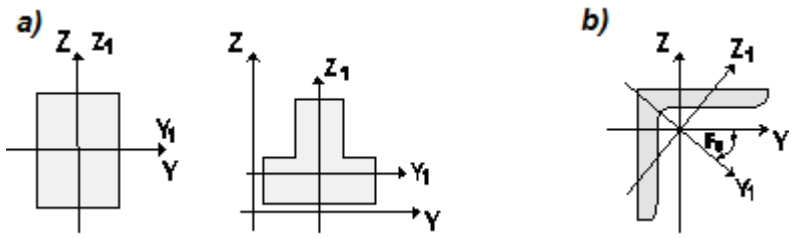


Figure 3.3-4

The systems of structural axes *y*O*z* and principal axes of inertia Y_1OZ_1 usually coincide, or are at least parallel to each other as shown in Fig. 3.3-4,*a*. In this case the pre-constructed system $X_1^0Y_1^0Z_1^0$ will be identical to the system $X_1Y_1Z_1$. However, there are cases when the axes (Y_1, Z_1) are inclined to the axes (*y*, *z*) at an angle F_U (angle between the principal axes of inertia and the structural axes of the section), as shown in Fig. 3.3-4,*b*. This angle is specified in the assortments of rolled profiles or is determined in **Section Builder**, **TONUS** and **Consul**, as an angle between the horizontal structural axis *y* and the principal axis of inertia U, with respect to which the section has the maximum rigidity. If the angle $F_U \neq 0$, then the final position of the principal axes of inertia can be determined by rotating the pre-determined axes Y_1^0 and Z_1^0 by this angle, the Y_1 axis will coincide with the U axis, and the V axis will coincide with the Z_1 axis. It should be noted that only the axes but not the actual cross-section change their position in space.

If the variant of orientation of the bar in space obtained by the above method does not satisfy the requirements to the orientation of the bar cross-section, the software enables to rotate the bar (its section) about the longitudinal axis by any angle required according to the conditions of the problem by explicitly specifying "the angle of orientation of the structural axes"[1].

[1] In the previous editions of the manual this angle was referred to as an "angle of pure rotation", then an "angle of orientation of the principal axes of inertia", and then it got its button in the **Assign** toolbar. The name "angle of orientation of the structural axes" reflects the essence of the problem more accurately because the transition to the principal axes of inertia which do not coincide with the structural ones is performed automatically.

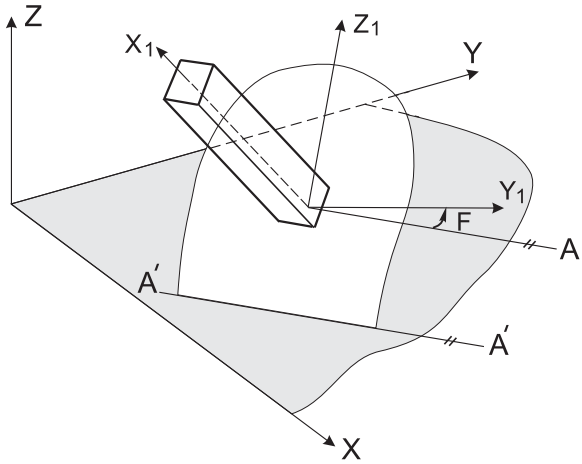


Figure 3.3-5

It is an angle between the Y_1 axis and a line related to the global coordinate system. The line is an intersection between a plane parallel to XOY and the plane of the bar cross-section Y_1OZ_1 passing through the beginning of the flexible part of the bar (Fig. 3.3-5). This line (the A axis) is oriented in such a way so that the X_1AZ_1 system is right-hand. It is assumed that the $X_1Y_1Z_1$ axes form a right-hand system, and the Z_1 axis is always is oriented towards the upper half-space (in the direction of the increase of the Z coordinate).

The positive value of the orientation angle F is determined by rotating the A axis counter-clockwise when viewed from the end of the X_1 axis until it coincides with the Y_1 axis.

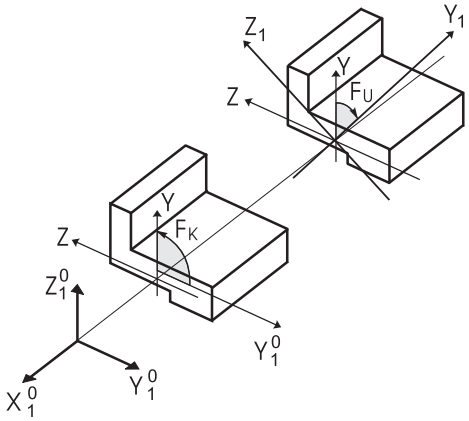


Figure 3.3-6

The final position of the principal axes of inertia of the cross-section is defined by the value of the sum of the "angle of orientation of the structural axes F_K" between the structural axis Y (Z) and axis $Y_1^0\left(Z_1^0\right)$ and the angle between the principal axes of inertia and the structural axes of the section F_U (Fig. 3.3-6). When specifying the sections from the assortment of rolled profiles or when using the **Section Builder**, **TONUS** and **Consul** for this purpose, the angle F_U is taken into account by the system automatically.

The positive value of the angles F_K (F_U) is defined by a counter-clockwise rotation of the section when viewed from the end of the X_1 axis. Thus, the angle F_K in Fig. 3.3-6 is positive, and the angle F_U is negative.

A more convenient way to specify the position of the local axes with respect to the global coordinates is to introduce an additional point (an orientation point) which must have a positive coordinate Y_1 or Z_1 (see below). For example, for all vertical columns of a cylindrical structure (Fig. 3.3-7) the same point of orientation of F_K with the coordinates of the center of the circle of the structure is specified (it is assumed that the global axes pass through this center).

To position a bar in space in a certain way, it is necessary to specify which angle is used: the one with the structural axes or the one with the principal axes of inertia. You can specify which axis Y_1 or Z_1 (similarly *y* or *z*) is directed towards the orientation point. The result will obviously change (Fig. 3.3-8).

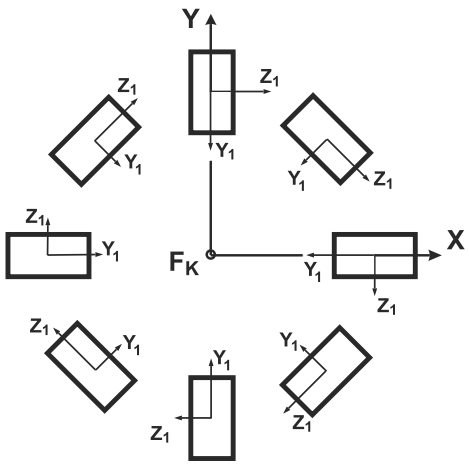


Figure 3.3-7

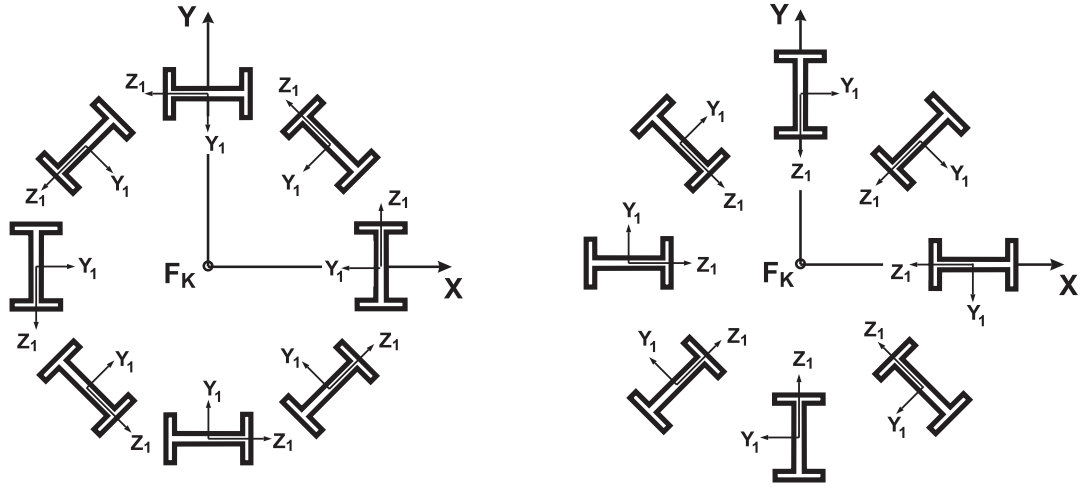


Figure 3.3-8

Keep in mind, however, that when you use sections with a nonzero angle F_U, the "flat frame" model can turn out to be incorrect. This is due to the fact that the principal axes of inertia of the bar do not lie in the selected force plane. Such errors are detected during the check of the initial data. This situation, it would seem, should not take place in the analysis of structures where bars work only in tension/compression (elements of type 1 or 4). However, since SCAD provides, for example, the local load in the directions of the principal axes of inertia for such cases as well, it can now act in a plane other than the plane of the frame. Therefore, the prohibition remains in this case as well.

General Description of Bar Finite Elements

As shown in Table 3.3-1, bar finite elements are implemented as special cases of universal bar elements of type 10 and 110.

If there is no shear in the bars of type 10, the differential equation of bending has a general solution in the form of a third-degree polynomial [16, 55]. When a bar compliant to shear is considered, and/or "compressed bar" with a non-zero longitudinal force, a Timoshenko bar differential equation modified to take into account the shear is applied; its general solution is represented by transcendental Kornoukhov functions [29].

In bars of type 110, normal displacements and rotation angles are approximated by third-degree polynomials, which are solutions of the Euler system of homogeneous differential equations of equilibrium if there is no prestressing and elastic subgrade [30,92]. If the shear stiffness properties are not specified for the Timoshenko bar, then the Bernoulli beam will work.

If the bar finite elements implement the exact solution, then to refine the solution, it is not necessary to divide the bar finite elements. Otherwise, the solution is approximate and can be refined by dividing the finite elements.

The longitudinal displacements and a rotation about the bar axis are approximated by first-degree polynomials.

The software enables to specify the rigid inserts at the ends for each type of bar elements, elastic subgrade in one (types 2, 3, 7, 102, 103) or two directions of the principal axes of inertia (types 5, 6, 10, 105, 110), shear and prestressing (types 2, 5, 6, 10, 102, 105, 110).

The end sections of the flexible part of a bar can be attached to the nodes of the structure or to the ends of the rigid inserts in various ways because any constrained degree of freedom of the bar related to the $X_1Y_1Z_1$ coordinates can be released for these sections. The release of a constrained linear degree of freedom provides slipping, and the removal of an angular constraint provides the free rotation (cylindrical hinge). The software makes sure that the combination of the removed constraints does not result in a dimensional instability (taking into account the current index of the model type).

Various local loads acting both in the $X_1Y_1Z_1$ coordinate system and in the global XYZ coordinate system can be applied to the bar.

The finite elements of type 10 and 110 can work with all the indices of the model type applied in the structural analysis. In other special cases it might be necessary to adjust the index of the model type to the type of the used bar element. The correspondence between the indices of the model type and the type of the bar element is given in Table 3.3-1.

Types of Bar Finite Elements

A list of different bar finite elements with the indication of the special cases of their application in different types of systems and the possible variant of their location in space is given in Table 3.3-1 and Fig. 3.3-9.

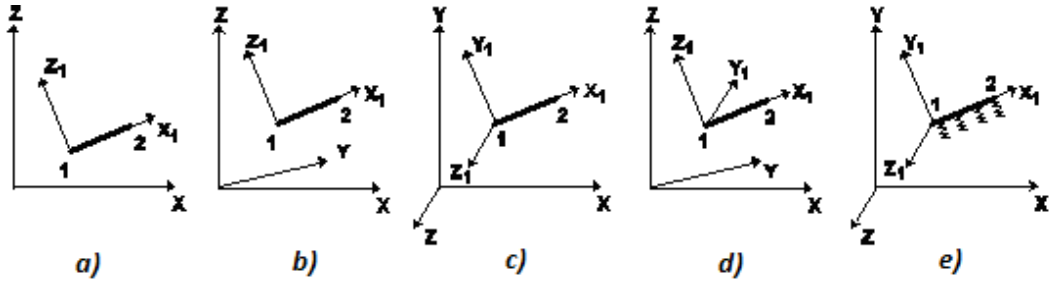


Figure 3.3-9

Table 3.3-1

FE No.	FE name	Index of the model type	Location	Degrees of freedom	Calculated forces
1	Plane truss bar (Fig. 3.3-9, *a*)	1, 2, 4, 5, 8, 9	XOZ plane	X, Z	N, M_Y, Q_Z, r_X
2 102	Plane frame bar (Fig. 3.3-9, *b*)	2, 5, 8, 9	XOZ plane	X, Z, U_Y	N, M_Y, Q_Z, r_X, r_Z
3 103	Grade beam bar (Fig. 3.3-9, *c*)	3, 5, 8, 9	XOY plane	Z, U_X, U_Y	M_K (M_X), M_Y, Q_Z, r_Z
4	Space truss bar (Fig. 3.3-9, *d*)	4, 5, 8, 9	any	X, Y, Z	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X
5 105	Space bar (Fig. 3.3-9, *d*)	5, 8, 9	any	X, Y, Z, U_X, U_Y, U_Z	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X, r_Y, r_Z
6	Space bar allowing for shear (Fig. 3.3-9, *d*)	5, 8, 9	any	X, Y, Z, U_X, U_Y, U_Z	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X, r_Y, r_Z
7	Grade beam bar on the elastic subgrade (Fig. 3.3-9, *e*)	3, 5, 8, 9	XOY plane	Z, U_X, U_Y	M_K (M_X), M_Y, Q_Z, r_Z
10 110	Universal bar	any	according to the index of the model type	according to the index of the model type or the element type	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X, r_Y, r_Z

Table 3.3-1 also provides the forces calculated in bars the description of which is given in Section 13.6.

Peculiarities of Specifying the Initial Information for Bar Finite Elements

The initial information related to the bar finite elements contains the data on their stiffness properties including the description of the section, boundary conditions and loads applied to the elements.

Description of the Section

EF — axial stiffness (T);
EI_Y — bending stiffness about axis Y_1 (Tm^2);
EI_Z — bending stiffness about axis Z_1 (Tm^2);
GI_{kr} — torsional stiffness (Tm^2);
GF_Y — shear stiffness about axis Y_1 (T);
GF_Z — shear stiffness about axis Z_1 (T);
$l_{Y,1}$, $l_{Y,2}$, $l_{Z,1}$ and $l_{Z,2}$ — section core sizes in (m).

Table 3.3-2

Element type	List of the specified stiffness properties
1	EF
2, 102	EF EI_Y [GF_Y]
3, 7, 103	EI_Y GI_{kr}
4	EF
5, 6, 105	EF EI_Y EI_Z GI_{kr} [GF_Y GF_Z]
10, 110	According to the index of the model type

Section core sizes (Fig. 3.3-10) are necessary for selecting the design combinations of forces.

When performing some types of the calculations the numerical properties have to be supplemented by the information on the elastic modulus of the material and the Poisson's ratio.

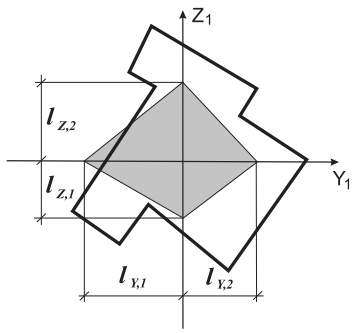


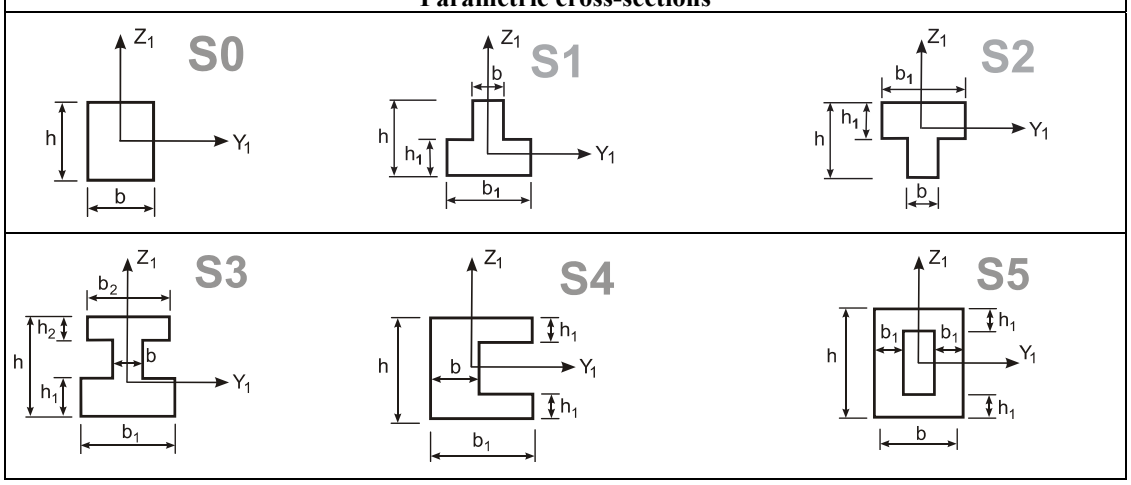
Figure 3.3-10

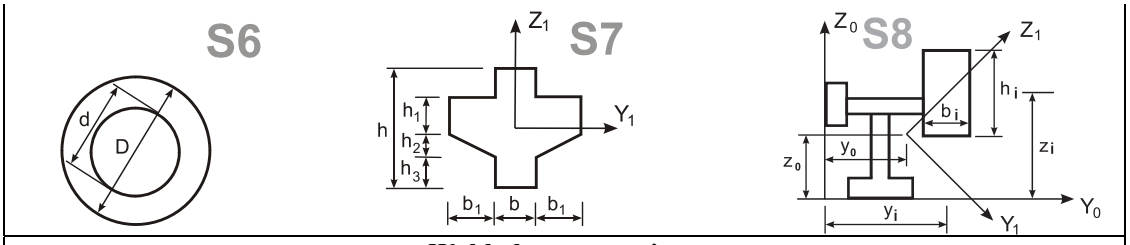
Functions of the specification of the stiffness properties of bar elements implemented in the software enable to describe them in the following ways:

- numerically as a number of stiffness properties;
- as parametric (standard) sections;
- as a numerical and parametric description when both parametric section and numerical description are specified;
- by selecting a section from the assortment of rolled profiles;
- by specifying a compound section from the assortment of rolled profiles;
- with the help of **Section Builder**, **Consul** or **TONUS**.

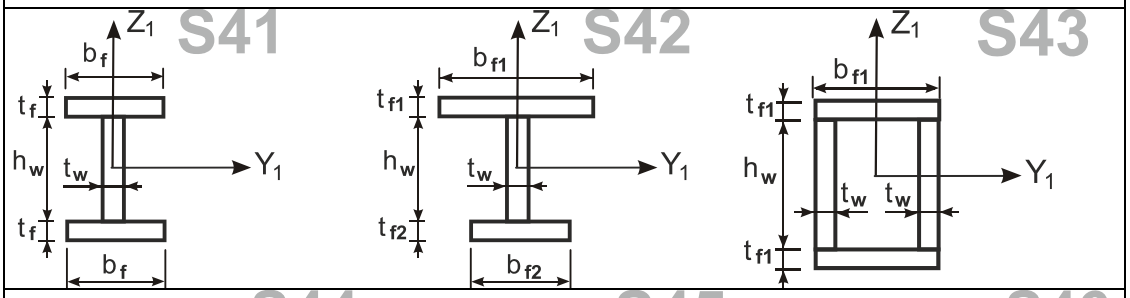
Table 3.3-3

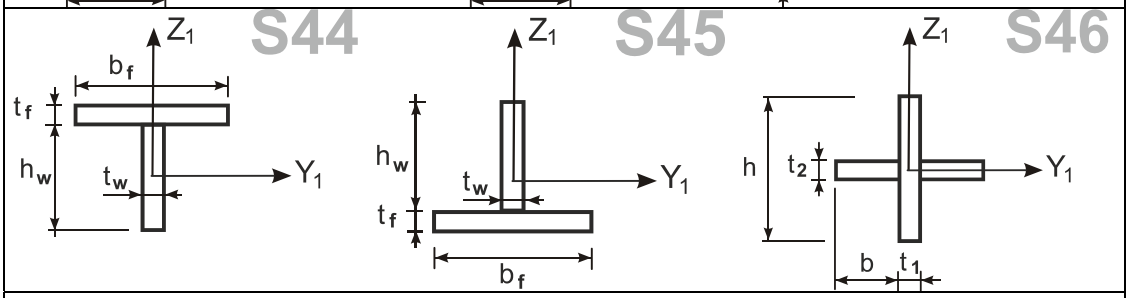
Parametric cross-sections



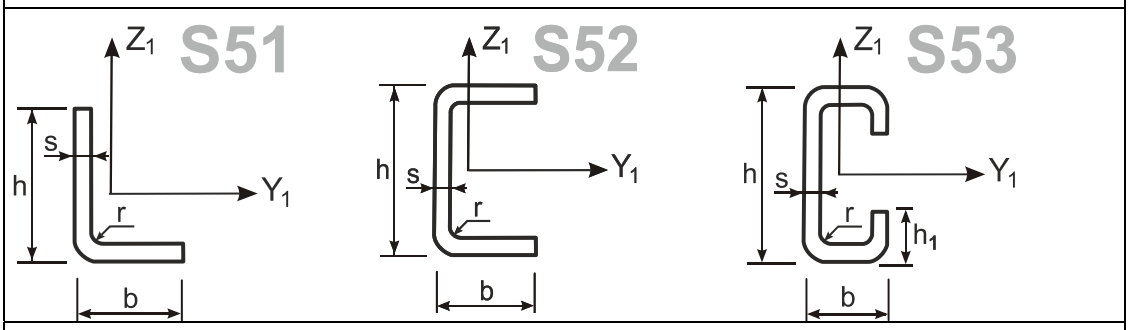


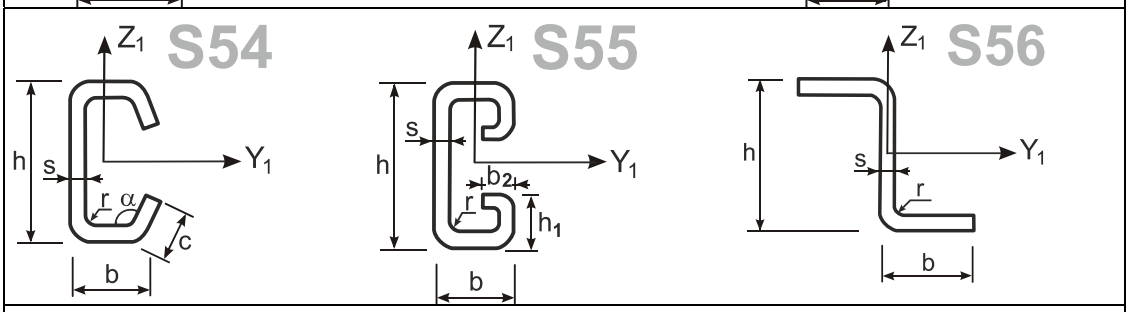
Welded cross-sections

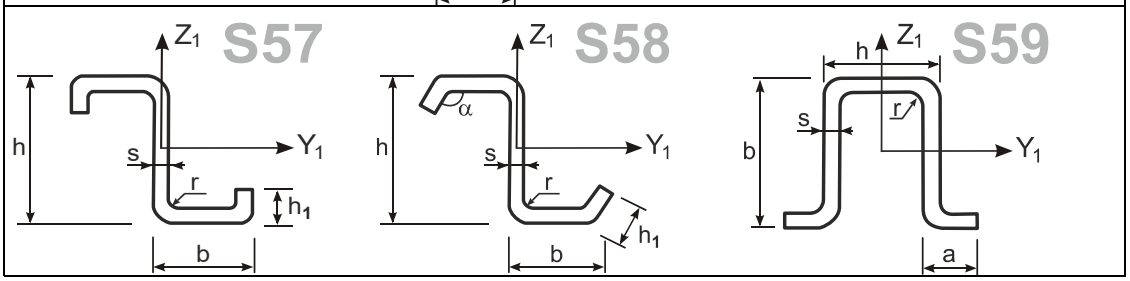


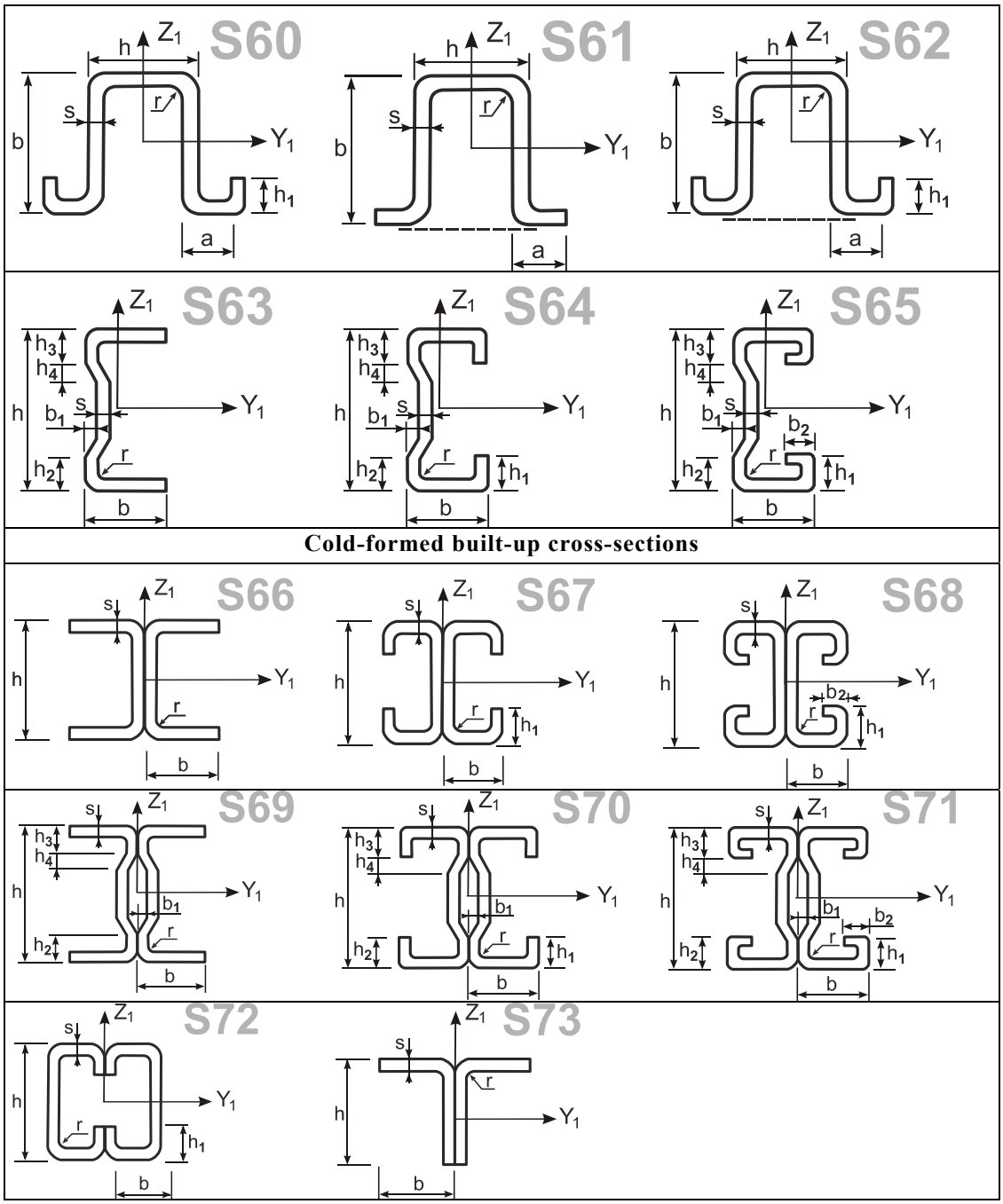


Cold-formed cross-sections









If a bar finite element is described **numerically**, in the general case the stiffness properties[1] of the flexible part of the bar listed in Table 3.3-2 are specified.

If a bar finite element is described **parametrically**, sections of nine types given in Table. 3.3-3 can be specified. Besides the sizes of the section it is necessary to specify the elastic modulus of the material E. If the basic units of measurement are used, the sizes of the section are given in cm, and the elastic modulus is given in T/m^2. The default value of the Poisson's ratio is 0,2. The identifiers **S0–S8** shown next to the drawings of the sections are used in the text format of the initial data archiving for marking the respective sections. In this case, it is possible to specify a variable section of the bar: the corresponding geometric dimensions change linearly from the beginning of the flexible part to its end.

[1] Hereinafter basic units of measurement are used. Obviously when other units of measurement are selected, the units of measurement of the used values change as well.

When the **numerical and parametric description** is used both the numerical stiffness properties and the parametric section are specified. In this case the numerical properties are taken into account when determining the stress-strain state of the structure, and type and sizes of the section are used when selecting the design combinations, reinforcement, etc.

When the stiffness properties of the elements are specified by selecting the sections from the **assortment of rolled profiles** all the properties are taken from the database, which is constantly updated and currently includes the following assortments:

Assortment name	File name in the root directory
GOST	russian.prf
Reduced GOST	Russhort.prf
Old Russian (1926, 1932 years)	Rus_old.prf
Assortment of the Chelyabinsk Metallurgical Plant (STO ASChM 20-93)	Aschm.prf
British Standard Sections	British1.prf
Overseas Shapes	British3.prf
Welded profiles	WELD.PRF
OTUA (France)	Otua.prf
DIN (Germany)	Din.prf
ARBED (Germany)	ARBED.PRF
ASTM (USA)	Usa.prf
Japanese Steel	Japan.prf
Indian	India.prf
Aluminium profiles	Alum.prf
Zinc coated steel wire ropes	Cable.prf
Rebars GOST	RebarRU.prf
Rebars of Belarus Republic	RebarBY.prf
Rebars ENV 10080:1996	RebarEC.prf
Poland	Poland.prf
Thermal roll-formed steel profiles	Term2007.prf
Cold-formed profiles	Tu1121.prf
Cold-formed zinc-coated steel profiles	Tu112101.prf
Assortment of China	China.prf

The default value of the elastic modulus of steel for all assortments is 2,1e7 T/m^2, specific weight — 7,85 T/m^3, Poisson's ratio — 0,3. More detailed information on the contents of the database of the assortments is given in the **Appendix**.

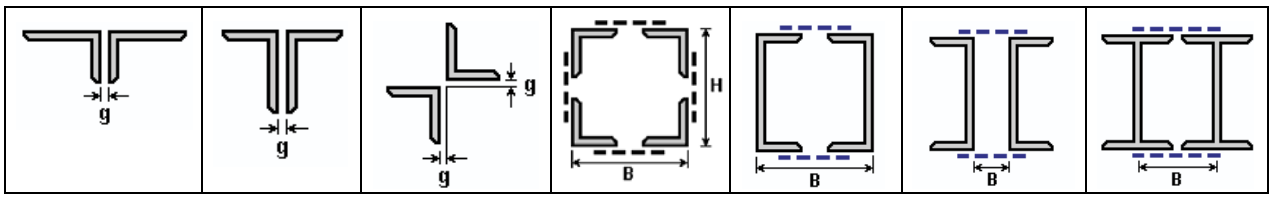


Figure 3.3-11. *Compound sections*

Table 3.3-4

Basic profile	Arrangement type	Section type	Code of the arrangement type	Parameters	
				1	2
Equal angle	connected on short legs		0, 1	g	
	cross-shaped		2	g	
	hollow section		3	H	B

Basic profile	Arrangement type	Section type	Code of the arrangement type	Parameters	
				1	2
Unequal angle	connected on short legs		0	g	
	connected on long legs		1	g	
Channel	hollow section		0	B	
	I-section		1	B	
I-beam			0	B	

When specifying the stiffness properties from the assortment of rolled profiles **compound sections** from angles, channels, and I-beams can be used. Types of sections (Fig. 3.3-11) and additional parameters specified together with them are given in Table 3.3-4.

Elastic Subgrade

The characteristics of the elastic subgrade can be specified in the direction of the local axes of the element X_1, Y_1 or Z_1. In the general case the bi-parametric elastic subgrade (Pasternak model) can be used, but if there is no data on the value of the subsoil parameter C_2, the software automatically switches to the Winkler model of the elastic subgrade with one subsoil parameter C_1.

The following values are specified for the directions Y_1 and Z_1:

h(b) — the width of a strip where the section interacts with the elastic subgrade in this direction (m);
C_1 — Winkler coefficient (T/m^3);
C_2 — Pasternak coefficient (T/m).

The following values are specified for the direction X_1:

a — the width of a strip where the section interacts with the elastic subgrade in this direction (m);
C_{1x} — Winkler coefficient (T/m ^{3}).

Shear and Prestressing

To take into account the shear compliance of an element:

- you just have to specify the values of shear stiffness for FE of types 6, 102, 103, 105 and 110;
- you have to specify an additional parameter in stiffness properties for elements of type 2, 5 and 10.

When the shear is taken into account for elements 2, 5, 6, 10, the transition from the classical design model of the Euler-Bernoulli bar to the Timoshenko model is carried out [38, 46].

You can specify the value of prestressing for the elements 2, 5, 6, 10,102, 105 and 110. You should keep in mind that prestressing applies only to the considered element and is not transferred to the system (as in the mounted prestressed precast reinforced concrete bar). Timoshenko model is used as the design model of the bar for elements 2, 5, 6, 10 when specifying the value of prestressing [29].

If it is necessary to specify the prestressing in the system, the user has to control the correctness of this parameter, in the sense that the values of the prestressing forces must correspond to the equilibrium state of the system as a whole (it can be achieved by specifying the corresponding constant thermal loading). In other words, in most cases these values must be consistent for all the elements of the prestressed part of the structure.

You should keep in mind that a statically determinate system cannot be prestressed.

Rigid Inserts

When a rigid insert is introduced into a bar element, the flexible part of the bar is moved from the nodes of the element. It keeps the old local coordinate system $X_1Y_1Z_1$, where the X_1 axis coincides with the axis of the flexible part of the element, and the Y_1 and Z_1 axes are directed along the principal axes of inertia of the cross-section. A new coordinate system $X_2Y_2Z_2$ appears (see Fig. 3.3-12), where the X_2 axis is directed from the start node of the element to the end one, the Y_2 axis is parallel to the Y_1Z_1 plane, and the Z_2 axis forms a right-hand system together with this system.

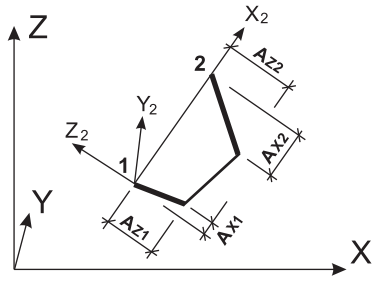


Figure 3.3-12

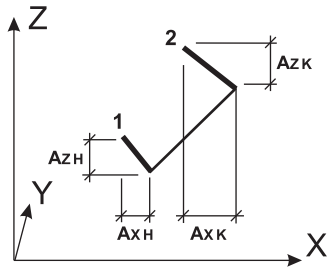


Figure 3.3-13

SCAD provides the following two variants of specifying the rigid inserts:

- the lengths of rigid inserts are specified in the coordinate system $X_2Y_2Z_2$, where the X_2 axis goes from the first node to the second one (Fig. 3.3-12). The rotation angle of the principal axes of inertia in the coordinate system $X_2Y_2Z_2$ is assumed equal to that specified for the flexible part of the bar;
- the lengths of the rigid inserts are specified in the global coordinate system as vectors directed from the node to the flexible part (Fig. 3.3-13).

When specifying the lengths of the rigid inserts the sign "minus" means that the size is measured in the direction opposite to that of the respective axis. The values are specified in meters in the basic system of units.

The position of the rigid inserts in space is specified as the values of the projections of the rigid inserts in the beginning and in the end of the bar on the axes of the selected coordinate system.

Loads on Bar Finite Elements

The following local loads can be applied to the bar finite elements:

- concentrated load specified in the local coordinate system of the element;
- concentrated load specified in the global coordinate system;
- uniformly distributed load specified in the local coordinate system of the element;
- uniformly distributed load specified in the global coordinate system;
- trapezoidal load specified in the local coordinate system of the element;
- trapezoidal load specified in the global coordinate system;
- thermal action given as the general heating/cooling with respect to the restraint temperature, or the temperature difference between the "upper" and "lower" faces in the cross-section of the element [34].

Forces and moments can act in the directions along the axes X_1, Y_1, Z_1 or X, Y, Z.

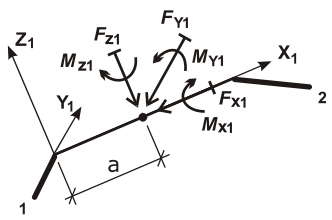
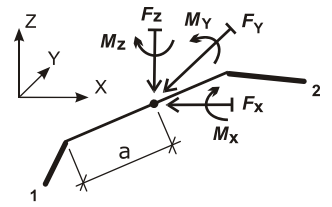
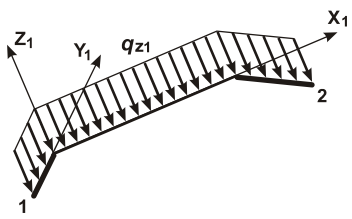
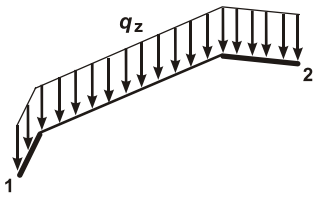
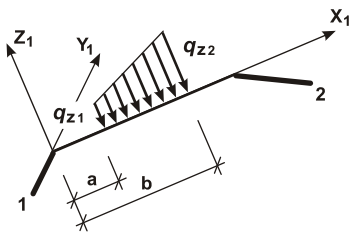
The concentrated loads are specified as values of forces or moments that act in a given direction and are applied at a point on the flexible part of the element, at the distance ***a*** from its beginning (the distance ***a*** can be also specified as a percentage of the flexible part of the element). Hence, the zero snap will turn the local load on the bar into the respective nodal load only if the bar has no rigid inserts.

The uniformly distributed load acts in the given direction, and it is specified as the value of the force or moment per unit length. This load can be applied to the rigid inserts too, if the respective parameter is not specified when the solver is invoked (see Sec.11).

The trapezoidal load is specified as two values of forces or moments per unit length determined at the beginning and the end of the loaded area and acting in the given direction. It is assumed that the beginning and the end of the loaded area are at the distance ***a*** and ***b*** , respectively, from the beginning of the flexible part of the bar (values ***a*** and ***b*** can be also specified as a percentage of the flexible part of the element). The load applied to the rigid inserts can be taken into account. ***a*** (***b***) is specified as the value of the projection on the X_1 axis of the point of the beginning (end) of the loaded area on the rigid insert.

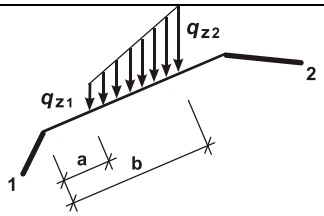
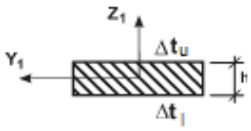
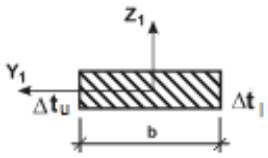
A full list of loads on the bar finite elements is given below in Table 3.3-5. When the basic system of units is used, the loads are specified in the following units of measurement: forces in T, moments in Tm, temperature in the degrees Celsius, snaps in m.

Table 3.3-5

Load arrangement	Direction	Specified values	FE type
Concentrated load in the local coordinate system[1]			
	along the X_1 axis along the Y_1 axis along the Z_1 axis about the X_1 axis about the Y_1 axis about the Z_1 axis	F_{X1} a F_{Y1} a F_{Z1} a M_{X1} a M_{Y1} a M_{Z1} a	1, 2, 4, 5, 6, 10, 102, 105, 110 4, 5, 6, 10, 105, 110 1–7, 10, 102, 103, 105, 110 3, 5, 6, 7, 10, 103, 105, 110 2, 3, 5, 6, 7, 10, 102, 103, 105, 110 5, 6, 10, 105, 110
Concentrated load in the global coordinate system			
	along the X axis along the Y axis along the Z axis about the X axis about the Y axis about the Z axis	F_X a F_Y a F_Z a M_X a M_Y a M_Z a	1, 2, 4, 5, 6, 10, 102, 105, 110 4, 5, 6, 10, 105, 110 1–7, 10, 102, 103, 105, 110 3, 5, 6, 7, 10, 103, 105, 110 2, 3, 5, 6, 7, 10, 102, 103, 105, 110 5, 6, 10, 105, 110
Uniformly distributed load in the local coordinate system			
	along the X_1 axis along the Y_1 axis along the Z_1 axis about the X_1 axis about the Y_1 axis about the Z_1 axis	q_{X1} q_{Y1} q_{Z1} m_{X1} m_{Y1} m_{Z1}	1, 2, 4, 5, 6, 10, 102, 105, 110 4, 5, 6, 10, 105, 110 1–7, 10, 102, 103, 105, 110 3, 5, 6, 7, 10, 103, 105, 110 2, 3, 5, 6, 7, 10, 102, 103, 105, 110 5, 6, 10, 105, 110
Uniformly distributed load in the global coordinate system			
	along the X axis along the Y axis along the Z axis about the X axis about the Y axis about the Z axis	q_X q_Y q_Z m_X m_Y m_Z	1, 2, 4, 5, 6, 10, 102, 105, 110 4, 5, 6, 10, 105, 110 1–7, 10, 102, 103, 105, 110 3, 5, 6, 7, 10, 103, 105, 110 2, 3, 5, 6, 7, 10, 102, 103, 105, 110 5, 6, 10, 105, 110
Self-weight	along the Z axis	k – self-weight factor[2]	any
Trapezoidal load in the local coordinate system			
	along the X_1 axis along the Y_1 axis along the Z_1 axis about the X_1 axis about the Y_1 axis about the Z_1 axis	q_{X1} a q_{X2} b q_{Y1} a q_{Y2} b q_{Z1} a q_{Z2} b m_{X1} a m_{X1} b m_{Y1} a m_{Y1} b m_{Z1} a m_{Z1} b	1, 2, 4, 5, 6, 10, 102, 103, 105, 110 4, 5, 6, 10, 105, 110 1–7, 10, 102, 103, 105, 110 3, 5, 6, 7, 10, 103, 105, 110 2, 3, 5, 6, 7, 10, 102, 103, 105, 110 5, 6, 10, 105, 110

[1] Snap of the concentrated and trapezoidal loads on the bar finite elements is specified from the beginning of the flexible part or in fractions of the length of the flexible part.

[2] In order to take the self-weight into account it is necessary to specify the characteristics of the section and the specific weight of the material.

Load arrangement	Direction	Specified values	FE type
Trapezoidal load in the global coordinate system			
	along the X axis along the Y axis along the Z axis about the X axis about the Y axis about the Z axis	q_{X1} a q_{X2} b q_{Y1} a q_{Y2} b q_{Z1} a q_{Z2} b m_{X1} a m_{X2} b m_{Y1} a m_{Y2} b m_{Z1} a m_{Z2} b	1, 2, 4, 5, 6, 10, 102, 105, 110 4, 5, 6, 10, 105, 110 1-7, 10, 102, 103, 105, 110 3, 5, 6, 7, 10, 103, 105, 110 2, 3, 5, 6, 7, 10, 102, 103, 105, 110 5, 6, 10, 105, 110
Thermal load			
 Variation of the average temperature of the bar	tension (compression) along the X_1 axis	Δt, [α] Δt — increase (decrease) of the temperature with respect to the normal conditions of operation (restraint temperature)	1, 2, 4, 5, 6, 10, 102, 105, 110
 *Difference of the temperature across the section of the bar, the X_1OZ_1 plane*	bending of the bar about the Y_1 axis	Δt_u, Δt_l, h, [α] Δt_u — temperature of the upper face of the section (in the direction of the Z_1 axis); Δt_l — temperature of the lower face of the section (opposite to the direction of the Z_1 axis); h — section height.	2, 3, 5, 6, 7, 10, 102, 103, 105, 110
 *Difference of the temperature across the section of the bar, the Y_1OZ_1 plane*	bending of the bar about the Z_1 axis	Δt_u, Δt_l, b, [α] Δt_u — temperature of the left face of the section (in the direction of the Y_1 axis); Δt_l — temperature of the right face of the section (opposite to the direction of the Y_1 axis); b — section width.	5, 6, 10, 105, 110
Note: Thermal expansion coefficients specified in the stiffness properties of the respective element are used in the calculation. When converting the files of the previous versions the thermal expansion coefficient specified when describing the load is used. The respective warning is generated. Thermal expansion coefficients can be specified for a certain load only when preparing the data in the text format.			

3.4. Flat Finite Elements for the Calculation of Deep Beams, Thin Plates, and Shallow Shells

Class of the Solved Problems

Flat finite elements described in this section are used for solving the following types of problems of the theory of elasticity [1, 2, 8, 10, 32, 52, 66]:

- analysis of the plane stress state;

- analysis of the plane deformation;
- bending of thin plates;
- bending of plates according to the Reissner-Mindlin theory;
- analysis of shallow shells.

When the shell structures are analyzed, a versatile method for representing their geometry is to use a polyhedron located close to the midsurface; the smaller the faces of the polyhedron, the more accurate the representation of this surface. If this method is used (it is implemented in **SCAD**), it is advisable to use the elements of zero curvature (flat finite elements) with an independent approximation of the normal and tangential displacements. The generation of such a finite element is a simple combination of finite elements for the plane stress state and for the bending of the plate. Mesh refinement increases the accuracy of the approximation of the shell surface by an inscribed polyhedron, and the convergence of FEM has a theoretical proof [6].

Material of Structures

It is assumed that the material of the elements is homogeneous throughout the thickness and different variants of the elastic symmetry of the material can be taken into account. Let's describe them for the XOY plane using the following designations:

ε_x, ε_y, γ_{xy} — relative linear and angular strains;

σ_x, σ_y, σ_z, τ_{xy} — normal and shear stresses;

E_i — Young's modulus for the i-th principal direction of elasticity (if they are equal, the designation E is used for all i);

G_{ij} — shear modulus which characterizes the change of the angle between the i-th and j-th principal directions (if they are equal, the designation G is used for all pairs i and j);

ν, ν_{ij} — Poisson's ratios which characterize a lateral reduction at compression or expansion at tension in the direction of the coordinate axes. The first index indicates the direction of the deformation, and the second one — the direction of the force;

$\eta_{ij,k}$ — first-order interaction coefficients which characterize shear in the planes parallel to the coordinate ones under the action of normal stresses;

$\eta_{k,ij}$ — second-order interaction coefficients which characterize the elongations in the directions parallel to the coordinate axes caused by the shear stresses.

The following variants of the elastic properties of the material are implemented in **SCAD**:

a) ***isotropic body*** for the plane stress state and bending of the plate:

$$\varepsilon_x = \frac{1}{E}(\sigma_x - \nu\sigma_y), \quad \varepsilon_y = \frac{1}{E}(\sigma_y - \nu\sigma_x), \quad \varepsilon_{xy} = \frac{1}{G}\tau_{xy};$$

b) ***isotropic or transverse isotropic body*** for the plane deformation:

$$\varepsilon_x = (\sigma_x - \bar{\nu}\sigma_y)/\bar{E}, \ \varepsilon_y = (\sigma_y - \bar{\nu}\sigma_x)/\bar{E}, \ \varepsilon_{xy} = \tau_{xy}/\bar{G}, \ \sigma_z = \nu_{xz}\sigma_x + \nu_{yz}\sigma_y,$$

where for the isotropic body:

$$\bar{E} = \frac{E}{1-\nu^2}, \quad \bar{\nu} = \frac{\nu}{1-\nu}, \quad \bar{G} = \frac{E}{2(1+\nu)} \quad \left(\nu = \nu_{zy} = \nu_{yx}\right),$$

and for the transverse isotropic body:

$$\bar{E} = \frac{E}{1-\nu_{xz}\nu_{zx}}, \quad \bar{\nu} = \frac{\nu + \nu_{xz}\nu_{zx}}{1-\nu_{xz}\nu_{zx}}, \quad \bar{G} = \frac{E}{2(1+\nu)} \quad \left(\nu = \nu_{zy} = \nu_{yx}\right);$$

c) ***orthotropic body*** for the plane stress state and bending of the plate:

$$\varepsilon_x = \frac{1}{E_x}\sigma_x - \frac{\nu_{xy}}{E_y}\sigma_y, \quad \varepsilon_y = -\frac{\nu_{xy}}{E_x}\sigma_x + \frac{1}{E_y}\sigma_y, \quad \varepsilon_{xy} = \frac{1}{G_{xy}}\tau_{xy};$$

d) ***orthotropic body*** for the plane deformation:

$$\varepsilon_x = \frac{(\sigma_x - \bar{\nu}_{xy}\sigma_y)}{\bar{E}_x}, \quad \varepsilon_y = \frac{(\sigma_y - \bar{\nu}_{xy}\sigma_x)}{\bar{E}_y}, \quad \varepsilon_{xy} = \frac{\tau_{xy}}{\bar{G}_{xy}}, \quad \sigma_z = \nu_{xz}\sigma_x + \nu_{yz}\sigma_y,$$

where

$$\bar{E}_x = \frac{E_x}{1-\nu_{xz}\nu_{zx}}, \quad \bar{E}_y = \frac{E_y}{1-\nu_{yz}\nu_{zy}}, \quad \bar{\nu}_{xy} = \frac{\nu_{xy}+\nu_{xz}\nu_{zy}}{1-\nu_{xz}\nu_{zy}}, \quad \bar{\nu}_{yz} = \frac{\nu_{yx}+\nu_{yz}\nu_{zx}}{1-\nu_{yz}\nu_{zx}}, \quad \bar{G}_{xy} = G_{xy};$$

e) ***anisotropic body*** for the plane stress state and bending of the plate:

$$\varepsilon_x = \frac{1}{E_x}(\sigma_x - \nu_{yx}\sigma_y + \eta_{xy,x}\tau_{xy}), \ \varepsilon_y = \frac{1}{E_y}(-\nu_{xy}\sigma_x + \sigma_y + \eta_{xy,y}\tau_{xy}), \ \varepsilon_{xy} = \frac{1}{G_{xy}}(\eta_{x,xy}\sigma_x + \eta_{y,xy}\sigma_y + \tau_{xy});$$

f) ***anisotropic body*** for the plane deformation:

$$\varepsilon_x = \frac{1}{\bar{E}_z}(\sigma_x - \bar{\nu}_{yz}\sigma_y + \bar{\eta}_{zy,x}\tau_{xy}), \ \varepsilon_y = \frac{1}{\bar{E}_y}(-\bar{\nu}_{xy}\sigma_x + \sigma_y + \bar{\eta}_{xy,y}\tau_{xy}),$$

$$\varepsilon_{xy} = \frac{1}{\bar{G}_{xy}}(\bar{\eta}_{x,xy}\sigma_x + \bar{\eta}_{y,xy}\sigma_y + \tau_{xy}), \ \sigma_z = \nu_{xz}\sigma_x + \nu_{yz}\sigma_y - \eta_{xy,z}\tau_{xy},$$

where

$$\bar{E}_x = \frac{E_x}{1-\nu_{xz}\nu_{zx}}, \quad \bar{E}_y = \frac{E_y}{1-\nu_{yz}\nu_{zy}}, \quad \bar{\nu}_{xy} = \frac{\nu_{xy}+\nu_{xz}\nu_{zy}}{1-\nu_{xz}\nu_{zy}}, \quad \bar{\nu}_{yz} = \frac{\nu_{yx}+\nu_{yz}\nu_{zx}}{1-\nu_{yz}\nu_{zx}},$$

$$\bar{\eta}_{xy,x} = \frac{\eta_{xy,x}+\nu_{zx}\eta_{xy,x}}{1-\nu_{xz}\nu_{zx}}, \quad \bar{\eta}_{xy,y} = \frac{\eta_{xy,y}+\nu_{zy}\eta_{xy,z}}{1-\nu_{yz}\nu_{zy}}, \quad \bar{\eta}_{x,xy} = \frac{\eta_{x,xy}+\nu_{xz}\eta_{z,xy}}{1-\eta_{xy,z}\eta_{z,xy}},$$

$$\bar{\eta}_{y,xy} = \frac{\eta_{y,xy}+\nu_{yz}\eta_{z,xy}}{1-\eta_{xy,x}\eta_{z,xy}}, \ \bar{G}_{xy} = \frac{G_{xy}}{1-\eta_{xy,z}\eta_{z,xy}}.$$

It is considered that symmetry conditions hold for any anisotropic material:

$$E_x\nu_{xy} = E_y\nu_{yx}, \quad E_x\eta_{x,y} = G_{xy}\eta_{xy,x}, \quad E_y\eta_{y,xy} = G_{xy}\eta_{xy,y}.$$

Reissner-Mindlin Plates

In the Kirchhoff-Love thin plate theory the deformations are expressed in terms of one unknown, the deflection w(x, y).

In the Reissner–Mindlin theory [5], where the hypothesis on maintaining the normal to the deformed surface does not hold, the following three functions are the unknowns: deflection w(x, y) and two rotation angles of the normal to the surface β_x(x, y) and β_y(x, y). In the expression of the potential energy (2.2.2) we have:

$$\mathbf{u} = \left[w, \beta_x, \beta_y\right]^T, \quad \boldsymbol{\varepsilon} = \left[\gamma_{xz}, \gamma_{yz}, \chi_{xx}, \chi_{yy}, \chi_{xy}\right]^T, \ \boldsymbol{\sigma} = \left[Q_x, Q_y, M_x, M_y, M_{xy}\right]^T.$$

Strains $\boldsymbol{\varepsilon}$ are expressed in terms of the displacements $\mathbf{u}$:

$$\gamma_{xz} = \frac{\partial w}{\partial x} + \beta_x, \quad \gamma_{yz} = \frac{\partial w}{\partial x} - \beta_y,$$

$$\chi_{xx} = -\frac{\partial \beta_x}{\partial x}, \quad \chi_{yy} = \frac{\partial \beta_y}{\partial y}, \quad \chi_{xy} = -\frac{\partial \beta_x}{\partial y} + \frac{\partial \beta_y}{\partial x}.$$

Stresses $\boldsymbol{\sigma}$ are related to the strains $\boldsymbol{\varepsilon}$ in the following way for an isotropic body:

$$Q_x = G\,k\,h\,\gamma_{xz}, \ Q_y = G\,k\,h\,\gamma_{yz}, \ k = \frac{5}{6},$$

$$M_x = D\chi_{xx}, \ M_y = D\chi_{yy}, \ M_{xy} = \frac{1}{2}(1-\nu)D\chi_{xy},$$

$$D = \frac{Eh^3}{12(1-\nu^2)}$$ — bending stiffness of the plate.

Allowing for the Elastic Subgrade

The elastic subgrade can be taken into account in all plane plate and shell finite elements:

- along the Z1 axis for plates and shells. In the general case a bi-parametric model of the elastic subgrade in contact with the structure (Leontiev-Vlasov, Pasternak model) is implemented [8]. The following values are specified: :

  C_1 — a coefficient that characterizes the resistance of the elastic subgrade to compression;

  C_{2X}, C_{2Y} — coefficients that characterize the resistance of the elastic subgrade to shear when the shear area is perpendicular to the X_1 and Y_1 axes respectively of the local coordinate system of the element. For the isotropic elastic subgrade $C_{2X} = C_{2Y}$;

  C_{2XY} — a coefficient that characterizes the resistance of the elastic subgrade to torsion in the X_1OY_1 plane of the local coordinate system of the element. For the isotropic and orthotropic elastic subgrade $C_{2XY} = 0$.
- for grade beams and shells in the plane of the element. The following value is specified

  C_{1uv} – a coefficient that characterizes the resistance of the elastic subgrade in the X_1Y_1 plane.

If there is no data on the value of the subsoil parameter C_2 (its value is equal to 0), the software automatically switches to the Winkler model of the elastic subgrade with one subsoil parameter C_1.

The estimates of the coefficient C_1 can be obtained with the help of **Cross** included in the **SCAD Office** system.

Local Coordinate Systems

Three different local coordinate systems can be used with every flat finite element:

- local coordinate system of the element;
- coordinate system of the orthotropy axes;
- coordinate system for calculating forces and stresses.

The two latter coordinate systems coincide with the local coordinate system of the element by default. Local coordinate systems of all elements are defined by the order of numbering of their nodes. They have certain peculiarities due to the fact that a plate should be located in the XOY plane or parallel to it, while elements used in the plane problem of the theory of elasticity are usually located in the XOZ plane (parallel to this plane) and only shell elements can take any position in space.

All plate and shell elements have their local coordinate system X_1OY_1, in which the X_1 axis goes from the first node to the second one. The Y_1 axis lies in the plane of the element, it is orthogonal to X_1 and directed towards the third node, the Z_1 axis forms a right-hand system together with the X_1 and Y_1 axes. For the plate elements the Z_1 axis always points upwards in the direction of the increase of the Z coordinate.

All elements used in the plane problem of the theory of elasticity (except for the triangular element of type 22) have a local coordinate system X_1OZ_1, in which the X_1 axis goes from the first node to the second one, and the Z_1 axis lies in the plane of the element and is directed towards the third node. The local coordinate system X_1OZ_1 of the element of type 22 coincides with XOZ.

A special coordinate system for the output of forces can be specified for a group of elements. This system is defined by the direction of the vector, the projection of which on the plane of the element is taken as the X_1 axis, the direction of the Z_1 axis remains unchanged for the plate and shell elements, and the Y_1 axis complements the system to a right-hand one. For the elements of a plane problem, these are the axes Y1 and Z1 respectively.

Types of Flat Finite Elements

SCAD includes various flat finite elements with a triangular and quadrangular shape. The number of nodes in some types of elements can be greater than the number of vertices. In this case additional nodes lie on **one** or **several** sides of the element and their numbers follow after the numbers of vertices in any order.

Elements for the Analysis of Thin Flexural Plates

These FEs are always located in the XOY plane and there are three degrees of freedom in each of their nodes: w — vertical displacement (deflection), and U_X, U_Y — rotation angles about the X and Y axes. The FEs can be used in the models with indices 3, 5, 8 and 9 and have an isotropic, orthotropic, or

anisotropic material. Moments M_X, M_Y, M_{XY} and shear forces Q_X and Q_Y are calculated. When the subsoil parameter C_1 is specified, R_z is calculated (see Sec. 13.6).

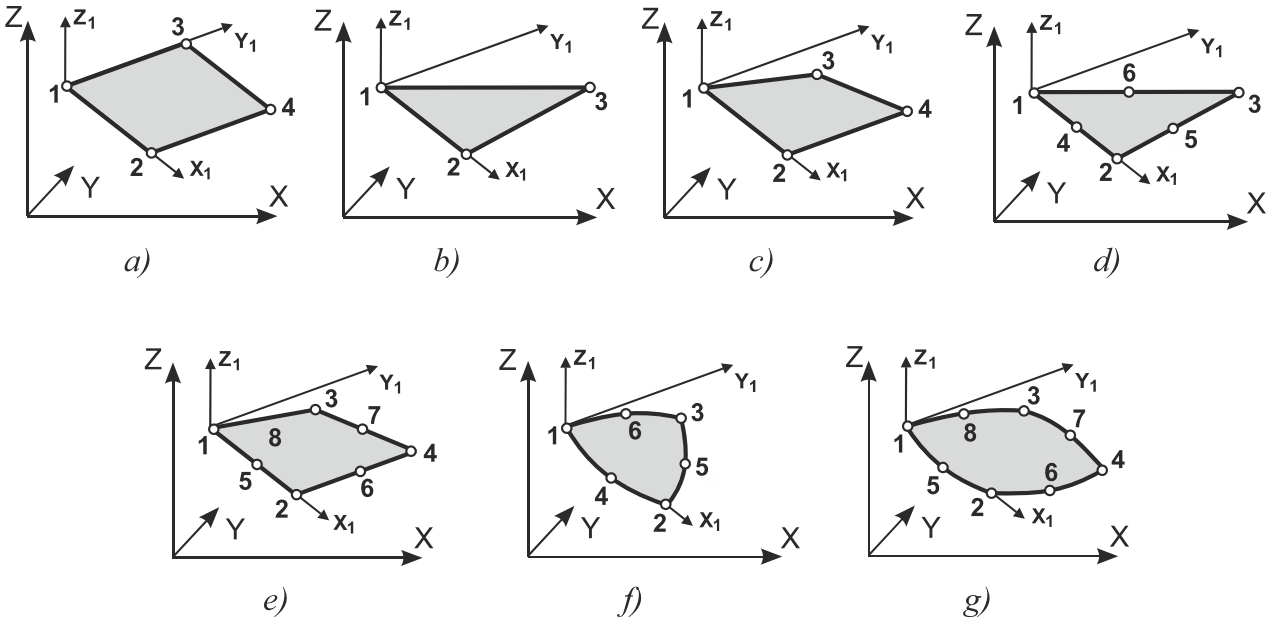


Figure 3.4-1. *Plate finite elements*

The list of FEs for the analysis of thin flexural plates is given in Table 3.4-1.

Table 3.4-1. Types of elements for the analysis of thin plates (Kirchhoff-Love theory)

Type	Name	Number of nodes	Node numbering order and local axes	Comments
11	Rectangular	4	Fig. 3.4-1, *a*	semi-consistent [18,22,23]
12, 14	Triangular	3	Fig. 3.4-1, *b*	inconsistent [18,22,23]
13	Rectangular	4	Fig. 3.4-1, *a*	inconsistent [28]
15	Triangular	3-6	Fig. 3.4-1, *d*	*SubAreas*[1], *option* 1, [24,23]
16	Quadrangular	4-8	Fig. 3.4-1, *e*	*SubAreas*, *option* 2 [25,23]
18	Triangular	3-6	Fig. 3.4-1, *d*	*SubAreas*, *option* 2 [24,23]
19	Quadrangular	4	Fig. 3.4-1, *c*	*SubAreas*, [25,23]
20	Quadrangular	4-8	Fig. 3.4-1, *e*	*SubAreas*, [24,22], *option* 1

Bending of Plates according to the Reissner-Mindlin Theory

These elements are used for the analysis of medium thickness plates and implement the Reissner-Mindlin theory. They are completely similar to the elements used for the analysis of thin plates in terms of the specification of the initial data. They differ from the elements given in Table 3.4-1 only in their type number, which is greater by 100. For example, element 120 is a quadrangular element with the number of nodes from 4 to 8, similarly to the element 20.

Each node of the elements has three degrees of freedom: w — vertical displacement (deflection), the positive direction of which coincides with the direction of the Z axis, and the rotation angles U_X and U_Y about the X and Y axes.

[1] *SubAreas* – subdomain method: triangular and quadrangular elements are divided, respectively, by medians and diagonals into triangles. Approximation polynomials of the appropriate degree are used on each of these triangles in such a way that compatibility is ensured. We obtain piecewise polynomial approximations.

Table 3.4-2. Types of elements for the analysis of medium-thickness plates (Reissner-Mindlin theory)

Type	Name	Number of nodes	Node numbering order and local axes	Comments
111	Rectangular	4	Fig. 3.4-1, *a*	*JIDR4*[1] [91,23]
112	Triangular	3	Fig. 3.4-1, *b*	*JIDR3* [91,23]
115	Triangular	3-6	Fig. 3.4-1, *d*	*JIDR3-6* [91,23]
116	Quadrangular	4-8	Fig. 3.4-1, *g*	isoparametric, *JIDR* [91,23]
118	Triangular	3-6	Fig. 3.4-1, *f*	isoparametric, *JIDR* [91,23]
119	Quadrangular	4	Fig. 3.4-2, *b*	*JIDR* [91,23]
120	Quadrangular	4-8	Fig. 3.4-1, *e*	*JIDR, SubAreas* [91,23]
512	Triangular	3	Fig. 3.4-1, *b*	modified, *DSG3*[2],[23]
517	Quadrangular	4	Fig. 3.4-1, *c*	isoparametric, *MITC4*[3] [2,75]
518	Triangular	3	Fig. 3.4-1, *b*	*DSG3* [76]

The list of FEs for the analysis of plates according to the Reissner-Mindlin theory is given in Table 3.4-2.

Elements for Solving the Plane Stress and Plane Strain Problems of the Theory of Elasticity

All elements considered in this section enable to analyze both **plane-stress** and **plane-strain** systems (depending on the index, which is specified when describing the stiffness properties of the elements).

There are the following groups of element types:

- lying in the XOZ plane and having two degrees of freedom in each node — U and W, which are the tangential displacements. FEs can be used in models with indices 3, 5, 8 and 9;
- can take any position in space and have three degrees of freedom in their nodes — U, V and W (types 23, 24, 26, 27, 28). The degree of freedom V is introduced to provide the connection between the solid elements of the structure. It is equal to zero in the local coordinate system of the element. FEs can be used in models with indices 4, 5, 8 and 9;
- lying in the XOZ plane and having three degrees of freedom in each node — U and W, and U_y – rotation angle about the normal to the axis[4]. FEs can be used in models with indices 2, 5, 8 and 9.

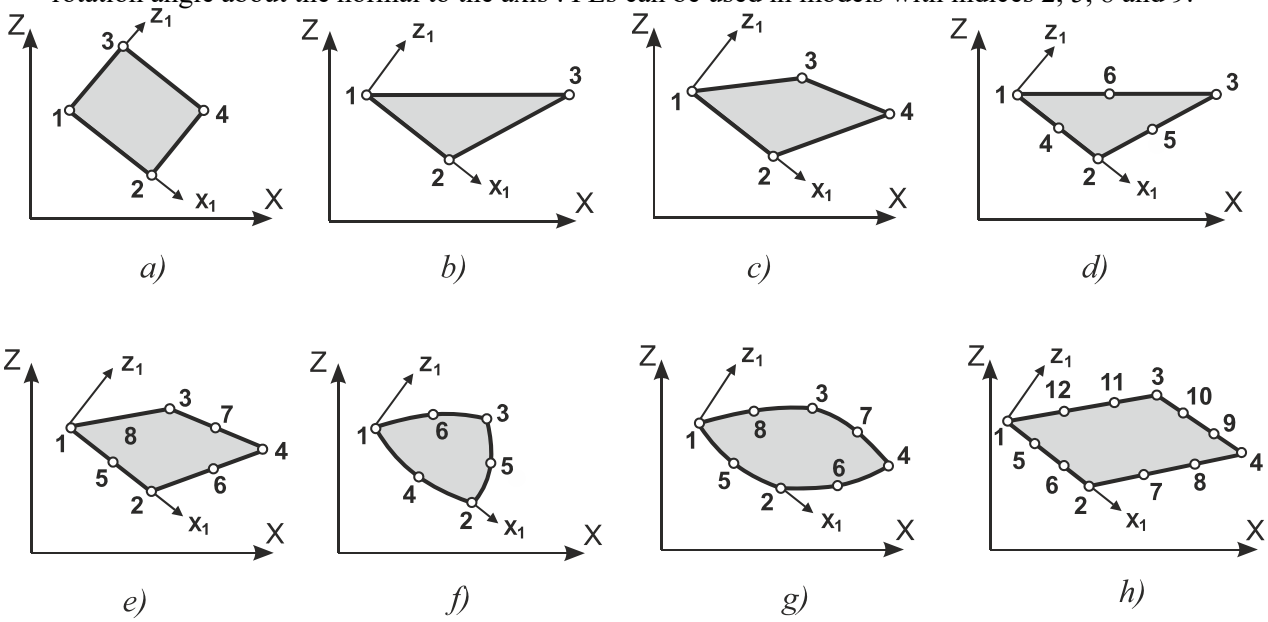


Figure 3.4-2. *Plane finite elements*

[1] *JIDR, JOINT INTERPOLATION OF DISPLACEMENTS AND ROTATIONS* [91,23].
[2] *DSG, Discrete Shear Gap* [76].
[3] *MITC, Mixed Interpolation of Tensorial Components* [3,75].
[4] This can be either an average rotation angle or a quasi-rotational degree of freedom [90].

Table 3.4-3. Types of elements for the plane problem of the theory of elasticity

Degrees of freedom of nodes	Type	Name	Number of nodes	Node numbering order and local axes	Comments
X, Z	21	Rectangular	4	Fig. 3.4-2, *a*	polylinear shape functions
	22	Triangular	3	Fig. 3.4-2, *b*	linear shape functions
	25	Triangular	3-6	Fig. 3.4-2, *d*	*SubAreas* [16, 23]
	29	Quadrangular	4-12	Fig. 3.4-2, *h*	*SubAreas* [26, 23]
	30	Quadrangular	4-8	Fig. 3.4-2, *e*	*SubAreas* [26, 23]
X, Y, Z	23	Rectangular	4	Fig. 3.4-2, *a*	polylinear shape functions
	24	Triangular	3	Fig. 3.4-2, *b*	linear shape functions
	26	Quadrangular	4-8	Fig. 3.4-2, *g*	isoparametric
	27	Quadrangular	4-8	Fig. 3.4-2, *e*	*SubAreas* [26, 23]
	28	Triangular	3-6	Fig. 3.4-2, *f*	isoparametric
Elements with drilling (*DDF*) and quasi-rotational degrees of freedom (QRDF)					
X, Y, U_Y	121	Rectangular	4	Fig. 3.4-2, *a*	*DDF*[1], inconsistent [90, 23]
	122	Triangular	3	Fig. 3.4-2, *b*	*DDF*, inconsistent [90, 23]
	125	Triangular	3-6	Fig. 3.4-2, *d*	*DDF, SubAreas* [90, 23]
	129	Quadrangular	4-8	Fig. 3.4-2, *e*	*DDF, SubAreas*, inconsistent [90, 23]
	130	Quadrangular	4-8	Fig. 3.4-2, *e*	*DDF, SubAreas* [90, 23]
	526	Quadrangular	4	Fig. 3.4-2, *c*	*QRDF*[2], isoparametric [90, 23]
	527	Quadrangular	4	Fig. 3.4-2, *c*	*QRDF, SubAreas* [90, 23]
	528	Triangular	3	Fig. 3.4-2, *b*	*QRDF* [90, 23]

All elements can have isotropic, orthotropic or anisotropic material, as well as transverse isotropic for plane deformation.

The list of elements and their main properties are given in Table 3.4-3.

The following stresses are calculated — N_X, N_Z, N_{XZ}, as well as N_Y — at the analysis of structures in the plane strain state. When the subsoil parameter C_{uv} is specified, R_x and R_z are calculated (see Sec. 13.6).

[1] *DDF – Drilling degrees of freedom* [23]
[2] *QRDF – Quasi-rotational degrees of freedom* [23]

Finite Elements for the Analysis of Thin Shallow Shells

Finite elements used for the analysis of thin shallow shells can take any position in space.

Six degrees of freedom are determined in the nodes of the elements — U, V, W, U_X, U_Y and U_Z (three linear displacements along and three rotation angles about the coordinate axes). The degrees of freedom U, V correspond to the membrane deformations, and W, U_X, U_Y correspond to the bending ones.

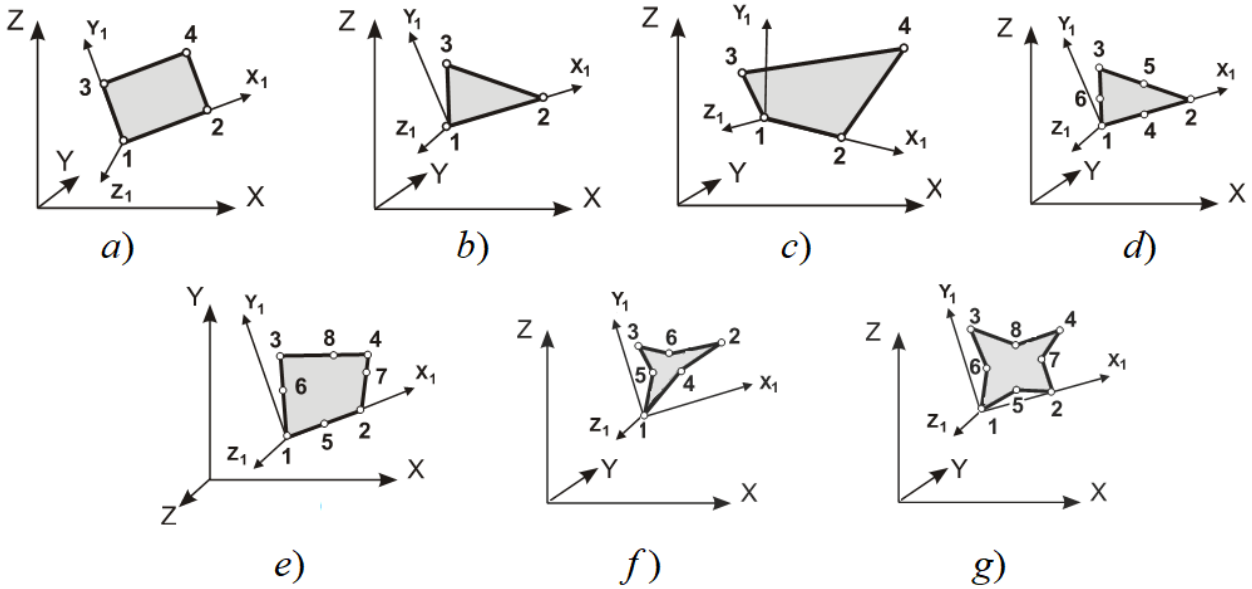


Figure 3.4-3. *Shell elements*

There are the following groups of element types:

- a group where the rotation angle U_Z in the local coordinate system of the element is always equal to zero. It is introduced to provide the connection between the elements not lying in one plane, and it is required to take into account the spatial behavior of the structure;
- a group where the angle U_Z is the rotation angle about the normal to the axis, which can be either an average rotation angle or a quasi-rotational degree of freedom [90, 23]

Material — isotropic, orthotropic, and anisotropic.

The list of elements is given in Table 3.4-4.

Table 3.4-4. Elements for the analysis of thin shells (Kirchhoff-Love theory)

Type	Name	Number of nodes	Node numbering order and local axes	Degrees of freedom	Comments
41	Rectangular	4	Fig. 3.4-3, *a*	U, V	polylinear shape functions
				W, U_X, U_Y	semi-consistent [18, 22, 23]
42	Triangular	3	Fig. 3.4-3, *b*	U, V	linear shape functions
				W, U_X, U_Y	inconsistent [18, 22, 23]
43	Rectangular	4	Fig. 3.4-3, *a*	U, V	polylinear shape functions
				W, U_X, U_Y	inconsistent [28, 23]
44	Quadrangular	4	Fig. 3.4-3, *c*	U, V	*SubAreas* [26, 23]
				W, U_X, U_Y	*SubAreas* [25, 23]
45	Triangular	3-6	Fig. 3.4-3, *d*	U, V	*SubAreas* [16, 23]
				W, U_X, U_Y	*SubAreas* [24, 23], option 1
46	Quadrangular	4-8	Fig. 3.4-3, *c*	U, V	*SubAreas* [26, 23]
				W, U_X, U_Y	*SubAreas*, option 2 [25, 23]
48	Triangular	3-6	Fig. 3.4-3, *d*	U, V	*SubAreas* [16, 23]
				W, U_X, U_Y	*SubAreas*, option 2 [24, 23]
50	Quadrangular	4-8	Fig. 3.4-3, *e*	U, V	*SubAreas* [26, 23]
				W, U_X, U_Y	*SubAreas*, option 1 [25, 23]

Type	Name	Number of nodes	Node numbering order and local axes	Degrees of freedom	Comments
Elements with drilling (*DDF*) and quasi-rotational degrees of freedom (QRDF)					
91	Rectangular	4	Fig. 3.4-3, *a*	U, V, UZ	*DDF*, inconsistent [90, 23]
				W, U_X, U_Y	semi-consistent [18, 22, 23]
92	Triangular	3	Fig. 3.4-3, *b*	U, V, UZ	*DDF*, inconsistent [90, 23]
				W, U_X, U_Y	inconsistent [18, 22, 23]
93	Rectangular	4	Fig. 3.4-3, *a*	U, V, UZ	*DDF*, inconsistent [90, 23]
				W, U_X, U_Y	inconsistent [28, 23]
94	Quadrangular	4	Fig. 3.4-3, *e*	U, V, UZ	*DDF*, *SubAreas*, inconsistent [90, 23]
				W, U_X, U_Y	*SubAreas*, [25, 23]
95	Triangular	3-6	Fig. 3.4-3, *d*	U, V, UZ	*DDF*, *SubAreas* [90, 23]
				W, U_X, U_Y	*SubAreas* [24, 23]
96	Quadrangular	4-8	Fig. 3.4-3, *e*	U, V, UZ	*DDF*, *SubAreas* [90, 22]
				W, U_X, U_Y	*SubAreas* [25, 23]
97	Quadrangular	4-8	Fig. 3.4-3, *e*	U, V, UZ	*DDF*, *SubAreas,* inconsistent [90, 23]
				W, U_X, U_Y	*SubAreas* [25, 23]
591	Rectangular	4	Fig. 3.4-3, *a*	U, V, UZ	*QRDF4* [90, 23]
				W, U_X, U_Y	semi-consistent [18, 22, 23]
592	Triangular	3	Fig. 3.4-3, *b*	U, V, UZ	*QRDF3* [90, 23]
				W, U_X, U_Y	inconsistent [18,22, 23]
593	Rectangular	4	Fig. 3.4-3, *c*	U, V, UZ	*QRDF4*, [90, 23]
				W, U_X, U_Y	inconsistent [28, 23]
594	Quadrangular	4	Fig. 3.4-3, *c*	U, V, UZ	*QRDF4*, *SubAreas* [23]
				W, U_X, U_Y	*SubAreas* [25, 23]

Stresses N_X, N_Z, N_{XZ}, moments M_X, M_Y, M_{XY} and shear forces Q_X and Q_Y are calculated. When the subsoil parameter C_1 is specified, R_z is calculated, and when C_{uv} is specified, R_x and R_y are calculated (see Sec. 13.6).

Finite Elements for the Analysis of Shells according to the Reissner-Mindlin Theory

These elements are used for the analysis of shallow shells allowing for shear and implement the Reissner-Mindlin theory. They are completely similar to the elements used for the analysis of thin shells in terms of the specification of the initial data.

The list of elements is given in Table 3.4-5.

Table 3.4-5. Elements for the analysis of medium-thickness shells (Reissner-Mindlin theory)

Type	Name	Number of nodes	Node numbering order and local axes	Degrees of freedom	Comments
141	Rectangular	4	Fig. 3.4-3, *a*	U, V	polylinear shapes
				W, U_X, U_Y	*JIDR*, inconsistent [91, 23]
142	Triangular	3	Fig. 3.4-3, *б*	U, V	linear shape functions
				W, U_X, U_Y	*JIDR3* [91, 23]
143	Quadrangular isoparametric	4	Fig. 3.4-3, *в*	U, V	polylinear shape functions
				W, U_X, U_Y	*MITC4,* isoparametric [3, 75, 23]
144	Quadrangular	4	Fig. 3.4-3, *в*	U, V	*SubAreas* [26, 23]
				W, U_X, U_Y	*JIDR4, SubAreas* [91, 23]
145	Triangular	3-6	Fig. 3.4-3, *г*	U, V	*SubAreas* [26, 23]
				W, U_X, U_Y	*JIDR, SubAreas* [91, 23]
146	Quadrangular isoparametric	4-8	Fig. 3.4-3, *д*	U, V	isoparametric
				W, U_X, U_Y	*JIDR,* isoparametric [91, 23]

147	Triangular, isoparametric	3-6	Fig. 3.4-3, *е*	U, V	isoparametric
				W, U_X, U_Y	*JIDR*, isoparametric [91, 23]
148	Triangular	3	Fig. 3.4-3, *б*	U, V	linear shape functions
				W, U_X, U_Y	*DSG3M* [23]
149	Triangular	3	Fig. 3.4-3, *б*	U, V	linear shape functions
				W, U_X, U_Y	*DSG3* [76, 23]
150	Quadrangular	4-8	Fig. 3.4-3, *д*	U, V	*JIDR, SubAreas* [26, 23]
				W, U_X, U_Y	*JIDR, SubAreas* [91, 23]
Elements with drilling (*DDF*) and quasi-rotational degrees of freedom (QRDF)					
191	Rectangular	4	Fig. 3.4-3, *а*	U, V, UZ	*DDF,* inconsistent [90, 23]
				W, U_X, U_Y	*JIDR* [91, 23]
192	Triangular	3	Fig. 3.4-3, *б*	U, V, UZ	*DDF, SubAreas*, [90, 23]
				W, U_X, U_Y	*JIDR, SubAreas* [91, 23]
194	Quadrangular	4	Fig. 3.4-3, *в*	U, V, UZ	*DDF, SubAreas* [90, 23]
				W, U_X, U_Y	*JIDR, SubAreas* [91, 23]
195	Triangular	3-6	Fig. 3.4-3, *г*	U, V, UZ	*DDF, SubAreas* [90, 23]
				W, U_X, U_Y	*JIDR, SubAreas* [91, 23]
196	Quadrangular	4-8	Fig. 3.4-3, *д*	U, V, UZ	*DDF, SubAreas* [90, 23]
				W, U_X, U_Y	*JIDR, SubAreas*, [91, 23]
197	Quadrangular	4-8	Fig. 3.4-3, *д*	U, V, UZ	*DDF,* inconsistent, *SubAreas* [90, 23]
				W, U_X, U_Y	*JIDR, SubAreas*, [91, 23]
542	Triangular	3	Fig. 3.4-3, *б*	U, V, UZ	*QRDF* [90, 23]
				W, U_X, U_Y	*JIDR* [91, 23]
543	Quadrangular	4	Fig. 3.4-3, *в*	U, V, UZ	*QRDF,* isoparametric [90, 23]
				W, U_X, U_Y	*MITC4,* isoparametric [3, 75, 23]
544	Quadrangular	4	Fig. 3.4-3, *а*	U, V, UZ	*QRDF, SubAreas* [90, 23]
				W, U_X, U_Y	*JIDR, SubAreas* [91, 23]
546	Quadrangular	4	Fig. 3.4-3, *в*	U, V, UZ	*QRDF,* isoparametric [90, 23]
				W, U_X, U_Y	*JIDR,* ,isoparametric [91, 23]
547	Triangular	4	Fig. 3.4-3, *б*	U, V, UZ	*QRDF* [90, 23]
				W, U_X, U_Y	*DSG3M* [23]
548	Triangular	3	Fig. 3.4-3, *б*	U, V, UZ	*QRDF* [90, 23]
				W, U_X, U_Y	*DSG3* [76, 23]

Peculiarities of Specifying the Initial Information for Flat Finite Elements

The following stiffness properties are specified depending on the class of the solved problems:

- $E(T/m^2)$, ν —for the isotropic material;
- $E_1(T/m^2)$, $E_2(T/m^2)$, ν_{12}, ν_{21}, $G_{12}(T/m^2)$ — for the orthotropic material;
- $E_1(T/m^2)$, $E_2(T/m^2)$, ν_{12}, ν_{21}, $G_{12}(T/m^2)$, $\eta_{12,1}$, $\eta_{12,2}$, $\eta_{1,12}$, $\eta_{2,12}$ — for the anisotropic material.

The thickness of the element δ is always indicated.

The following properties can be taken into account for plane strain:

- $E_3(T/m^2)$, ν_{13}, ν_{31} — transverse isotropy;
- $E_3(T/m^2)$, ν_{13}, ν_{31}, ν_{23}, ν_{32} — orthotropy;
- $E_3(T/m^2)$, ν_{13}, ν_{31}, ν_{23}, ν_{32}, $\eta_{12,3}$, $\eta_{3,12}$ — anisotropy.

When the orthotropic and anisotropic plates are analyzed, it is necessary to control the directions for which the rigidities are specified. The orthotropy axis X_A can be specified in the global coordinate system by a vector or coordinates of a point. The given vector is projected on the plane of the element to define the X_A orthotropy axis. The Y_A axis lies in the plane of the element, it is directed towards the third node and is

orthogonal to X_A, and the system X_A, Y_A, Z_A is right-hand. You can change the direction of the axes by specifying the rotation angle of the local axes.

Winkler coefficient C_1(T/m^3) (compression) and Pasternak coefficient C_2(T/m) (shear) are specified in order to take into account the elastic subgrade. The orthotropy and anisotropy can be taken into account when the elastic subgrade resists shear.

When the results are used, for example, in the reinforcement selection postprocessor, it is necessary to specify the direction of the axes for the stress calculation, when the position of the reinforcement is assumed to be known. The direction is specified in the global coordinate system by the increments with respect to the first node of the element or by the coordinates of the point lying on this axis. The given direction is projected on the plane of the element to define the X_N axis for the calculation of stresses. The Y_N axis lies in the plane of the element, the Z_N axis coincides with the Z_1 axis, and the system X_N,Y_N, Z_N is right-hand. You can change the direction of the axes by specifying the rotation angle of the local axes.

For all flat finite elements the stresses (forces) are calculated both in the center of the elements and in their nodes. The control of their output is described in the Chapters 7 and 13.

Loads on Flat Finite Elements

The following local loads can be applied to the flat finite elements:

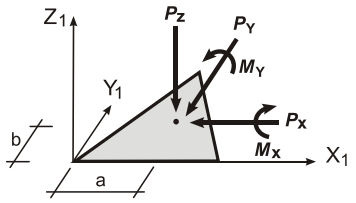
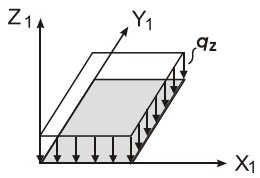
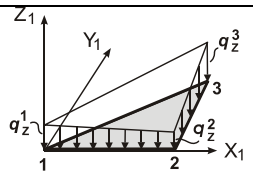
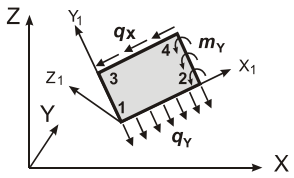
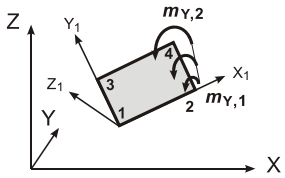
- concentrated loads which are specified as a value of a force or moment acting in the given direction and applied in the point on the surface of the element. The loads and the coordinates of the point of its application can be specified both in the local coordinate system of the element and in the global coordinate system of the problem;
- uniformly distributed loads which act in the given direction and are specified by the value of the force or moment per unit area. The load acts over the entire surface of the element and can be specified both in the local and in the global coordinate systems;
- self-weight which acts along the Z axis of the global coordinate system. The specific weight of the material of plates must be specified. The self-weight factor is specified for the element;
- trapezoidal loads which act in the given direction and are specified as the force or moment values determined in all vertices of the element. The load acts over the entire surface of the element where it is approximated by a linear or bilinear law. The direction of its action can be specified both in the local and in the global coordinate systems. The load values are specified only in the vertices of the elements and the order of their specification must correspond to the order of numbering of the nodes of the element described in this section;
- uniformly distributed loads along the line which act in the given direction on the line connecting two nodes of the element. The line is defined by the node numbers in accordance with the description of the element and the value of the force or moment per unit length. The load can be specified both in the local and in the global coordinate systems;
- trapezoidal loads along the line which act in the given direction on the line connecting two nodes of the element. The line is defined by the node numbers in accordance with the description of the element and the value of the force or moment per unit length. The load can be specified both in the local and in the global coordinate systems;
- thermal action in the form of the general heating/cooling with respect to the restraint temperature and/or temperature difference between the “upper” and “lower” surfaces of the plate [58].

The list of the main local loads and the rules for specifying them are given in the tables below. You should keep in mind that when specifying a load in the global coordinate system their following projections on the axes of the local coordinate system of the element will be ignored:

- normal to the surface of the elements of the type 21–30, 121-140;
- moment about the local Z_1 axis for the elements of type 41–50, 141-150.

The loads are specified in the following units of measurement in the basic system of units: concentrated forces in T, forces distributed over an area in T/m^2, forces distributed over a line in T/m, concentrated moments in Tm, moments distributed over an area in Tm/m^2, moments distributed over a line in Tm/m, temperature in Celsius degrees, snaps in m.

Table 3.4-6. Loads on plates and shells in the local coordinate system

Load arrangement	Direction	Values	FE type
Concentrated forces and moments			
	along the X_1 axis along the Y_1 axis	P_X, a, b P_Y, a, b	shells
	along the Z_1 axis about the X_1 axis about the Y_1 axis	P_Z, a, b M_X, a, b M_Y, a, b	plates shells
	about the Z_1 axis	M_Z, a, b	91-97, 191-197, 591-599
Uniformly distributed load			
	along the X_1 axis along the Y_1 axis	q_X q_Y	shells
	along the Z_1 axis about the X_1 axis about the Y_1 axis	q_Z m_X m_Y	plates shells
	about the Z_1 axis	m_Z	91-97, 191-197 591-599
Trapezoidal load			
	along the X axis along the Y axis	$q_X^1, q_X^2, \dots, q_X^n$ $q_Y^1, q_Y^2, \dots, q_Y^n$	shells
	along the Z axis about the X axis about the Y axis	$q_Z^1, q_Z^2, \dots, q_Z^n$ $m_X^1, m_X^2, \dots, m_X^n$ $m_Y^1, m_Y^2, \dots, m_Y^n$	plates shells
	about the Z axis	$m_Z^1, m_Z^2, \dots, m_Z^n$	91-97, 191-197, 591-599
Uniformly distributed load along the line connecting two nodes of the element			
	along the X_1 axis along the Y_1 axis	$n_1\ n_2\ q_X$ [1] $n_1\ n_2\ q_Y$	shells
	along the Z_1 axis about the X_1 axis about the Y_1 axis	$n_1\ n_2\ q_Z$ $n_1\ n_2\ m_X$ $n_1\ n_2\ m_Y$	plates shells
	about the Z_1 axis	$n_1\ n_2\ m_Z$	91-97, 191-197, 591-599
Trapezoidal load along the line connecting two nodes of the element			
	along the X_1 axis along the Y_1 axis	$n_1\ n_2\ q_{X,1}\ q_{X,2}$ $n_1\ n_2\ q_{Y,1}\ q_{Y,2}$	shells
	along the Z_1 axis about the X_1 axis about the Y_1 axis	$n_1\ n_2\ q_{Z,1}\ q_{Z,2}$ $n_1\ n_2\ m_{X,1}\ m_{X,2}$ $n_1\ n_2\ m_{Y,1}\ m_{Y,2}$	plates shells
	about the Z_1 axis	$n_1\ n_2\ m_{Z,1}\ m_{Z,2}$	91-97, 191-197, 591-599

[1] n_1 , n_2 — ordinal numbers of nodes defined in the description of the element.

Table 3.4-7. Loads on plates and shells in the global coordinate system

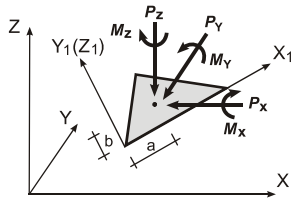
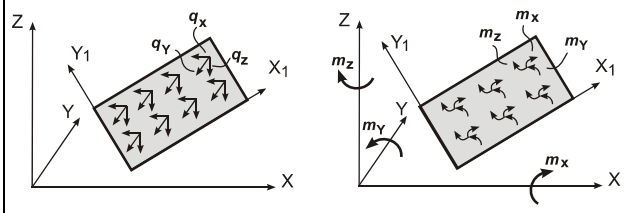
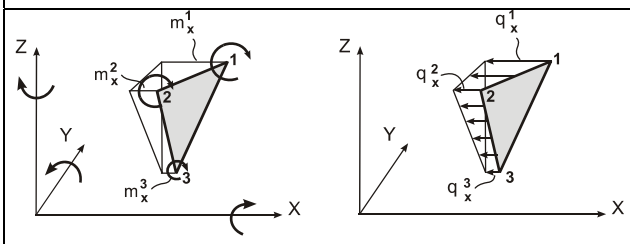
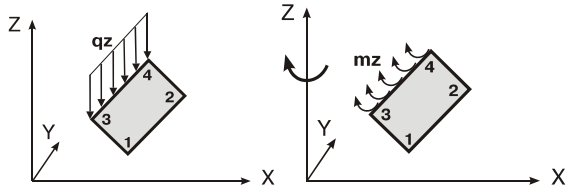
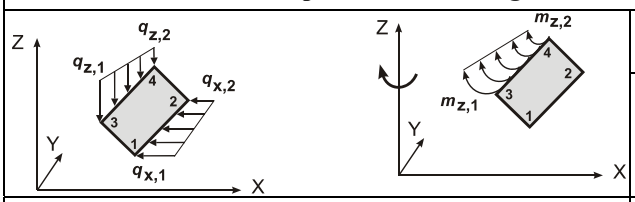
Load arrangement	Direction	Values	FE type
Concentrated load in the local coordinate system			
	along the X axis along the Y axis	P_X, a, b P_Y, a, b	shells
	along the Z axis about the X axis about the Y axis	P_Z, a, b M_X, a, b M_Y, a, b	plates shells
	about the Z axis	M_Z, a, b	shells
Uniformly distributed load			
	along the X axis along the Y axis	q_X q_Y	shells
	along the Z axis about the X axis about the Y axis	q_Z m_X m_Y	plates shells
	about the Z axis	m_Z	shells
Trapezoidal load			
	along the X axis along the Y axis	$q_X^1, q_X^2, \dots, q_X^n$ $q_Y^1, q_Y^2, \dots, q_Y^n$	shells
	along the Z axis about the X axis about the Y axis	$q_Z^1, q_Z^2, \dots, q_Z^n$ $m_X^1, m_X^2, \dots, m_X^n$ $m_Y^1, m_Y^2, \dots, m_Y^n$	plates shells
	about the Z axis	$m_Z^1, m_Z^2, \dots, m_Z^n$	shells
Uniformly distributed load along the line connecting two nodes of the element			
	along the X axis along the Y axis	$n_1\ n_2\ q_X$ $n_1\ n_2\ q_Y$	shells
	along the Z axis about the X axis about the Y axis	$n_1\ n_2\ q_Z$ $n_1\ n_2\ m_X$ $n_1\ n_2\ m_Y$	plates shells
	about the Z axis	$n_1\ n_2\ m_Z$	shells
Trapezoidal load along the line connecting two nodes of the element			
	along the X axis along the Y axis	$n_1\ n_2\ q_{X,1}\ q_{X,2}$ $n_1\ n_2\ q_{Y,1}\ q_{Y,2}$	shells
	along the Z axis about the X axis about the Y axis	$n_1\ n_2\ q_{Z,1}\ q_{Z,2}$ $n_1\ n_2\ m_{X,1}\ m_{X,2}$ $n_1\ n_2\ m_{Y,1}\ m_{Y,2}$	plates shells
	about the Z axis	$n_1\ n_2\ m_{Z,1}\ m_{Z,2}$	shells

Table 3.4-8

Loads in the local coordinate system on the elements for the analysis of the plane stress and plane strain states

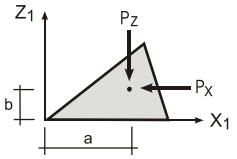
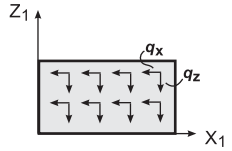
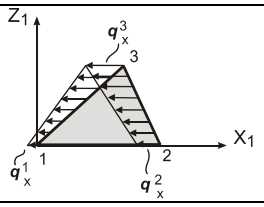
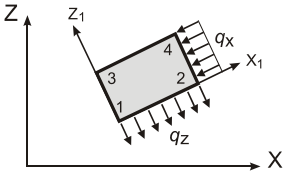
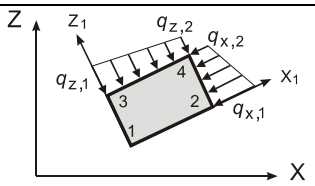
Load arrangement	Direction	Values	FE type
Concentrated load			
	along the X_1 axis along the Z_1 axis	P_X, a, b P_Z, a, b	21-30,121-130, 521-530
	about the Y axis	M_Y, a, b	121-130, 521-530
Uniformly distributed load			
	along the X_1 axis along the Z_1 axis	q_X q_Z	21-30,121-130, 521-530
	about the Y axis	m_Y	121-130, 521-530
Trapezoidal load			
	along the X_1 axis along the Z_1 axis	$q_X^1, q_X^2, \dots, q_X^n$ $q_Z^1, q_Z^2, \dots, q_Z^n$	21-30,121-130, 521-530
	about the Y axis	$m_Y^1, m_Y^2, \dots, m_Y^n$	121-130, 521-530
Uniformly distributed load along the line connecting two nodes of the element			
	along the X_1 axis along the Z_1 axis	$n_1\ n_2\ q_X$ $n_1\ n_2\ q_Z$	21-30,121-130, 521-530
	about the Y axis	$n_1\ n_2\ m_Y$	121-130, 521-530
Trapezoidal load along the line connecting two nodes of the element			
	along the X_1 axis along the Z_1 axis	$n_1\ n_2\ q_{X,1}\ q_{X,2}$ $n_1\ n_2\ q_{Z,1}\ q_{Z,2}$	21-30,121-130, 521-530
	about the Y axis	$n_1\ n_2\ m_{Y,1}\ m_{Y,2}$	121-130, 521-530

Table 3.4-9

Loads in the global coordinate system on the elements for the analysis of the plane stress and plane strain states

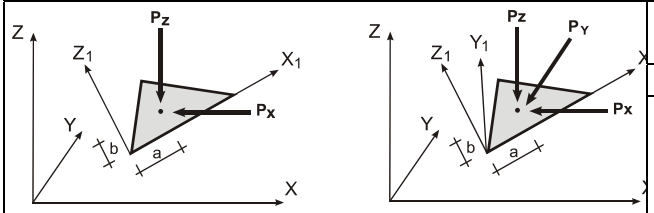
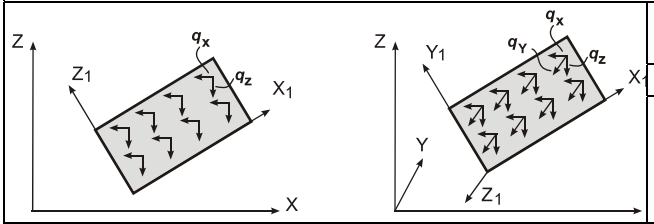
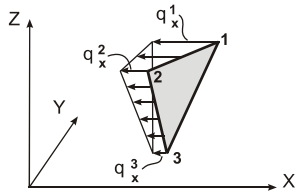
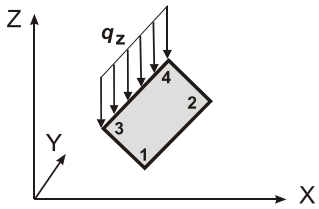
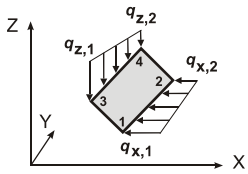
Load arrangement	Direction	Values	FE type
Concentrated load in the local coordinate system			
	along the X axis along the Z axis	P_X, a, b P_Z, a, b	21–30, 121-130, 521-530
	along the Y axis	P_Y, a, b	23, 24, 26, 27, 28
	about the Y axis	M_Y, a, b	121-130, 521-530
Uniformly distributed load			
	along the X axis along the Z axis	q_X q_Z	21–30, 121-130, 521-530
	along the Y axis	q_Y	23, 24, 26, 27, 28
	about the Y axis	m_Y	121-130, 521-530
Trapezoidal load			
	along the X axis along the Z axis	$q_X^1, q_X^2, \dots, q_X^n$ $q_Z^1, q_Z^2, \dots, q_Z^n$	21–30, 121-130, 521-530
	along the Y axis	$q_Y^1, q_Y^2, \dots, q_Y^n$	23, 24, 26, 27, 28
	about the Y axis	$m_Y^1, m_Y^2, \dots, m_Y^n$	121-130, 521-530
Uniformly distributed load along the line connecting two nodes of the element			
	along the X axis along the Z axis	$n_1\ n_2\ q_X$ $n_1\ n_2\ q_Z$	21–30, 121-130, 521-530
	along the Y axis	$n_1\ n_2\ q_Y$	23, 24, 26, 27, 28
	about the Y axis	$n_1\ n_2\ m_Y$	121-130, 521-530
Trapezoidal load along the line connecting two nodes of the element			
	along the X axis along the Z axis	$n_1\ n_2\ q_{X,1}\ q_{X,2}$ $n_1\ n_2\ q_{Z,1}\ q_{Z,2}$	21–30, 121-130, 521-530
	along the Y axis	$n_1\ n_2\ q_{Y,1}\ q_{Y,2}$	23, 24, 26, 27, 28
	about the Y axis	$n_1\ n_2\ m_{Y,1}\ m_{Y,2}$	121-130, 521-530

Table 3.4-10

Thermal load on the flat elements

FE type	Action	Values	Notes
plates shells	Difference of the temperature on the upper and lower surfaces.	Δt, [α]	Δt — difference of the temperature on the upper and lower surfaces; α — linear expansion coefficient of the material; α_x, α_y — linear expansion coefficients for the X_1 or Y_1 axes respectively of the local coordinate system of the element in which the stiffness properties are specified. They enable to take into account the orthotropy of the material; t — increase (decrease) of the temperature of the middle layer of the plate with respect to the normal conditions of operation (restraint temperature see [58]).
		Δt, [α_X]	
		Δt, [α_Y]	
		Δt, [α_X, α_Y]	
21-30 121-130, 521-530 shells	General heating (cooling) with respect to the restraint temperature.	t, [α]	
		t, [α_X]	
		t, [α_Y]	
		t, [α_X, α_Y]	
shells	General heating (cooling) with respect to the restraint temperature and simultaneously difference of the temperature on the upper and lower surfaces.	t, Δt, [α]	
		t, Δt, [α_X]	
		t, Δt, [α_Z]	
Note: Thermal expansion coefficients specified in the stiffness properties of the respective element are used in the calculation. When converting the files of the previous versions the thermal expansion coefficient specified when describing the load is used. The respective warning is generated. Thermal expansion coefficients can be specified for a certain load only when preparing the data in the text format.			

3.5. Finite Elements for Solving the Spatial Problem of the Theory of Elasticity (Solid Elements)

General Information

These elements are used to determine the stress-strain state of continual objects and solid structures. The following designations are used to describe the properties of the material:

ε_x, ε_y, ε_z, γ_{xy}, γ_{xz}, γ_{yz} — relative linear and angular strains;

σ_x, σ_y, σ_z, τ_{xy}, τ_{xz}, τ_{yz} — normal and shear stresses;

E_x, E_y, E_z — Young's moduli for the principal directions of elasticity (if they are equal, the designation E is used for all directions);

G_{xy}, G_{xz}, G_{yz} — shear moduli which characterize the change of the angle between the principal directions (if they are equal, the designation G is used for all directions);

ν, ν_{ij} — Poisson's ratios which characterize a lateral reduction at compression or expansion at tension in the direction of the coordinate axes. The first index indicates the direction of the deformation, and the second one — the direction of the force.

The following variants of the elastic properties of the material are implemented in **SCAD**:

isotropic body:

$$\varepsilon_x = \frac{1}{E}\left(\sigma_x - \nu\sigma_y - \nu\sigma_z\right),\ \varepsilon_y = \frac{1}{E}\left(\sigma_y - \nu\sigma_x - \nu\sigma_z\right),\ \varepsilon_z = \frac{1}{E}\left(\sigma_z - \nu\sigma_x - \nu\sigma_y\right),$$

$$\gamma_{xy} = \frac{1}{G}\tau_{xy};\qquad \gamma_{yz} = \frac{1}{G}\tau_{yz};\qquad \gamma_{zx} = \frac{1}{G}\tau_{zx};$$

transverse isotropic and orthotropic body :

$$\varepsilon_x = \frac{1}{E_x}\sigma_x - \frac{\nu_{12}}{E_y}\sigma_y - \frac{\nu_{13}}{E_z}\sigma_z,\ \varepsilon_y = -\frac{\nu_{21}}{E_x}\sigma_x + \frac{1}{E_y}\sigma_y - \frac{\nu_{23}}{E_z}\sigma_z,\ \varepsilon_z = -\frac{\nu_{31}}{E_x}\sigma_x - \frac{\nu_{32}}{E_y}\sigma_y + \frac{1}{E_z}\sigma_z,$$

$$\varepsilon_{xy} = \frac{1}{G_{xy}}z_{xy},\ \varepsilon_{xz} = \frac{1}{G_{xz}}z_{xz},\ \varepsilon_{yz} = \frac{1}{G_{yz}}z_{yz}.$$

For the transverse isotropic body $E_x = E_y = E$, $\nu_{12} = \nu_{21}$, $G_{xz} = G_{yz}$. It is considered that symmetry conditions hold in all cases:

$E_1\nu_{12} = E_2\nu_{21}$, $E_1\nu_{13} = E_3\nu_{31}$, $E_2\nu_{23} = E_3\nu_{32}$.

Each node of the finite elements has three degrees of freedom: u, v, w — displacements along the x, y, z axes respectively. When the finite elements are created, the displacements (u, v, w) are approximated by the functions of the following type: linear — for a tetrahedron, linear or bilinear for each direction — for a triangular prism and parallelepiped [15], rational — for isoparametric elements [16, 19, 23].

Local Coordinate Systems

Each finite element can have three different local coordinate systems for solving the spatial problem of the theory of elasticity which in the general case coincide:

- local coordinate system of the element;
- coordinate system of the orthotropy axes;
- coordinate system for calculating forces and stresses.

The two latter coordinate systems coincide with the local coordinate system of the element by default.

The finite elements of type 31 and 33 have a local coordinate system, in which the X_1 axis goes from the first node to the second one. The Y_1 axis lies in the plane formed by the first three nodes, it is orthogonal to X_1 and directed towards the third node. The Z_1 axis forms a right-hand system together with the X_1 and Y_1 axes. The local coordinate system coincides with the global one for other elements.

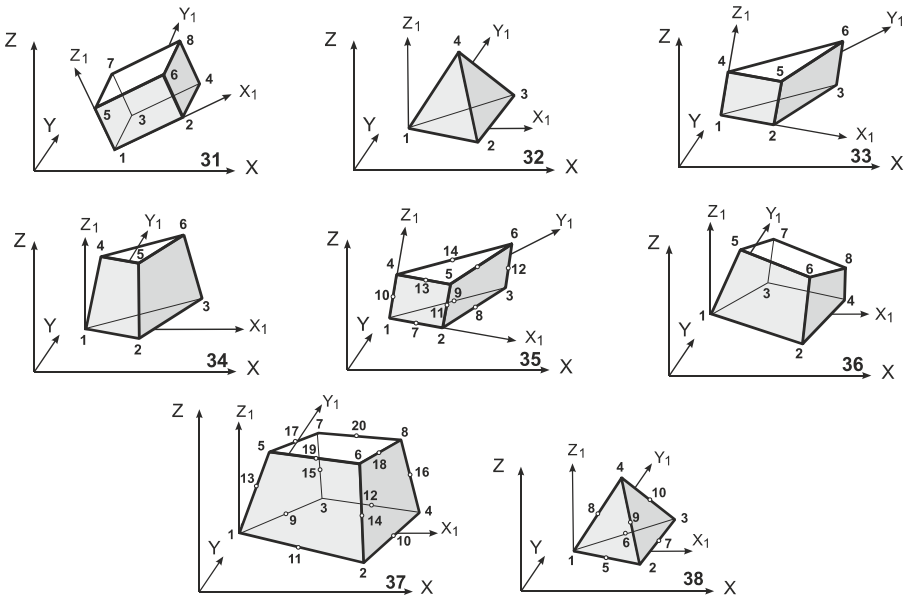
A special coordinate system for the output of forces can be specified for a group of elements. This system is defined by the direction of the X_N and Y_N axes, and the Z_N axis complements the system to a right-hand one.

Set of Finite Elements

FEs for solving the spatial problem of the theory of elasticity can be used in the models with indices 4, 5, 8 and 9. The following stresses are calculated in them: N_X, N_Y, N_Z, T_{XY}, T_{XZ}, T_{YZ} (see Sec. 13.6).

Table 3.5-1 lists finite elements included in **SCAD** together with their local coordinate systems.

Table 3.5-1

FE type	Name	Node numbering and local axes
31	Parallelepiped	
32	Tetrahedron	
33	Triangular prism	
34	Spatial isoparametric six-node finite element	
35	Fifteen-node triangular prism	
36	Spatial isoparametric eight-node finite element	
37	Spatial isoparametric twenty-node finite element	
38	Ten-node pyramid	

The numbers of the nodes lying on the sides of the elements are specified in any order after the numbers of the nodes in the vertices.

Peculiarities of Specifying the Initial Information for Solid Finite Elements

The following stiffness properties are specified depending on the class of the solved problems:

$E(T/m^2)$, ν — for the isotropic material;

$E(T/m^2)$, $E_3(T/m^2)$, ν_{12}, ν_{13}, $G_{13}(T/m^2)$ —for the transverse isotropic material;

$E_1(T/m^2)$, $E_2(T/m^2)$, $E_3(T/m^2)$, ν_{12}, ν_{21}, ν_{13}, ν_{31}, ν_{23}, ν_{32}, $G_{12}(T/m^2)$, $G_{13}(T/m^2)$, $G_{23}(T/m^2)$ — for the orthotropic material.

When calculating the bodies from orthotropic and anisotropic materials it is necessary to control the directions for which the rigidities are specified. They can be specified in the global coordinate system in the following ways:

- ΔX_1, ΔY_1, ΔZ_1, ΔX_2, ΔY_2, ΔZ_2 — by the projections on the axes of the global coordinate system of two vectors which define the directions of the orthotropy axes X_A and Y_A. The Z_A axis is orthogonal to them, and the X_A, Y_A, Z_A system is right-hand;
- in the cylindrical coordinate system when it is necessary to specify X_C, Y_C, Z_C — the coordinates of the point through which the axis of the cylinder passes, and ΔX, ΔY, ΔZ — vector which defines the direction of the axis of the cylinder. The X_A axis is orthogonal to the axis of the cylinder and passes through the point lying on it. The Z_A axis lies of the generatrix of the cylinder;
- in the spherical coordinate system when it is necessary to specify X_C,Y_C, Z_C — coordinates of the center of the sphere and ΔX, ΔY, ΔZ — vector which defines the direction of the Y_A axis. The X_A axis goes from the first node of the element to the center of the sphere.

You can change the direction of the axes by specifying the Euler angles of rotation of the local axes.

When the results are used, it is necessary to specify the direction of the axes for the stress calculation. Their specification in the global coordinate system is completely identical to the specification of the orthotropy axes.

The stresses (forces) are calculated for all solid finite elements both in their center and in their nodes. The control of their output is described in Chapters 7 and 13.

Loads on Solid Finite Elements

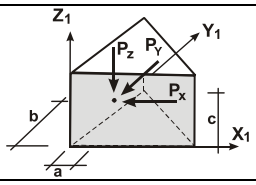
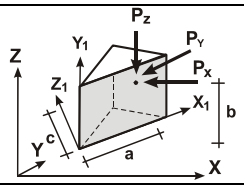
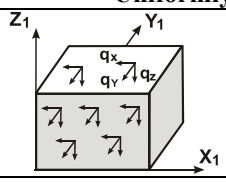
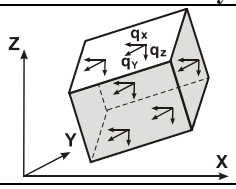
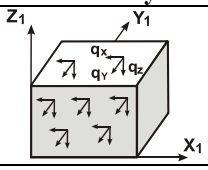
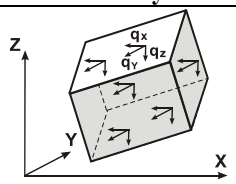
The following loads can be applied to the finite elements used for solving the spatial problem of the theory of elasticity:

- concentrated loads which are specified as the values of forces acting in the given direction and applied in the point on the surface of the element. The load and the coordinates of the point of its application can be specified both in the local coordinate system of the element and in the global coordinate system of the problem;
- uniformly distributed loads which act in the given direction and are specified by the value of the force per unit volume. The load acts over the entire volume of the element and can be specified both in the local and in the global coordinate systems;
- self-weight which acts along the Z axis of the global coordinate system. The specific weight of the material and the self-weight factor are specified for the element;
- trapezoidal loads on the finite elements of type 31 and 33 which act in the given direction and are specified as the force values determined in all vertices of the element. The load acts over the entire volume of the element where it is approximated by a linear law in each direction. The direction of its action can be specified both in the local and in the global coordinate systems. The order of the specification of the load values must correspond to the order of numbering of the nodes of the element described in this section;
- uniformly distributed loads over the face of the element which act in the given direction. They are defined by the numbers of the faces in accordance with the description of the element given in this section (see Fig. 3.5-1), and the value of the force or moment per unit area. The load can be specified both in the local and in the global coordinate systems;
- thermal action in the form of the general heating (cooling) with respect to the restraint temperature.

The loads are specified in the following units of measurement in the basic system of units: concentrated forces in T, forces distributed over an area in T/m^2, forces distributed over a volume in T/m^3, temperature in Celsius degrees, snaps in m.

The main local loads for solid elements are given in Table 3.5-2.

Table 3.5-2

Load arrangement	Direction	Values	FE type
Concentrated load in the local coordinate system			
	along the X_1 axis along the Y_1 axis along the Z_1 axis	P_X, a, b, c P_Y, a, b, c P_Z, a, b, c	31, 33
Concentrated load in the global coordinate system, snaps in the local coordinate system			
	along the X axis along the Y axis along the Z axis	P_X, a, b, c P_Y, a, b, c P_Z, a, b, c	31, 33
Uniformly distributed load in the local coordinate system			
	along the X_1 axis along the Y_1 axis along the Z_1 axis	q_X q_Y q_Z	31, 33
Uniformly distributed load in the global coordinate system			
	along the X axis along the Y axis along the Z axis	q_X q_Y q_Z	31–40
Uniformly distributed load over a face in the local coordinate system			
	along the X_1 axis along the Y_1 axis along the Z_1 axis	$q_X\ n$[1] $q_Y\ n$ $q_Z\ n$	31, 33
Uniformly distributed load over a face in the global coordinate system			
	along the X axis along the Y axis along the Z axis	$q_X\ n$ $q_Y\ n$ $q_Z\ n$	31–40

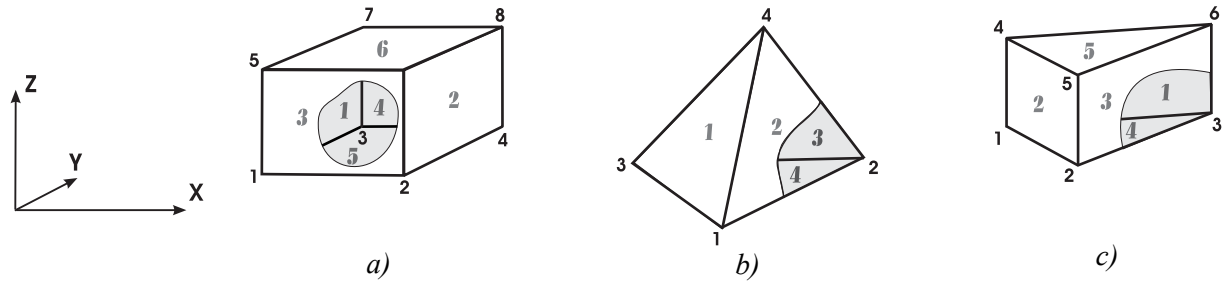


Figure 3.5-1. *Numbering of the faces of the load application in the solid elements*

[1] n — number of the face of the element (see Fig. 3.5-1)

Thermal load on the solid elements **Table 3.5-3**

Action	Values	Notes
General heating (cooling) with respect to the restraint temperature.	t, [α]	t — increase (decrease) of the temperature of the middle layer of the plate with respect to the normal conditions of operation (restraint temperature); α — linear expansion coefficient of the material; α_X, α_Y, α_Z — linear expansion coefficients for the X_1 or Y_1 axes respectively of the local coordinate system of the element in which the stiffness properties are specified. They enable to take into account the orthotropy of the material.
	t, [α_X]	
	t, [α_Y]	
	t, [α_Z]	
	Δt, [α_X, α_Y, α_Z]	
Note: Thermal expansion coefficients specified in the stiffness properties of the respective element are used in the calculation. When converting the files of the previous versions the thermal expansion coefficient specified when describing the load is used. The respective warning is generated. Thermal expansion coefficients can be specified for a certain load only when preparing the data in the text format.		

3.6. Universal Finite Elements for Solving the Axisymmetric Problem of the Theory of Elasticity

General Information

These elements are used to determine the stress-strain state of the solids of revolution (axisymmetric rigid bodies) from homogeneous (in the direction of any axis r, Z of the element) isotropic, transverse isotropic or orthotropic linear elastic material subjected to an axisymmetric load [60]. If the deformation is axisymmetric, the stress-strain state in any section along the symmetry axis of the body is considered as a function of radius r and height Z only and it is defined by two components of displacements. Such sections are shown in Fig. 3.6-1. One of the following cases of the elastic symmetry can be considered:

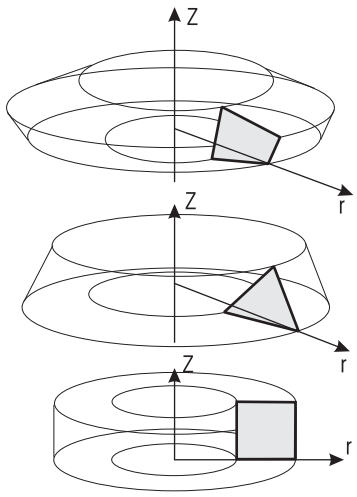


Figure 3.6-1.

- full symmetry (isotropic body);
- isotropic plane (transverse isotropic body);
- three planes of elastic symmetry (orthotropic-anisotropic or orthotropic body).

The following designations are used to describe them:

- ε_r, ε_z, ε_θ, γ_{rz} — relative linear and angular strains;
- σ_r, σ_z, τ_{rz} — normal and shear stresses in the cross-sectional plane;
- σ_θ — normal stress for the direction orthogonal to the cross-sectional plane;
- E, E_i — Young's moduli for the principal directions of elasticity;
- G — shear modulus;
- ν,ν_{ik} — Poisson's ratios which characterize a lateral reduction at compression or expansion at tension in the directions of the coordinate axes. The first index indicates the direction of the deformation, and the second one – the direction of the force.

Relative linear and angular strains $\varepsilon_r, \varepsilon_z, \varepsilon_\theta, \gamma_{rz}$ can be expressed in terms of the displacements in the following way:

$$\varepsilon_z = \frac{\partial v}{\partial z},\ \varepsilon_r = \frac{\partial u}{\partial r},\ \varepsilon_\theta = \frac{u}{r},\ \gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial v}{\partial r}.$$

There are the following relationships between the strains and stresses:

isotropic body:

$$\varepsilon_z = \frac{1}{E}\left[\sigma_z - \nu\left(\sigma_r + \sigma_\theta\right)\right],$$

$$\varepsilon_r = \frac{1}{E}\left[\sigma_r - \nu\left(\sigma_z + \sigma_\theta\right)\right],$$
$$\varepsilon_\theta = \frac{1}{E}\left[\sigma_\theta - \nu\left(\sigma_z + \sigma_r\right)\right],$$
$$\gamma_{rz} = \tau_{rz} / G,$$
$$G = \frac{E}{2(1+\nu)};$$

transverse isotropic body :

$$\varepsilon_r = \frac{1}{E}\sigma_r - \frac{\nu}{E}\sigma_\theta - \frac{\nu'}{E_z}\sigma_z,$$
$$\varepsilon_\theta = -\frac{\nu}{E}\sigma_r + \frac{1}{E}\sigma_\theta - \frac{\nu'}{E_z}\sigma_z,$$
$$\varepsilon_z = -\frac{\nu''}{E}\left(\sigma_r + \sigma_\theta\right) + \frac{1}{E_z}\sigma_z,$$
$$\gamma_{rz} = \tau_{rz} / G_r,$$
$$\nu' = \nu_{rz} = \nu_{\theta z}, \nu'' = \nu_{zr} = \nu_{z\theta};$$

orthotropic body *:*

$$\varepsilon_z = \frac{1}{E_z}\sigma_z - \frac{\nu_{zr}}{E_r}\sigma_r - \frac{\nu_{z\theta}}{E_\theta}\sigma_\theta,$$
$$\varepsilon_r = \frac{1}{E_r}\sigma_r - \frac{\nu_{r\theta}}{E_\theta}\sigma_\theta - \frac{\nu_{rz}}{E_z}\sigma_z,$$
$$\varepsilon_\theta = \frac{1}{E_\theta}\sigma_\theta - \frac{\nu_{\theta r}}{E_r}\sigma_r - \frac{\nu_{\theta z}}{E_z}\sigma_z.$$
$$\gamma_{rz} = \tau_{rz} / G$$

It is considered that symmetry conditions hold: $E_r\nu_{r\theta} = E_\theta\nu_{\theta r}$, $E_z\nu_{zr} = E_r\nu_{rz}$, $E_\theta\nu_{\theta z} = E_z\nu_{z\theta}$, which are obligatory for any anisotropic material where the existence of an elastic potential can be postulated.

Set of Finite Elements

Like in the case of the plane problem of the theory of elasticity the approximating functions were used when creating elements.

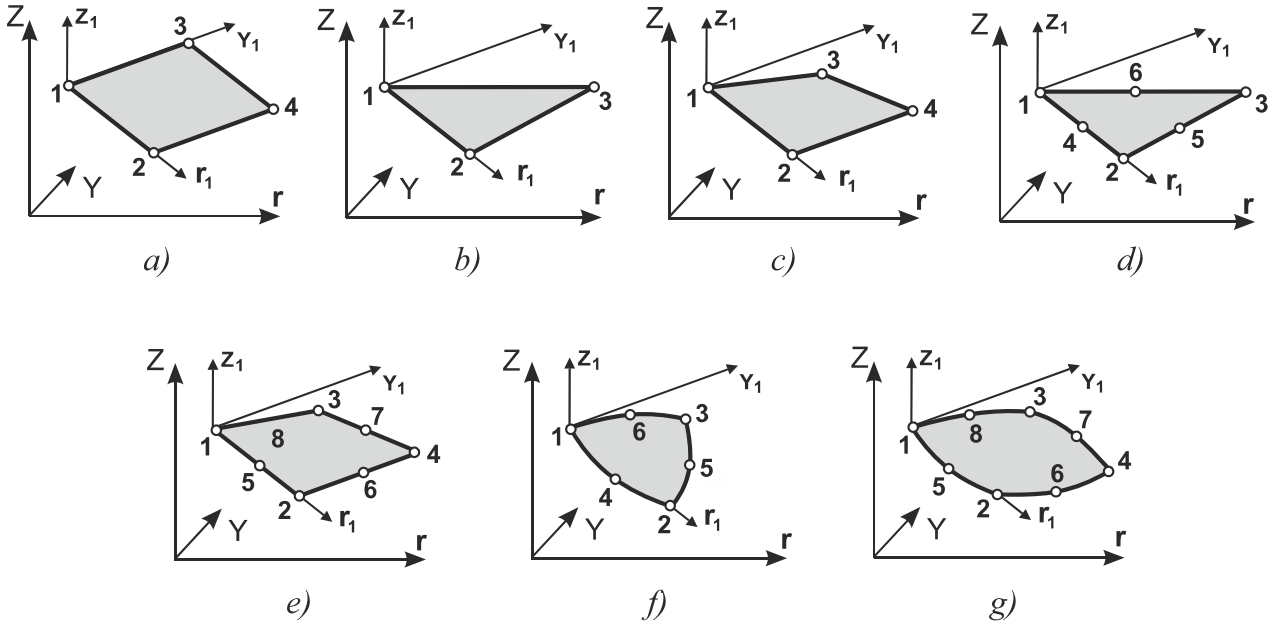


Figure 3.6-2. *Axisymmetric finite elements*

There are the following groups of element types that can lie in the rOz plane:

- having two degrees of freedom in each node — R and W, which are the tangential displacements, along the axes R and Z respectively. FEs can be used in models with indices 11 and 12;
- having three degrees of freedom in each node — R and W, and U – rotation angle about the normal to the ROZ plane [1]. FEs should only be used in models with indice 12.

Table 3.6-1. Types of elements for the axisymmetric problem of the theory of elasticity

Degrees of freedom of nodes	Type	Name	Number of nodes	Node numbering order and local axes	Comments
R, W	61	Rectangular	4	Fig. 3.6-2, *a*	polylinear shape functions
	62	Triangular	3	Fig. 3.6-2, *b*	linear shape functions
	64	Quadrangular	4	Fig. 3.6-2, *c*	*SubAreas* [26, 23]
	66	Quadrangular	4-8	Fig. 3.6-2, *g*	isoparametric
	65	Triangular	3-6	Fig. 3.6-2, *d*	*SubAreas* [26, 23]
	68	Triangular	3-6	Fig. 3.6-2, *f*	isoparametric
	70	Quadrangular	4-8	Fig. 3.6-2, *e*	*SubAreas* [26, 23]
Elements with drilling (*DDF*) and quasi-rotational degrees of freedom (QRDF)					
R, W, U	162	Triangular	3	Fig. 3.6-2, *b*	*DDF* [90, 23]
	164	Quadrangular	4	Fig. 3.6-2, *c*	*DDF, SubAreas* [90, 23]
	165	Triangular	3-6	Fig. 3.6-2, *d*	*DDF, SubAreas* [90, 23]
	170	Quadrangular	4-8	Fig. 3.6-2, *e*	*DDF, SubAreas* [90, 23]
	562	Triangular	3	Fig. 3.6-2, *b*	*QRDF* [90, 23]
	564	Quadrangular	4	Fig. 3.6-2, *e*	*QRDF, SubAreas* [90, 23]
	566	Quadrangular	4	Fig. 3.6-2, *e*	*QRDF* [90, 23]

[1] This can be either an average rotation angle or a quasi-rotational degree of freedom [90].

The local coordinate system of all axisymmetric elements coincides with the global one, and the forces will be calculated in the global coordinate system by default. The following stresses are calculated: N_X (N_r), N_Y (N_θ), N_Z, T_{XZ} (T_{rZ}) (see Sec. 13.6).

Peculiarities of Specifying the Initial Information for Axisymmetric Finite Elements

All elements of this type must lie in the symmetry plane rOz and the axis normal to this plane Y_1 always coincides with the Y axis. Since a spatial problem of the theory of elasticity with axial symmetry is solved for the axisymmetric FEs, the stiffness properties must be specified in the same way as those of solid finite elements with the symmetry limitations imposed by the considered problem.

Only special finite elements described in Section 3.8 can be included in the design model together with the given elements.

When specifying the nodal forces it is necessary to multiply the value of the force acting on the unit length of a circular arc by the perimeter of this circle.

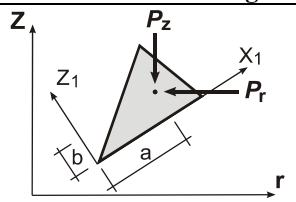
Local Loads

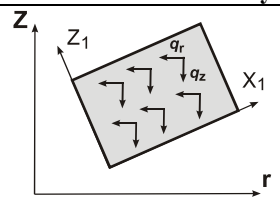
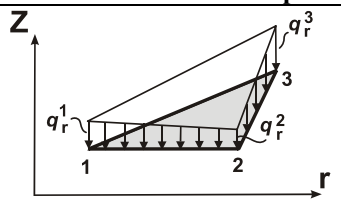
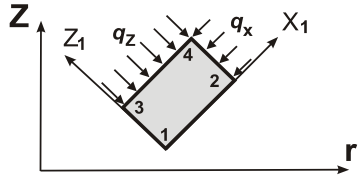
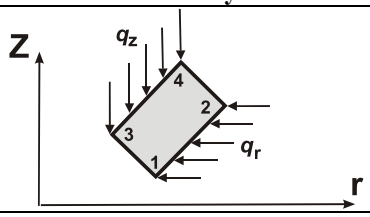
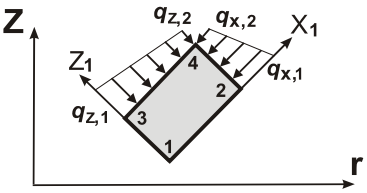
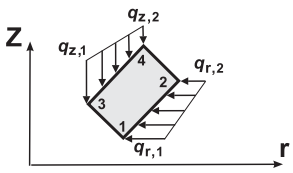
An auxiliary local coordinate system for specifying a load is defined on the finite element for solving an axisymmetric problem of the theory of elasticity. It is generated in the same way as the local coordinate system X_1OZ_1 for the elements of the plane problem of the theory of elasticity (see Sec 3.4). The X_1 axis goes from the first node to the second one, and the Z_1 axis lies in the rOz plane and it is directed towards the third node. Only uniformly distributed and trapezoidal loads along the line can be specified in the local coordinate system. A local load on the axisymmetric finite elements can be specified both in the local and in the global coordinate system.

The following local loads can be applied:

- concentrated loads which are specified as the values of forces acting in the given direction and applied in the point of the element;
- uniformly distributed loads which act in the given direction and are specified by the value of the force per unit volume. The load acts over the entire volume obtained by revolving the element about the symmetry axis Z;
- trapezoidal loads which act in the given direction and are specified as the force values determined in all vertices of the element. The load acts over the entire volume obtained by revolving the element about the symmetry axis Z. The order of the specification of the load values must correspond to the order of numbering of the nodes of the element described in this section;
- uniformly distributed loads “along the line” which act in the given direction. They actually act on a surface obtained by revolving a line connecting two nodes of the element about the Z axis. The beginning and end of the line are defined by the node numbers described in this section and the value of the force per unit area;
- trapezoidal loads “along the line” which act in the given direction on the line connecting two nodes of the element. I.e. they actually act on a surface obtained by revolving the line the beginning and end of which are defined by the node numbers described in this section about the Z axis. The value of the force per unit area in the beginning and in the end of the line is specified;
- thermal action in the form of the general heating/cooling with respect to the restraint temperature.

Table 3.6-2

Load arrangement	Direction	Values
Concentrated load in the global coordinate system with the snaps in the local coordinate system		
	along the radius along the Z axis	P_r, a, b P_z, a, b

Load arrangement	Direction	Values
Uniformly distributed load in the global coordinate system		
	along the radius along the Z axis	q_r q_z
Trapezoidal load in the global coordinate system		
	along the radius along the Z axis	$q_r^1, q_r^2, \dots, q_r^n$ $q_z^1, q_z^2, \dots, q_z^n$
Uniformly distributed load along the line in the local coordinate system		
	along the X_1 axis along the Z_1 axis	n_1 n_2 q_x n_1 n_2 q_z
Uniformly distributed load along the line in the global coordinate system		
	along the radius along the Z axis	n_1 n_2 q_r n_1 n_2 q_z
Trapezoidal load along the line in the local coordinate system		
	along the X_1 axis along the Z_1 axis	n_1, n_2, $q_{x,1}$, $q_{x,2}$ n_1, n_2, $q_{z,1}$, $q_{z,2}$
Trapezoidal load along the line in the global coordinate system		
	along the radius along the Z axis	n_1, n_2, $q_{r,1}$, $q_{r,2}$ n_1, n_2, $q_{z,1}$, $q_{z,2}$

Thermal load on the axisymmetric elements **Table 3.6-3**

Action	Values	Notes
General heating (cooling) with respect to the restraint temperature.	Δt, [α]	Δt — increase (decrease) of the temperature of the middle layer of the plate with respect to the normal conditions of operation (restraint temperature); α — linear expansion coefficient of the material; α_R, α_Z — linear expansion coefficients for the r or Z axes respectively of the global coordinate system. They enable to take into account the orthotropy of the material.
	Δt, [α_R]	
	Δt, [α_Z]	
	Δt, [α_R, α_Z]	
Note: Thermal expansion coefficients specified in the stiffness properties of the respective element are used in the calculation. When converting the files of the previous versions the thermal expansion coefficient specified when describing the load is used. The respective warning is generated. Thermal expansion coefficients can be specified for a certain load only when preparing the data in the text format.		

3.7. Finite Elements for the Calculation of the Piecewise Inhomogeneous throughout the Thickness (Laminated) Plates and Shallow Shells

Specifying Finite Elements

These finite elements are used for the strength analysis of thin and medium-thickness single-layer and multilayer shallow orthotropic (isotropic, transverse isotropic) shells and plates of laminated structure throughout the thickness (Fig. 3.7-1) with drastically different elastic properties of the layers, in which case the Kirchhoff–Love hypothesis of straight normals becomes inapplicable, because it may yield distorted results [7, 50]. The effect of the deformations from the transverse shear, compression of layers and curvature, as well as those of the interlayer shear and curvature on the stress-strain state of the structure is taken into account.

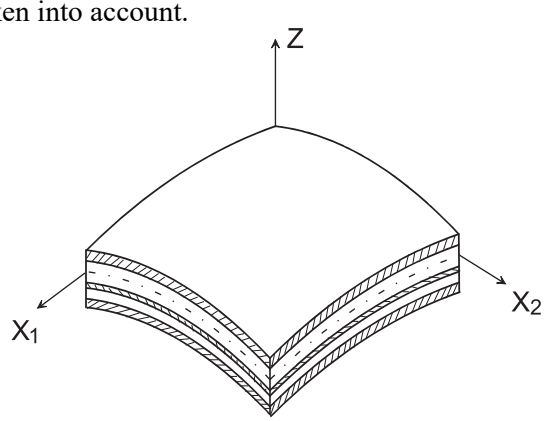


Figure 3.7-1

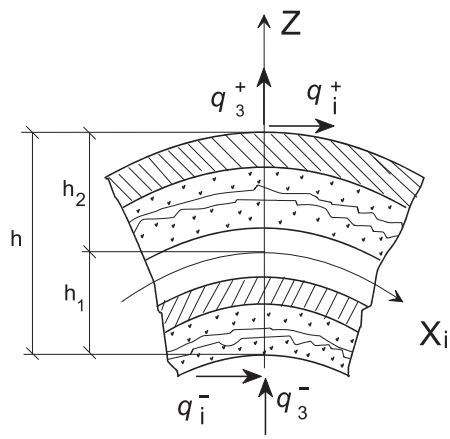


Figure 3.7-2

Models taking into account the variation of the displacements throughout the thickness of the package of layers different from the linear distribution are used when generating the governing relationships. Finite elements are created on the basis of the following original hypotheses and relations of the applied theory of multilayer orthotropic shallow shells and plates [7, 50].

Consider a multilayer shell of a constant thickness (h=const) referred to an orthogonal system of curvilinear coordinates X_1, X_2, Z (Fig. 3.7-2), the coordinate lines X_1, X_2 of which coincide with the lines of the principal curvatures of the reference surface (Z=0), and the Z axis is normal to that surface. The number of layers and their order are arbitrary.

In the most general case the material of layers can be orthotropic but its physical and mechanical characteristics within the k-th layer are assumed to be constant (k=1, 2,...,n), though they can vary from one layer to another throughout the thickness of the package.

The boundary surfaces of the layers coincide with the equidistant surfaces $Z = Z_k$ (k =2, 3,..., n), where $Z = Z_1 = -h_1$ is the lower boundary surface, and $Z = Z_{n+1} = h_2$ is the upper boundary surface (Fig. 3.7-2). The shell is subjected to an arbitrarily oriented load $q^-_j(x_i)$ and $q^+_j(x_i)$ (i=1, 2; j=1, 2, 3), applied to the upper and lower surfaces, and body forces F_j (X_i, Z), acting in the direction of the coordinate axes.

Values that describe the geometric shape of the shell surface (coefficients of the first quadratic form A_i and principal curvatures K_{ik}; i, k = 1, 2) are assumed to be constant within one finite element. The error caused by this assumption is proportional to the size of the finite element.

Like in the case of the single-layer shell elements the approximating functions were used when creating elements.

Initial Hypotheses of the Model Accounting for the Transverse Shear, Compression of Layers and Curvature

The mathematical model is based on the hypotheses which define the law of the variation of the shear stresses σ_{j3}, the normal stress σ_{33}, and the lateral strain e_{33} throughout the thickness of a multilayer shell:

$$\sigma_{j3} = q_j^+ \frac{h_1 + z}{h} + q_j^- \frac{h_2 - z}{h} + G_{j3}(z) \quad f_j'(z)\psi_j(x_1, x_2),$$

$$\sigma_{33} = q_j^+ \frac{h_1 + z}{h} + q_j^- \frac{h_2 - z}{h} + \varphi \quad (z)p(x_1, x_2),$$

$$e_{33} = f_3''(z)\psi_3(x_1, x_2), \qquad j=1, 2 \ ,$$

where G_{j3} — piecewise continuous functions that describe the respective shear moduli of the layers;

$\psi_s(x_1, x_2)$ — sought-for functions which enable to allow for the transverse shear (s=1, 2) and compression (s=3). They complement the classical displacement functions $v_j(x_1, x_2)$ (j=1,2) and $w(x_1, x_2)$;

$f_s(z)$, $\varphi(z)$ — specified auxiliary functions describing the nonlinear law of the variation of stresses and strains depending on the stiffness properties. Their notation enables to select the position of the reference surface arbitrarily within the thickness of the package of layers, satisfies the conditions of continuity between the layers and the boundary conditions on the outer surfaces of the shell;

$p(x_1, x_2)$ — function that characterizes the lateral compression and is defined by the sought-for functions v_j, w and ψ_s.

The auxiliary functions $f_s(z)$, (s =1, 2, 3) are selected on the basis of the law of distribution of the values σ_{j3} (j = 1, 2) and σ_{33} throughout the shell thickness obtained based on the classical hypotheses, and the function of the lateral normal stress distribution throughout the thickness of the laminated shell $\varphi(z)$ is specified based on the condition of continuity and equality to zero on the outer surfaces of the shell $\varphi(-h_1) = \varphi(h_2) = 0$. The procedure of the determination and the form of these functions is given in [7].

The geometric model of the deformed state of an elastic laminated shell which is obtained from the Cauchy relationships and the assumed hypotheses, and enables to express the values of the displacements in any point in terms of the values of the sought-for functions $v_i(x_1, x_2)$, $w(x_1, x_2)$ and $\psi_s(x_1, x_2)$, has the following form:

$$u_j(x_1, x_2, z) = v_j(x_1, x_2) - \frac{1}{A_j} z \frac{\partial w}{\partial x_j} + f_j(z)\psi_j(x_1, x_2) - f_3(z)\frac{1}{A_j}\frac{\partial \psi_3}{\partial x_j} + u_j^{(0)}$$

$$u_3(x_1, x_2, z) = w(x_1, x_2) + f_3'(z)\psi_3(x_1, x_2), \qquad j = 1, 2,$$

where $u_j^{(0)}$ — additional terms corresponding to the loading of the outer surfaces of the shell by the tangential loads.

This model enables to take into account the displacements caused by the phenomena of transverse shear and normal compression in addition to the displacements determined on the basis of the classical shell theory.

The strain components in the reference plane e_{11}, e_{22}, e_{12} are determined by the Cauchy formulae in terms of the displacements using the simplifications usual for the shallow shells, and the transverse shear strain components are determined by the following relationship:

$$e_{j3} = \frac{1}{2}\left[f_j'(z)\psi_j(x_1, x_2) + u_{j,3}^{(0)}\right], \qquad j = 1, 2.$$

The stress tensor σ_{ij} (i, j = 1, 2, 3) is determined on the basis of the generalized Hooke's law for an orthotropic elastic body.

The generalized displacements that make up a vector of generalized displacements can be divided into two groups of displacements that have the same mathematical meaning:

a) v_i and ψ_i (i=1,2) — "membrane" group of generalized displacements included in the potential energy functional with their first-order derivatives at the most;

b) w and ψ_3 — "flexural" group of generalized displacements included in the potential energy functional with their second-order derivatives at the most.

Table 3.7-1

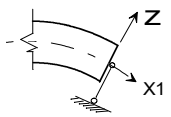
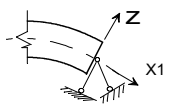
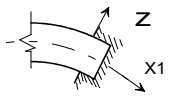
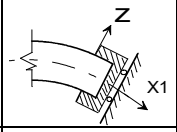
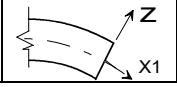
Boundary conditions that define the general fixation of the shell contour for the edge (X_1, X_2=const)					
External constraints (Z=0)	D r a w i n g	Geometric (kinematic) conditions			
		u_n	u_r	w	$\frac{\partial w}{\partial n}$
Roller support			*	*	
Hinge support		*	*	*	
Stiff clamping		*	*	*	*
Movable clamping		*	*		*
Free edge					

Table 3.7-2

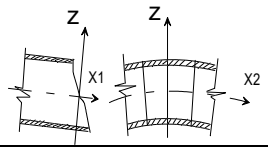
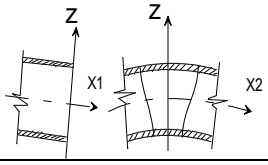
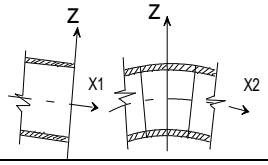
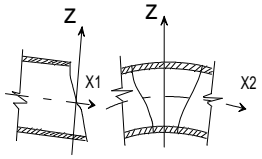
Boundary conditions that restrain the transverse shear and compressive deformations at the shell contour for the edge (X_1, X_2=const)					
End diaphragms (Z = 0)	D r a w i n g	Geometric (kinematic) conditions			
		ψ_n	ψ_τ	$\frac{d\psi_3}{dn}$	$\frac{d\psi_3}{d\tau}$
Flexible out of the end plane			*	*	
Flexible in the end plane		*			*
Stiff non-distortable		*	*	*	*
No diaphragm					

Table 3.7-1 provides various variants of external and end constraints. The former define the general fixation of the contour (support, clamping, etc.) and the latter restrain the transverse shear and compressive deformations at the contour. The asterisk "*" designates the degrees of freedom along which either the displacements or rotation angles are equal to zero. The contour of the shell (plate) or its part with a certain type of end diaphragms can have any external constraints (see Table 3.7-2). The following designations are used in the tables:

u_n, u_τ — displacements along the normal and along the tangent line to the contour lying on the surface of the shell;

ψ_n, ψ_τ — projection of the generalized displacements Ψ_1, Ψ_2 on n and τ;

$\frac{\partial w}{\partial n}$, $\frac{\partial \psi_3}{\partial n}$, $\frac{\partial \psi_3}{\partial \tau}$ — derivatives in the normal and tangential directions to the contour.

Thus, combining the groups of constraints given in the tables makes it possible to analyze shells and plates with various structural peculiarities at their contour; i.e. the boundary conditions enable to take into

account different types of stiffness diaphragms at the ends of the structure, which significantly expands the class of the problems that can be solved.

Initial Hypotheses of the Model Accounting for the Interlayer Shear and Curvature

If the compression of layers is not taken into account, the mathematical model can be based on the hypothesis which defines the law of the variation of the transverse shear stresses $\sigma_{j3}(x_i, z)$ throughout the thickness of a multilayer shell, and the assumption that there are no lateral normal stresses and strains:

$$\sigma_{j3} = G_{j3}(z)\sum_{p=1}^{2} f_{pj}(z)\psi_{pj}(x_1, x_2),$$

$$\sigma_{33}, e_{33} = 0; \qquad j = 1,2;$$

where $\psi_{pj}(x_1, x_2)$ — sought-for shear functions of the first and second approximation (at p = 1 — they supplement the classical displacement functions $v_j(x_1, x_2)$ in the first approximation and refine the hypothesis at the second stage of the model generation; at p = 2 — they allow for a contribution of the tangential loads to the shear stresses);

$f_{pj}(z)$ — specified functions of the distribution of the transverse shear throughout the thickness of the package of layers selected in such a way so as to satisfy the conditions of equality of the transverse shear stresses on the contact surfaces of layers and boundary conditions on the outer surfaces of the shell (at p = 2 — they allow for the effect of the properties of the material of layers on the shear stresses due to the action of the tangential loads);

s — number of the respective approximation of the shear model.

The kinematic model of the deformed state of an elastic laminated shell which is obtained from the Cauchy relationships and the assumed hypotheses, and enables to express the values of the displacements in any point in terms of the values of the sought-for functions $v_j(x_1, x_2)$, $w_j(x_1, x_2)$, $\psi^{(s)}_{pj}(x_1, x_2)$, has the following form:

$$u_j(x_1, x_2, z) = v_i(x_1, x_2) - \frac{1}{A_j} z \frac{\partial w}{\partial x_j} - \sum_{p=1}^{2} f_{pj}(z)\psi_{pj}(x_1, x_2).$$

This model enables to take into account the displacements caused by the phenomena of transverse shear in addition to the displacements determined on the basis of the classical model, as well as the tangential loads on the outer surfaces of the shell — at p = 2.

The strain components in the reference plane e_{ij} are determined by the Cauchy formulae in terms of the displacements using the simplifications usual for the shallow shells. The transverse shear strain components are determined by the following relationships:

$$e_{j3} = \frac{1}{2} f'_{pj}(z)\psi_{pj}(x_1, x_2), \qquad j = 1, 2.$$

The stress tensor σ_{ij} (i, j=1, 2, 3) is determined on the basis of the generalized Hooke's law for an orthotropic elastic body.

The generalized displacements that make up a vector of generalized displacements can be divided into two groups of displacements that have the same mathematical meaning:

a) v_i and ψ_{pi} — “membrane” group of generalized displacements included in the potential energy functional with their first-order derivatives at the most (i, p=1, 2);

b) w — “flexural” group of generalized displacements included in the potential energy functional with their second-order derivatives at the most.

Tables 3.7-1 and 3.7-2 provide various variants of external and end constraints (the former define the general fixation of the contour, and the latter restrain the transverse shear deformations at the contour). The asterisk “*” designates the degrees of freedom along which either the displacements or rotation angles are equal to zero. The contour of the shell (plate) or its part with a certain type of end diaphragms can have any external constraints (see Table 3.7-2), where n and τ — normal and tangent to the contour lying on the surface of the shell, Ψ_n, Ψ_τ — projections of the functions Ψ_1 and Ψ_2 on n and τ.

Combining the groups of constraints given in Tables 3.7-1 and 3.7-2 makes it possible to analyze shells and plates with various structural peculiarities at their contour; i.e. the boundary conditions enable to

take into account different types of stiffness diaphragms at the ends of the structure, which significantly expands the class of the problems that can be solved.

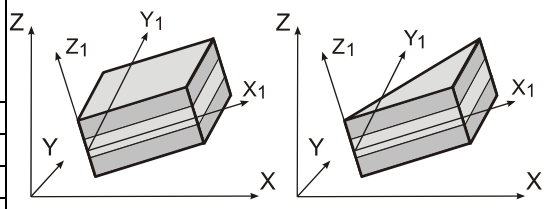
It should be noted that when the effect of the compression of layers or interlayer shear in the considered FEs is taken into account, it becomes possible to analyze even single-layer structures such as medium-thickness shells and plates.

Multilayer Elements in the FE Library

Table 3.7-3 lists multilayer elements included in **SCAD**.

Nodes lie on the midsurface. All elements have a local coordinate system X_1OY_1, where the X_1 axis goes from the first node to the second one, and the Y_1 axis lies in the plane of the element, it is orthogonal to X_1 and directed towards the third node.

Table 3.7-3

Shape	Element type		Local axes
	Allowing for the transverse shear, compression of layers and curvature	**Allowing for the interlayer shear and curvature**	
Rectangular	71	81	
Triangular	72	82	
Quadrangular	73	83	
Quadrangular with the number of nodes from 4 to 8	74	84	

The material of the layers can be isotropic, transverse isotropic, or orthotropic. The Winkler model of the elastic subgrade with one subsoil parameter C_1 can be used.

Multilayer elements of types 71–74 allowing for the transverse shear, compression of layers and curvature have twelve degrees of freedom in their nodes, the interpretation of which is given in Table 3.7-4.

Table 3.7-4

Name	Types of the degrees of freedom	Content
U, V, W	1, 2, 3	linear displacements along the X, Y, Z axes
$U_X = -\frac{\partial w}{\partial x}$, $U_Y = \frac{\partial w}{\partial y}$, U_Z	4, 5, 6	rotation angles about the X, Y, Z axes
$A_X=\Psi_1$, $A_Y=\Psi_2$, $A_Z=\Psi_3$	7, 8, 9	analogues of the linear displacements along the X, Y, Z axes, which model the compression of layers in the respective direction
$B_X = -\frac{\partial \psi_3}{\partial x}$, $B_Y = \frac{\partial \psi_3}{\partial y}$, B_Z	10, 11, 12	analogues of the rotation angles about the X, Y, Z axes, which model the effect of the normal compressive deformations

The first six degrees of freedom of the node are the main ones and correspond to the membrane and bending deformations, and the second ones correspond to the transverse shear and compressive deformations.

The rotation angle U_Z and its analogue B_Z in the local coordinate system of the element are equal to zero. They appear when the elements not lying in the same plane are connected and are required to take into account the spatial behavior of the structure.

The considered elements can be used only with the index of the model type 8.

Multilayer elements of types 81-84 allowing for the interlayer shear and curvature have twelve degrees of freedom in their nodes, the interpretation of which is given in Table 3.7-5.

Table 3.7-5

Name	Types of the degrees of freedom	Content
U, V, W	1, 2, 3	linear displacements along the X, Y, Z axes
$U_X = -\frac{\partial w}{\partial x}$, $U_Y = \frac{\partial w}{\partial y}$, U_Z	4, 5, 6	rotation angles about the X, Y, Z axes
$A_X = \Psi_{11}$, $A_Y = \Psi_{12}$, A_Z	7, 8, 9	analogues of the linear displacements along the X, Y, Z axes, which model the shear in the direction of the respective axis in the first-approximation model
$C_X = \Psi_{21}$, $C_Y = \Psi_{22}$, C_Z	13, 14, 15	analogues of the linear displacements along the X, Y, Z axes, which model the shear in the direction of the respective axis in the second-approximation model

The first six degrees of freedom are the main ones and correspond to the membrane and bending deformations, and their analogues correspond to the transverse shear deformations.

The rotation angle U_Z and the analogues of the linear displacement A_Z and C_Z in the local coordinate system of the element are equal to zero. They appear when the elements not lying in the same plane are connected and are required to take into account the spatial behavior of the structure.

The considered elements can be used only with the index of the model type 9.

Forces in all multilayer elements are calculated in their local coordinate system. The whole package of layers is considered in each point, and N_X, N_Y, N_Z, T_{XY}, T_{XZ}, T_{YZ} will be calculated both on the upper and lower borders of each layer (see Sec. 13.6). In addition, the horizontal displacements (U_G, V_G) and the deflection (W_G) can be calculated. In order to be able to determine the stresses on the midsurface of the shell, this surface has to be a boundary between the two layers. It can always be done by dividing the central layer into additional ones with the same physical and mechanical characteristics.

Peculiarities of Specifying the Initial Information for Multilayer Finite Elements

The description of each k-th layer of the package includes the following data depending on the properties of its material:

$E^{(k)}(T/m^2)$, $\nu^{(k)}$ — for the isotropic material;

$E^{(k)}(T/m^2)$, $\nu_{13}^{(k)}$, $\nu_{31}^{(k)}$, $G_{13}^{(k)}(T/m^2)$ — for the transverse isotropic material;

$E_1^{(k)}(T/m^2)$, $E_2^{(k)}(T/m^2)$, $E_3^{(k)}(T/m^2)$, $\nu_{12}^{(k)}$, $\nu_{21}^{(k)}$, $\nu_{13}^{(k)}$, $\nu_{31}^{(k)}$, $\nu_{23}^{(k)}$, $\nu_{32}^{(k)}$, $G_{12}^{(k)}(T/m^2)$, $G_{13}^{(k)}(T/m^2)$, $G_{23}^{(k)}(T/m^2)$ — for the orthotropic material.

The thickness of the layer $\delta^{(k)}$ is always indicated.

If there is the elastic subgrade, the subsoil parameter $C_1(T/m^3)$ has to be specified.

If the elements are homogeneous throughout their thickness, the stiffness properties can be specified as the rigidities of ordinary plates and shells described in Section 3.4. It is necessary to specify only the properties implemented in the multilayer elements, where, for example, the subsoil parameter $C_2(T/m)$ of the Pasternak model of the elastic subgrade is not taken into account.

For all finite elements the stresses (forces) are calculated both in the center of the elements and in their nodes. The control of their output is described in the Chapters 8 and 13.

Local Loads

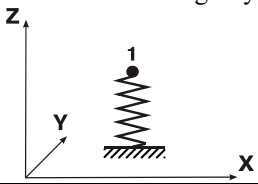
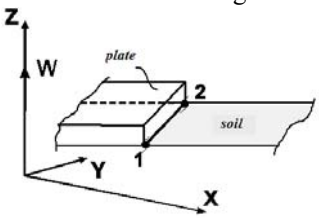
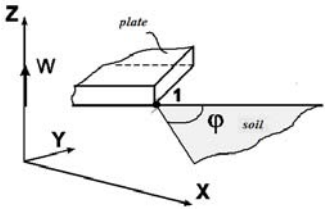
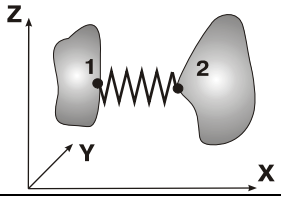
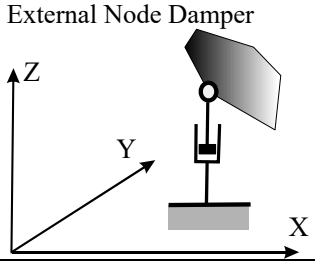
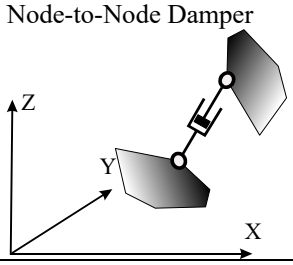
All types of loads (except for the thermal ones) can be specified both in the local and in the global coordinate systems with the snaps in the local and in the global coordinate systems for the multilayer FEs similarly to the thin shell elements (see Sec. 3.4).

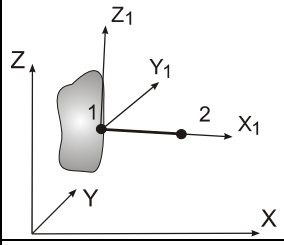
3.8. Special Finite Elements

These elements enable to restrain the linear and angular displacements in the directions of the axes of the local coordinate system (it does not coincide with the global coordinate system in the general case), introduce the constraints of finite rigidity in the directions of the axes of the global coordinate system, take into account the compliance of the material between the adjacent nodes (compliance of the mortar or metal

packing plates between the elements) etc. Table 3.8-1 provides the list of such elements, their description is given below.

Table 3.8-1

FE type	FE name	Index of the model type	Position in space	Degrees of freedom	Comments
51	Constraint of finite rigidity 	any	any	any	It is used to account for the compliance of the bearing constraint
53	Two-node peripheral finite element of elastic subgrade 	3, 5, 8, 9	in the horizontal plane	Z	It accounts for the effect of the strip of soil beyond the structure
54	Single-node peripheral finite element of elastic subgrade 	3, 5, 8, 9	any	Z	It accounts for the shear effect of the angular area of soil beyond the structure
55	Elastic constraint between nodes 	any	any	any	It accounts for the compliance of the structural joints
56	External Node Damper 	any	any	X, Y, Z	It is used to model the external dampers responsive to the velocity of the node displacement in the directions of the degrees of freedom in the global coordinate system.
57	Node-to-Node Damper 	any	any	X, Y, Z	It is used for taking into account the damping arising at the non-zero velocity of the relative motion of the adjacent nodes

FE type	FE name	Index of the model type	Position in space	Degrees of freedom	Comments
154	Null element 	any	any	any	It is used for the calculation for the specified displacements, and also for imposing constraints not parallel to the global coordinate system
200	Pseudo-element	any	any		It is used for deleting the elements from the design model without disrupting the regularity of its coding (it is used only in the text format); any local load specified for the element is ignored

General Type "Null Element" (FE Type – 154)

This FE is used to restrain the linear or angular displacements in the directions of the axes of the local coordinate system of this element, and also to perform the calculation for the specified displacements, the directions of which do not coincide with the directions of the axes of the global coordinate system.

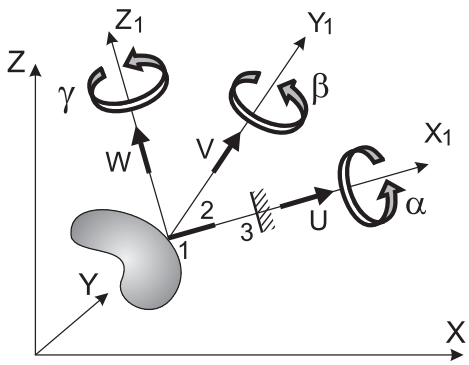


Figure 3.8-1.

Technically, a "Null element" is a three-node finite element [41] that consists of two bars of equal length which lie on one line and the rigidities of which in the direction of the restrained degree of freedom (in the local coordinate system) are specified by the finite values of opposite signs and are comparable to those of real structural members (to ensure the computational stability of the calculation). However, from the user's perspective, a "null element" is a two-node element with the nodes 1 and 2, where the node 1 belongs to the considered structure while the node 2 is free. The node 3 is used only when generating a stiffness matrix, and it is not explicitly specified.

You should keep in mind that technically the node 2 is restrained only by the springs lying on one line with positive and negative values of the "axial" stiffness, therefore it could have displacements in the plane orthogonal to this line, which would lead to an unstable system. Hence, not only the "axial", but also the shear stiffness values have to be specified when specifying the information on this element.

A schematic drawing of the FE is shown in Fig. 3.8-1. The local coordinate system is formed in a similar way as that for a spatial bar. The element can be used for any index of the model type.

When describing the element it is necessary to specify the directions in which the displacements (rotations) are restrained together with an arbitrary positive number for each direction which will be interpreted as the rigidity of the element. Theoretically, the result of the calculation should not depend on the value of this number. However, as noted above, it is selected in such a way so as to ensure the computational stability.

When performing the calculation for the specified displacements the value of the displacement for each direction is specified as the local load with respect to the axes of the local coordinate system in accordance with the used index of the model type. The linear displacements are specified in meters, and the angular ones – in radians in the basic system of units.

Forced displacements can be introduced only in the directions for which the nonzero rigidities (displacement of the constraint) are specified. The calculated forces in a "null element" are the reactions in the imposed constraints or the reactions from the forced displacement of the first node of the element in the

respective directions. Obviously, only the reactions in the directions of constraints specified using the "null element" will be non-zero. The forces are calculated for the section at the beginning of the element (its first node).

FE Modeling the Constraints of Finite Rigidity (FE Type – 51)

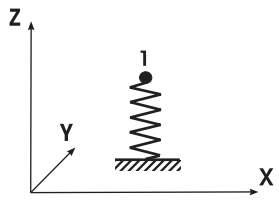


Figure 3.8-2.

This FE (Fig. 3.8-2) is used to impose the constraints of finite rigidity (springs) in the directions of the degrees of freedom in the global coordinate system and can be used, for example, to model the elastic subgrade. The constraints of finite rigidity are specified only in the directions of the axes of the global coordinate system. It makes no sense to impose a constraint of finite rigidity in a node with a rigid constraint in the respective direction. This element can be used with any index of the model type.

The number and order of the stiffness properties completely correspond to the used index of the model type. Non-zero values which can be interpreted in the following way are specified for the directions for which the constraints of finite rigidity are imposed in the node:

- for the displacements X, Y, Z — it is a force causing a unit displacement of the node in the given direction. When modeling the elastic "Winkler" subgrade in the form of elastic nodal supports the given stiffness can be specified by multiplying the subsoil parameter C_1(T/m^3) by the area of the surface which interacts with the node. The stiffness is specified in T/m in the basic system of units;
- for the rotation angles U_X, U_Y, U_Z — it is a moment causing unit rotations in the given direction. The stiffness is specified in Tm/rad in the basic system of units.

Forces (reactions in springs) for the degrees of freedom in the direction of which the constraints of finite rigidity are imposed are obtained in the result of the calculation.

Two-Node Peripheral FE of Elastic Subgrade (FE Type – 53)

This FE is used for modeling the pressure of the strip of soil beyond the plate and perpendicular to its contour (due to the shear resistance of soil). These elements must be used when the plates on the elastic subgrade are modeled with the help of a bi-parametric model of the elastic subgrade (Pasternak model), because otherwise there is a contradiction between the used model which allows for the distribution capacity of the soil and the fact that the behavior of the subsoil beyond the plate is not taken into account.

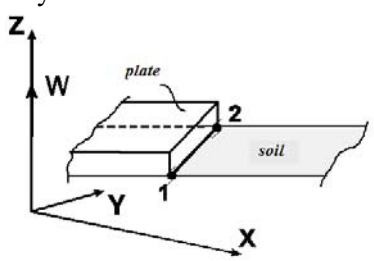


Figure 3.8-3.

A schematic drawing of the FE is shown in Fig. 3.8-3. The values of the subsoil parameters C_1 (T/m^3) and C_2 (T/m) are specified. Each node of the FE has one degree of freedom w — vertical displacement defined in the global coordinate system.

Nodal reactions R_Z are obtained in the result of the calculation.

Single-Node Peripheral FE of Elastic Subgrade (FE Type – 54)

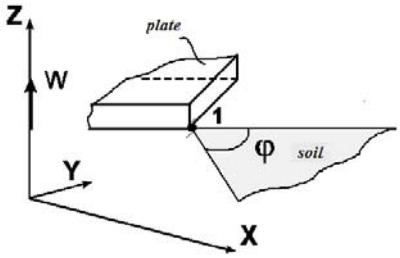


Figure 3.8-4.

This element is used for modeling the pressure of the angular area of soil adjacent to the plate (due to the shear resistance of soil). A schematic drawing of the FE is shown in Fig. 3.8-4. The element consists of one node with one degree of freedom w, — vertical displacement defined in the global coordinate system. The value of the subsoil parameter C_2(T/m) and angle φ are specified. The nodal reaction R_Z is obtained in the result of the calculation.

FE Modeling an Elastic Constraint between the Nodes (FE Type – 55)

This FE enables to take into account the compliance of the material between the adjacent nodes, for example, compliance of a grade beam or metal packing plates between the elements. This element can be used with any index of the model type and it enables to model both the linear and angular compliance of the constraint for any direction. Nodes between which the compliance of the constraint is modeled can have the same coordinates.

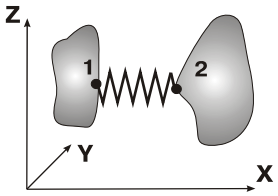


Figure 3.8-5.

The stiffness properties of these elements the number and order of which completely correspond to the used index of the model type can be interpreted in the following way:

- for the displacements X, Y, Z — it is a force (T/m, compressive stiffness), causing a mutual unit displacement of the nodes 1 and 2 in the given direction;
- for the rotation angles U_X, U_Y, U_Z — it is a moment (Tm/rad, shear stiffness), causing mutual unit rotations in the given direction.

This FE models an elastic constraint between the nodes in the global coordinate system by default. The element can be also considered in the local coordinate system. In order to do this you just have to specify the value of the rotation angle of the structural axes using any method described for the universal bar.

Local loads can not be applied within the area of the finite element.

Forces (reactions) for the degrees of freedom in the direction of which the constraints of finite rigidity are imposed are obtained in the result of the calculation.

FE Modeling the Behavior of an External Node Damper (FE Type – 56)

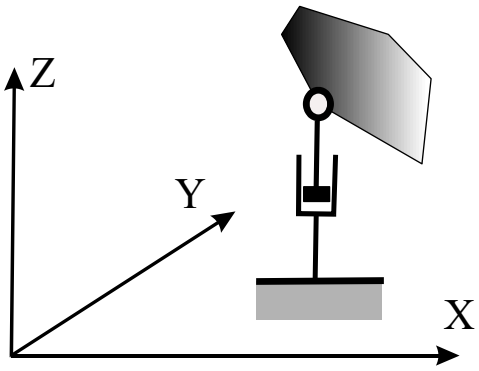


Figure 3.8-6.

This FE (Fig. 3.8-6) is used to model the external dampers responsive to the velocity of the node displacement in the directions of the degrees of freedom in the global coordinate system.

It is assumed that the damping is viscous, i.e. the motion resistance force is proportional to the respective velocity component. Viscous damping ratios are specified for each nodal displacement (rotation) independently and do not affect each other.

The number and the order of stiffness properties completely correspond to the used index of the model type. If the non-zero viscosity coefficients are specified in the directions in which the dampers are introduced in the node, they can be interpreted as follows:

- for displacements X, Y, Z — it is a reaction force arising at a unit velocity of the node displacement in the considered direction. It is measured in T/(m/s) in the basic system of units.

FE-56 can be used for the dynamic calculations only in the **Direct Integration of the Equations of Motion** mode**.** Other modes do not respond to its presence in the design model.

FE Modeling the Behavior of a Node-to-Node Damper (FE Type – 57)

This FE enables to take into account the damping arising at the non-zero velocity of the relative motion of the adjacent nodes.

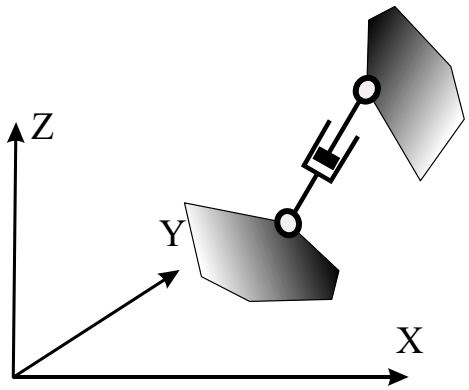


Figure 3.8-7.

The element can be used at any index of the model type. The stiffness properties of these elements, the number and the order of which completely correspond to the used index of the model type can be interpreted as follows:

- for displacements X, Y, Z — it is a reaction force arising at a unit velocity of the relative motion (for example, approach or separation)

It is measured in T/(m/s) in the basic system of units.

It is assumed that the damping is viscous, i.e. the motion resistance force is proportional to the respective velocity component. Viscous damping ratios are specified for each nodal displacement (rotation) independently and do not affect each other.

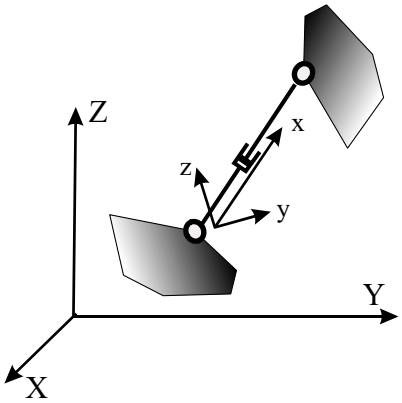


Figure 3.8-8

The mutual velocity can be specified both in the local and in the global coordinate system **(so far the global coordinate system case has been implemented).** The local coordinate system x,y,z is generated in the same way as for a bar located between a pair of the considered nodes (Fig. 3.8-8).

When the global coordinate system is used, the projection of the mutual velocity on the respective coordinate axes is considered (Fig. 3.8-9), as if the damper is installed between the start node and the intermediate point which implements the specified direction and has the same velocities as the end node.

FE-57 can be used for the dynamic calculations only in the **Direct Integration of the Equations of Motion** mode**.** Other modes do not respond to its presence in the design model.

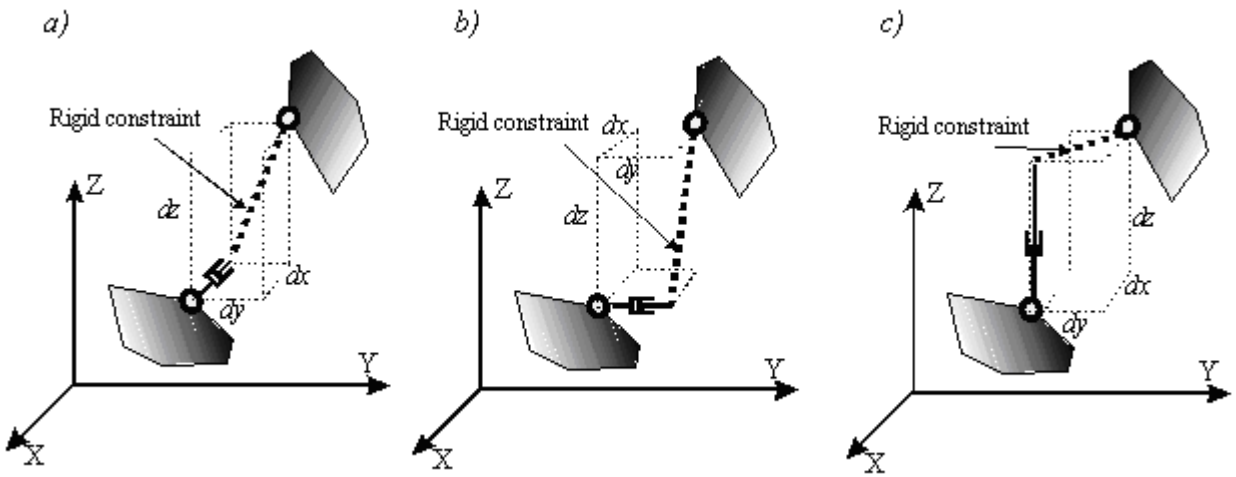


Figure 3.8-9.

Rigid Body (FE Type – 100)

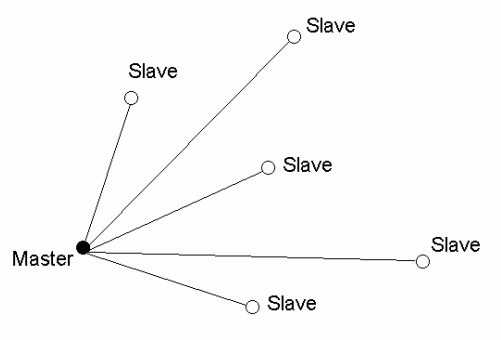


Figure 3.8-10.

The element has an arbitrary number of nodes, which are connected to each other as in the case of a rigid body (see Sec. 2.2).

A schematic drawing of the rigid body is shown in Fig. 3.8-10. The constraints of the nodes are specified depending on the number of the degrees of freedom: 1 – the nodes are restrained in the given direction (rotation), 0 – they are not restrained.

For a detailed description of this finite element, see [86].

Pseudo-Element Designed for Modeling Elements Excluded from the Design Model (FE Type – 200)

This FE is used to model the elements excluded from the design model without changing the numbering of the remaining elements. It is used only when the initial data are specified in the text format.

References to Chapter 3

1. S.A. Ambartsumyan, Theory of Anisotropic Plates (Strength, Stability and Oscillations). – M.: Nauka, 1987. – 360p.
2. V.A. Bazhenov, A.V. Perelmuter, O.V. Shishov, Structural Mechanics. Computer Technology and Modeling. – M.: SCAD Soft, 2014. – 911 p.
3. K.-J. Bathe, Finite Element Procedures. — Prentice Hall, Upper Saddle River, New Jercey, 1996. —1037 p.
4. K. Bathe, E. Wilson, Numerical Methods in Finite Element Analysis. – M.: Stroyizdat, 1982. – 448 p.
5. A.E. Belkin, S.S. Gavriushin, Finite Element Analysis of Plates. – Moscow, Bauman MSTU Publ., 2008. – 232 p.
6. K. Washizu, Variational Methods in the Theory of Elasticity and Plasticity. — M.: Mir, 1987. — 544 p.
7. V.E. Verizhenko, V.S. Karpilovsky, V.G. Piskunov et al., Finite Element Analysis of Heterogeneous Shallow Shells and Plates. — Kyiv, “Higher School”, — 1987. — 200 p.
8. V.Z. Vlasov, N.N. Leontiev, — Beams, Plates and Shells on the Elastic Subgrade. — M.: GIPML, 1960. — 492 p.
9. Richard H. Gallagher, Finite Element Analysis: Fundamentals. – M.: Mir, 1984. – 428 p.
10. L.G. Donnell, –- Beams, Plates and Shells. — M.: Nauka. Home edition physical and mathematical literature, 1982. — 568 p.
11. I.D. Evzerov, Error Estimates for Inconsistent Plate Finite Elements. — Deposited at Ukrainian Res. Inst. for Sci.-Tech. Information, No.1467-79. – Kyiv, 1979. – p.9.
12. I.D. Evzerov, Error Estimates for Displacements When Using Inconsistent Finite Elements. // Numerical Methods for Continuum Mechanics, Novosibirsk, 1983, Volume 14, No.5. – pp. 24–31.
13. I.D. Evzerov, FEM convergence in the case of basis functions that do not belong to the energy space. // Calculations with sparse matrices. – Novosibirsk: VTsSO AN USSR. 1981. – pp. 54–61.
14. I.D. Evzerov, V.S. Zdorenko, Convergence of Flat Thin Shell Finite Elements. // Structural Mechanics and Analysis of Structures. 1984, No1. — pp. 35-40.
15. O. Zenkevich, Finite Element Method. — M.: Mir, 1975. — 541 p.
16. O. Zenkevich, K. Morgan, Finite Elements and Approximations. — M.: Mir, 1986. — 318 p.
17. O. Zenkevich and I. Chang, Finite Element Method in the Theory of Structures and in Continuum Mechanics. – M.: Nedra,1974. – 238 p.
18. V.S. Karpilovsky, Study and Design of Some Types of Finite Elements for the Structural Mechanics Problems. PhD diss. 01.02.03 – “Structural Mechanics”. – Kyiv: National transport university (KADI), 1982. – 179 p.
19. V.S. Karpilovsky, V.I. Kudashov, D.N. Tsvetkov, Library of Isoparametric Finite Elements in “LIRA” // Proceedings of the Universities. Construction and Architecture. 1987, No.7. — pp. 28-32.
20. V.S. Karpilovsky, I.V. Loza, Inconsistent Quadrangular Deep Beam Finite Element. Deposited at Ukrainian Res. Inst. for Sci.-Tech. Information, №2446UK–86, 1985. – 24 p.
21. V.S. Karpilovsky, New consistent quadrangular deep beam finite element with rotational degrees of freedom. // Proceedings of the II International Scientific and Practical Conference “Modern Methods and Problem-Oriented Complexes for Structural Analysis and Their Application in Design and Educational Process” (Kyiv, 26–27 September 2018). Kyiv: 2018. – pp.57–59.

22. V.S. Karpilovsky, Construction of Inconsistent Finite Elements. Deposited at Ukrainian Res. Inst. for Sci.-Tech. Information, 1980, No.2153, — 49 p. URL: https://scadsoft.com/download/Karpil1980.pdf
23. V.S. Karpilovsky, Finite Element Method and Problems of the Theory of Elasticity. – Kyiv: Sophia-A, 2022. – 276 p. URL: https://scadsoft.com/download/KVSBook2022.pdf
24. V.S. Karpilovsky, Triangular Six-Node Plate Finite Element. // Proceedings of the Universities. Construction and Architecture, 1989, No.6. — pp. 35-39.
25. V.S. Karpilovsky, Quadrangular Eight-Node Plate Finite Element // Structural Mechanics and Analysis of Structures, 1990. — pp. 13-17.
26. V.S. Karpilovsky, Quadrangular Finite Elements for Solving the Plane Problem of the Theory of Elasticity. // Systems for Computer-Aided Design of Structures, — Kyiv: Budivelnik, 1991. —pp.35-43.
27. V.A. Kiselev, Structural Mechanics. Special Course. — M.: Stroyizdat, 1980. – 616 p.
28. R.W. Clough. The Finite Element Method in Plane Stress Analysis. – In the collection of works: Computers in the Design of Building Structures/ed. A.F. Smirnov. – M.: Stroyizdat, 1967. – pp. 142–170.
29. Y.Z. Klempert, V.I. Parikov, V.I. Slivker, Procedure for Calculating the Stiffness Matrix of a Prismatic Bar. // Analysis of Spatial Structures. Issue 16. —M.: Stroyizdat, 1974, pp. 179-189.
30. V.V. Lalin, A.V. Yavarov, E.S. Orlova, A.R. Gulov, Application of the Finite Element Method for the Solution of Stability Problems of the Timoshenko Beam with Exact Shape Functions. // Hydraulic Engineering. 2019, №6. – p.45–52. URL: http://www.gts.energy–journals.ru/index.php/GTS/article/view/655
31. L.D. Landau, E.M. Lifshitz, Theory of Elasticity. – M.: Institute of Physical Problems, 2003. – 264 p.
32. S.G. Lekhnitsky, Theory of Elasticity of an Anisotropic Body. — M.: Nauka, 1977.— 416 p.
33. A.I. Lurie, Nonlinear Theory of Elasticity. – M.: Nauka, 1980. – 512 p.
34. A.I. Lurie, Spatial Problems of the Theory of Elasticity. – M.: GITTL, 1955. – 491 p.
35. L.A. Lusternik, V.J. Sobolev, Elements of Functional Analysis. – M.: Nauka, 1965. – 520 p.
36. A. Lyav, Mathematical Theory of Elasticity. – M.: 1935. – 674 p.
37. S.G. Mikhlin, Variational Methods in Mathematical Physics. – M.: Nauka, 1970. – 512 p.
38. Yu.I. Nemchinov, Design of Spatial Structures (Finite Element Method). – Kyiv: Budivelnik, 1980. – 232 p.
39. V. Novatsky, Theory of Elasticity. – M.: Mir, 1975. – 882 p.
40. V.Z. Parton, P.I. Perlin, Mathematical Methods of the Theory of Elasticity. – M. Nauka, 1981. – 689 p.
41. A.V. Perelmuter, V.I. Slivker, Peculiarities of the Algorithmic Implementation of the Slope-Deflection Method with Additional Constraints Taken into Account.// Finite Element Method *and* Structural Mechanics. Proceedings of Leningrad Polytechnic Institute, Issue 349. — L. 1976. — pp.28-36.
42. A.V. Perelmuter, V.I. Slivker, Design Models of Structures and a Possibility of Their Analysis. – M.: DMK Press, 2007. – 596 p.
43. A.V. Perelmuter, V.I. Slivker, Stability of Equilibrium of Structures and Related Problems. – M.: SCAD Soft, 2010. – Volume 1 – 704 p., Volume 2 – 672 p., Volume 3 – 388 p.
44. V.G. Piskunov, V.S. Karpilovsky et al., Finite Element Analysis of Crane Structures. —M.: Mashinostroyeniye, 1991. — 240 p.
45. A.N. Podgorny, G.A. Marchenko, V.I. Pustynnikov, Fundamentals and Methods of Applied Theory of Elasticity. – Kyiv: "Higher School", 1981. – 328 p.
46. V.A. Postnov, I.Ya. Kharkhurim, Finite Element Method in the Analysis of Ship Structures. —L. Sudostroyeniye, 1974. — 342 p.
47. I.I. Pogorelov, Structural Mechanics of Thin-Walled Structures. – St. Petersburg: BHV-Petersburg, 2007. – 518 p.
48. Yu. N. Rabotnov, Solid Mechanics. – M.: Nauka, 1979. – 744 p.
49. Rasskazov A.O., Karpilovsky V.S., Kharchenko N.G., Finite Element of a Laminated Orthotropic Shallow Shell. Deposited at Ukrainian Res. Inst. for Sci.-Tech. Information, 223Uk-D84, 1984.
50. A.O. Rasskazov, I.I. Sokolovskaya, N.A. Shulga, Theory and Analysis of Laminated Orthotropic Plates and Shells. — Kyiv: Higher School, 1986, — 192 p.
51. K. Rectoris, Variational Methods in Mathematical Physics and Technology. – M.: Mir, 1985. – 590 p.
52. A.R. Rzhanitsyn, Structural Mechanics. — M.: Higher School, 1982. —400 p.
53. L.A. Rozin, Problems of the Theory of Elasticity and Numerical Methods of Their Solution. – St. Petersburg: St. Petersburg State Technical University Publishing House, 1998. – 530 p.
54. L.A. Rozin, Variational Formulations of Problems for Elastic Systems. – St. Petersburg: St. Petersburg State Technical University Publishing House, 1978. – 222 p.
55. L.A. Rozin, Bar Systems Treated as Systems of Finite Elements. — L.: LSU, 1975. – 237 p.
56. F.M. Svoisky, Boundary Conditions for Finite Elements with Rotational Degrees of Freedom. – St. Petersburg: NOU Express, 2005. – 106 p.
57. V.I. Slivker, Variational Principles of Structural Mechanics. – M.: ASV, 2005. – 708 p.
58. SP 20.13330.2016 Loads and Actions. Updated edition of SNiP 2.01.07-85* (with Modifications N 1, 2, 3) / Code of Practice from 03.12.2016.

59. SNiP 2.02.01-83*. Foundations of Structures. — M.: Stroyizdat, 1984.
60. K.V. Solyanik-Krassa, Axisymmetric Problem of the Theory of Elasticity. — M.: Stroyizdat, 1987. —338 p.
61. G. Strang, G. Fix, Analysis of the Finite Element Method. — M.: Mir, 1977. —350 p.
62. P.G. Ciarlet, The Finite Element Method for Elliptic Problems. – M.: Mir, 1980. – 512 p.
63. Ter-Martirosyan, Soil Mechanics, Moscow. – M.: ASV, 2009. – 552 p.
64. S.P. Timoshenko, J. N. Goodier, Theory of Elasticity. – M.: Nauka, 1975. – 576 p.
65. S.P. Timoshenko, S. Woinowsky-Krieger, Plates and Shells. – M.: Nauka, 1966. – 636 p.
66. S.P. Timoshenko, Course of the Theory of Elasticity. — Kyiv: Naukova Dumka, 1972. —508 p.
67. S.P. Timoshenko, Plates and Shells. – M.: Gostehizdat, 1948. – 388 p.
68. S.P. Timoshenko, Stability of Bars, Plates and Shells. – M.: Nauka, 1971. – 880 p.
69. Adini A., Clough R.W. Analysis of plate bending bythe finite element Method. – NSF Report, G7337, 1961.
70. Allman D J. A consistent triangular element including vertex rotations forplane elasticity analysis // Computers and structures, 1984, Vol. 19, pp. l–8. URL: https://doi.org/10.1016/0045-7949(84)90197-4
71. Allman D.J. A quadrilateral finite element including vertex rotations for plane elasticity analysis // International journal for numerical methods in engineering, 1988, Vol. 26, pp. 717–730. URL: https://doi.org/10.1002/nme.1620260314
72. Allman D.J. Evaluation of the constant strain triangle with drilling rotations // International journal for numerical methods in engineering. 1988. V.26. P. 2645–2655. DOI: 10.1002/NME.1620261205
73. Baseley G.P., Cheung Y.K., Irons B.M., Zienkiewicz O.C. Triangular elements in plate bending. – Conforming and noncorforming solutions//Proc. Conf. on Matrix Methods in Structural Mechanics (Nov. 1966), pp. 547–576, Wright Patterson AFB, Oct. 1965
74. Bathe K.J. Finite Element Procedures. New Jersey, Prentice Hall, 1996, 1038p.
75. Bathe K.J. Dvorkin, E.N. A four–node plate bending element based on Mindlin/Reissner plate theory and a mixed interpolation. //Int. J. Num. Meth. Engrg., 1985, Vol. 21, pp. 367–383. URL: https://doi.org/10.1002/nme.1620210213
76. Bletzinger K.U, Bischoff M, Ramm E: A unified approach for shear–locking–free triangular and rectangular shell finite elements. // Computers & Structures, 2000, Vol. 75, pp. 321–334. DOI:10.1016/S0045–7949(99)00140–6
77. Bogner F.K., Fox R.L., Schmit L.A. The Generation of Interelement Consistent Stiffness and Mass Matrices by the Use of Interpolation Formulas// Proceedings of the Conference on Matrix Methods in Structural Mechanics, Wright–Patterson Air Force Base, Ohio, October 1965, pp. 397–444.
78. Chapelle D., Bathe K.J. The Finite Element Analysis of Shells Funamentals. — Springer–Verlag Berlin Ytidelberg New York, 2003. —330p.
79. Cazzani A, Atluri S.N. Four–nodes mixed finite elements, using unsymmetric stressed, for linear analysis of membranes// Computational Mechanics, 1993, 11, pp.229–251
80. Chen X.M., Cen S., Sun J.Y., Li Y.G. Four–Node Generalized Conforming Membrane Elements with Drilling DOFs Using Quadrilateral Area Coordinate Methods. //Mathematical Problems in Engineering, 2015, Vol. 3–4, pp. 1–13. DOI: 10.1155/2015/328612
81. Choo Y.S., Choi N., Lee B.C. Choo Y.S., Choi N., Lee B.C. Quadrilateral and triangular 22plate elements with rota–tional degrees of freedom based on the hybrid Trefftz method. // Finite Elements in Analysis and Design, 2006, Vol. 42(11), pp. 1002–1008. DOI: 10.1016/j.finel.2006.03.006
82. Clough R.M., Tocher J. Finite elements stiffness matrices for the analysis of plate bending// Proc. Conf. Matrix Meth. Struct. Mech. – Ohio, 1965
83. Cook R. A plane hybrid element with rotational d.o.f. and adjustable stiffness. // International Journal for Numerical Methods in Engineering, 1987, Vol. 24(8), pp. 1499–1508. DOI: 10.1002/nme.1620240807
84. Cook R.D. On the Allman triangle and a related quadrilateral element // Computeis andstructures. 1986, Vol. 22(6), pp. 1065–1067. DOI: 10.1016/0045–7949(86)90167–7
85. Fellipa C.A. A study of optimal membrane triangles with drilling freedoms. // Computer Methods in Applied Mechanics and Engineering, 2003, Vol. 192(16–18), pp. 2125–2168. DOI: 10.1016/S0045–7825(03)00253–6
86. Fialko S.Yu. Application of rigid links in structural design models. / International Journal for Computational Civil and Structural Engineering , 2017, 13 (3), 119-137.
87. Fraeijs de Veubeke B. A conforming finite element for plate bending. // International Journal of Solids and Structures, 1968, Vol. 4, pp. 95–108. DOI:10.1016/0020-7683(68)90035-8
88. Irons B.M., Razzaque A. Experience with the path test. // In: The Matematical Foundations of the Finite Element Method with Application to Partial Differential Equations, Academic Press, 1972, pp. 557–587.
89. Iura M. Atluri S.N. Formalation of a membrane finite element with drilling degrees of freedom// Computational Mechanics, 9, 1992, pp.417–428. DOI:10.1007/BF00364007
90. Karpilovskyi V.S. Finite Elements of the Plane Problem of the Theory of Elasticity with Drilling Degrees of Freedom. // International Journal for Computational Civil and Structural Engineering, 2020, Vol. 16(1), pp. 48–72. URL: https://doi.org/10.22337/2587-9618-2020-16-1-48-72

91. Karpilovskyi V.S. Finite Elements for the Analysis of Reissner–Mindlin Plates With Joint Interpolation of Displacements and Rotations (JIDR). // International Journal for Computational Civil and Structural Engineering, 2021, Vol. 17(3), pp. 49–63, URL: https://doi.org/10.22337/2587-9618-2021-17-3-48-62
92. Luo Y., An Efficient 3D Timoshenko Beam Element with Consistent Shape Functions // Adv. Theor. Appl. Mech. 2008. Vol. 1(3). pp. 95–106.
93. Macneal R.H., Harder, R.L. A proposed standard set of problems to test finite element accuracy. // Finite elements in analysis and design, 1985, Vol. 1, pp. 3–20. DOI: 10.1016/0168–874X(85)90003–4
94. MacNeal R.H., Harder R.L. A proposed standart set of problems to test finite element accuracy. Finite Element Anal. Des., 1987, Vol. 1, pp.1793-1799. DOI:10.1016/016-874x(85)90003-4
95. MacNeal R.H., Harder R.L. A refined four-noded membrane element with rotational degrees of freedom // Computers and structures. 1988. V.28. JSbl. pp.75-84. URL:https://doi.org/10.1016/0045-7949(88)90094-6
96. MacNeal R.H., Harder R.L. A refined four–noded membrane element with rotational degrees of freedom. // Computers and structures, 1988, Vol. 28(1), pp. 75–84. DOI:10.1016/0045–7949(88)90094–6
97. Makiett R.N., Marcal P.V. Finite Element Analysis of Nonlinear Structures // Journal Structural Division. – Proc./ASCE, 1968, Vol. 94(9), pp.2081–2105. URL:https://doi.org/10.1061/JSDEAG.0002066
98. Oden J.T. Finite Elements of Nonlinear Continua.– New York: McGraw Hill Book Company, 1972. – 432p.
99. Oden J.T. Finite Elements of Nonlinear Continua. – Dover Publications, 2006. – 456p.
100. Robinson J: An evaluation of lower order membranes as contained in the MSC/NASTRAN, ASAS and PAFEC systems. Report to the Royal Aircraft Establishment, Farnborough (MOD Contract No. A93b/494), September 1979
101. Wilson E.L, Ibrahimbegovic A. Thick shell and solid elements with independent rotation fields //International journal for numerical methods in engineering, 1991, Vol. 31(7), pp. 1393–1414. DOI: 10.1002/nme.1620310711
102. White D W, Abel J F 1989 Testing of shell finite element accuracy and robustness. Finite Elements // Finite Elements in Analysis and Design, 1989, Vol.6(2), pp. 129-151. URL: https://doi.org/10.1016/0168-874X(89)90040-1
103. Xiao–Ming Chen,Song Cen, Jian–Yun Sun,Yun–Gui L. Four–Node Generalized Conforming Membrane Elements with Drilling DOFs Using Quadrilateral Area Coordinate Methods //Mathematical Problems in Engineering, Vol. 2015, Article ID 328612, 13p. URL:https://doi.org/10.1155/2015/328612
104. Yunus M., Saigal S., Cook R.D. On improved hybrid finite elements with rotational degrees of freedom. //Interntional journal for numerical methods in ingineering, 1989, Vol. 28, pp.785–800. DOI: 10.1002/nme.1620280405
105. Zhang H., Kuong J.S. Eight–node membrane element with drilling degrees of freedom for analysis of in–plane stiffness of thick floor plates// International journal for numerical methods in engineering, 2008, Vol. 76(13), pp. 2117–2136. DOI: 10.1002/nme.2395
106. O.C. Zienkiewicz, R.L. Taylor, The Finite Element Method. 3 volumes. — Planta Tree, Butterworth-Heinemann, 2000. (1 vol. —690 p., 2 vol. —459 p., 3 vol. —334 p).

4. Structure CAD for DUMMIES

Beginners… should not feel embarrassed, … when finding themselves in want of introductory knowledge even to read introductory information.

P.HALMOS

Graphical interface of **SCAD** is based on the MDI (Multi Documented Interface) technology. This makes it possible to work simultaneously with multiple tasks, each of which uses one window in the initial data preparation mode (in preprocessor). In the mode of the analysis of the calculation results (in postprocessor), the number of windows for one problem is not limited. They may contain various fragments of the design model, or different types of results. Different tasks can be simultaneously at different processing stages – preparation of data, calculation and analysis of results.

Window of the task that was accessed by the user last is considered to be active. The set and the state of controls of the graphical interface will correspond to the mode in which the task is in the active window.

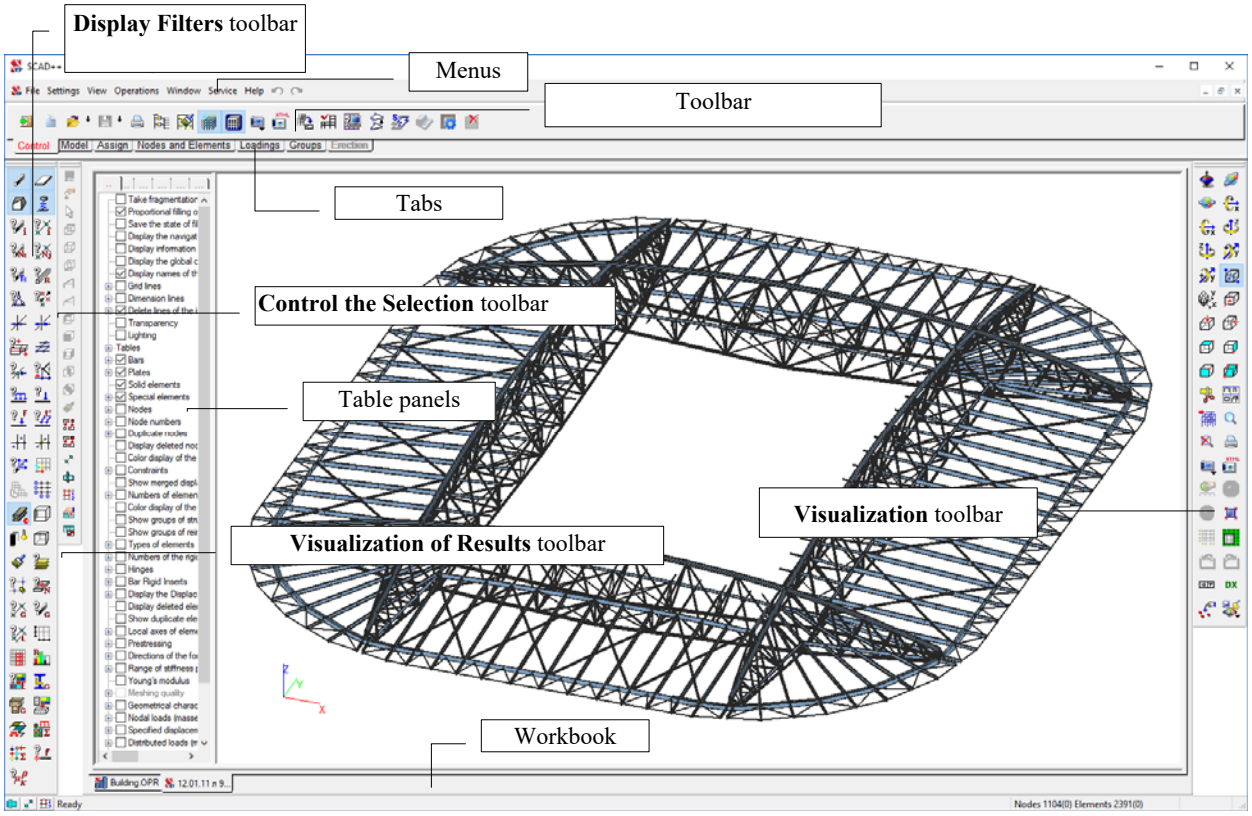


Figure 4-1. *General view of the* **SCAD** *window*

Controls of the graphical interface, data access, operations with windows (the admissibility of the simultaneous presence of several windows, the concept of the active window and activation methods, the ability to move windows, etc.) are implemented in full compliance with the rules adopted for the Windows operating environment.

4.1. Graphical Environment

The graphical user interface includes the following elements (Fig. 4-1):

Menus are located at the upper part of the **SCAD** window under the title, their set depends on the current active mode.

Toolbars are the main element of the graphical interface which is used to invoke operations of the model generation and the analysis of the calculation results. Toolbars consist of the set of buttons and lists. Each toolbar usually includes operations of the same purpose, for example, for working with nodes or elements, analysis of displacements or forces. Switching between toolbars is performed by clicking on "tabs" with their names. Functions and modes of toolbars of pre- and postprocessors are described in the respective sections.

Display filters — filters enable to control the display of the design model. They are used to display almost all parameters of the model, including the numbers of the elements and nodes, loads, boundary conditions, stiffness properties, etc. Moreover, filters are used to select the method of displaying the model (wire, realistic removing lines of the invisible contour, etc.).

Display filters are duplicated in the form of a tree structure in the panel extending along the left side of each window.

Visualization toolbar includes controls for positioning the model on the screen, fragmentation of the model, and also for printing the current image.

Control the Selection toolbar includes a set of buttons used to select a cursor, specify the rules for selecting nodes and elements depending on their position in the model, and also to organize the interaction between the graphical and table representation of the model.

Visualization of Results toolbar appears in the mode of the graphical analysis of the results and is used to control the display of the calculation results on the design model.

Table panels with the initial data and the calculation results are located near the left edge of each window and can be switched using the tabs (table panels are located in the same place with the duplicate **Display Filters** toolbar).

Workbook is a part of the window located along its lower boundary, which contains information on the operations performed in the course of work with the task in the current session, as well as error messages and warnings.

Dialog boxes — in those cases when an operation requires additional data or settings, the respective dialog box is displayed. Dialog boxes contain various controls – tables, buttons, input fields, lists, etc. Data can be entered, deleted or modified using the keyboard. If you exit the dialog box by clicking the **OK** button, all the selected settings are saved. If the **Cancel** button is used to exit the dialog box, the new settings are not saved, and the previously selected or default settings will be used.

Menus

The expression «Select from the menu …» means the following sequence of operations:

- place the cursor over one of the menus and left-click;
- once the list of operations is opened, you then have to select a line with the name of the desired operation by moving the cursor vertically and left-click.

The set and contents of menus depend on the current mode. For example, when the program is invoked from the program group of the **SCAD Office** system, there are five menus — **File**, **Settings**, **View**, **Service** and **Help**. Once the project is loaded, there will be seven menus — **File**, **Settings**, **View**, **Operations**, **Window**, **Service** and **Help**.

File menu (full set) can contain more than twenty items. There are fewer items in some modes. Moreover, some items may be inaccessible. For example, the **Save Project** item will be inaccessible if the project is not loaded or has not been modified.

The full list of sections of the ***File*** menu includes:

- **New** — creates a new project;
- **Open**... — opens the existing project;
- **Read the Project from the Text Format...** — loads the project in the format of a text file of the **SCAD++** initial data;
- **Read the Project from the Text Format (Previous Versions)** — loads the project in the format of a text file of the initial data of **SCAD** version 11.xx etc.;
- **Import** — loads the projects created by other applications in the following formats DXF, DWG, SDNF, IFC etc.;
- **Save Project** — saves the project in the file without changing the file name;

- **Save Project As...** — saves the project in the file with a new name;
- **Save Project for SCAD 11.xx** — saves the project in the file in the format of version 11.xx;
- **Close Project** — closes the current project;
- **Open Workspace** — loads the file with the list of active projects saved earlier;
- **Save Workspace** — saves the list of active projects in the file;
- **Save Workspace As** — saves the file with a list of active projects under a new name;
- **Close Workspace** — finishes the work with all the projects opened when loading the workspace;
- **Save Data As Text** — converts the data from internal formats of the project to the text file in the formats of the input language;
- **Export** — exports data in standard formats or in formats of other applications;
- **Send** — sends the project by email;
- **Preview** — view the current image of the model before printing;
- **Print** — prints the current image of the model;
- **Recent Workspaces** — list of files containing the description of the workspace;
- **Exit** — finishes the work with the program.

The lower lines of the **File** menu contain the file names of the last ten projects the user worked with. In those cases when you have to work with one of these projects, you just have to select it from this list.

The full list of sections of the ***Settings*** menu includes:

- **Immediate Execution** — this option becomes accessible when the operations with the selected nodes and elements are performed and provides their immediate execution without confirmation by the **OK** button;
- **Color Palette Settings** — invokes the multi-tab **Color Settings** dialog box, where you can specify the display color for all attributes and objects of the design model, results of the calculation, etc.;
- **Calculation Parameters** — invokes the **Calculation Parameters** dialog box, where you can select the calculation method and set the accuracy of solution of static and dynamic problems, the control of the solution and the stability analysis, allowable deviations of the shape of finite elements, etc.;
- **Environment Settings** — opens a dialog box where you can specify the graphic environment settings. They include, for example, activating the mode of separate display of plate elements, specifying the distance between the end of the element and the node at the 3D display of profiles of bar elements, “hatching” the distributed load on bar elements, specifying the estimation accuracy of the duplicate nodes, etc. Environment settings are considered in more detail in Section 10.4;
- **Set Reinforcement Spacing** — a list of allowable distances between rebars used in the selection of reinforcement is specified;
- **Setting the Design Codes** — the respective dialog box is used to select the design codes according to which the solution of design problems of elements of steel and reinforced concrete structures will be performed;
- **Settings** — invokes a multi-tab dialog box where you can set the units of measurement of the input and output data for the current problem, interface language, fonts, etc.;
- **Express Setting of the Units of Measurement** — the **Parameters** dialog box is used to set the main units of measurement on the basis of which all the units of input data and of the calculation results are assumed. It should be noted that the units of measurement of both the initial data and the results can change multiple times in the process of the generation of the model and the analysis of the results;
- **Format Pages for Printing** — invokes the **Format Pages for Printing** dialog box where you can specify the design elements of the printed graphic materials;
- **Specify Work Directories** — the **Specify Directories** dialog box invoked by this operation enables to specify the names of catalogues from which the project file should be taken by default and where the working files and the result files should be put. You can change the purpose of catalogues only if the projects are not active, i.e. you have to close the active project before invoking the operation (**Close Project** operation in the ***File*** menu);

- **Print Setup** — standard Windows procedure for selecting the type of printer and specifying the characteristics;
- **Visualization Settings of the Result Tables** — select the program (text editor) for the visualization of text documents including tables of initial data and results.

View menu is used for the control of the display of the model and duplicates a series of operations of the **Visualization** toolbar, including **Rotation**, **Initial position**, **Projections**, and also the selection of the rendering engine (**GDI** or **DirectX**). Moreover, the menu can be also used to enable and disable the display filters, visualization, control the selection and visualization of results toolbars.

Operations menu duplicates the buttons of the active section of the toolbar.

Window menu is typical for multi-window Windows-applications and enables to create a new window (only in postprocessor), and to automatically arrange the windows according to different rules.

Service menu in preprocessor enables to invoke different calculators: standard Windows calculator, formula calculator and measurement units converter.

Help menu contains items which enable to obtain the help information on the whole complex and detailed information on the preparation of the initial data, reading of the results, control of the modes, etc.

Toolbars

Toolbars are one of the main controls of the **SCAD** complex. Activation of the toolbars is performed by pointing on the tabs, the set of which depends on the mode (preparation of data or the analysis of results).

The set of tabs in the data preparation mode depends in the type of the generated model — on the basis of finite elements or an enlarged model on the basis of structural members. The following set of tabs is available when creating a finite element model: **Control**, **Model**, **Assign**, **Nodes and Elements**, **Loadings**, **Groups. Special**. If the model is prepared for the analysis allowing for the erection sequence of the structure, then in addition to the listed tabs the **Erection** tab will be available as well. If you are working with enlarged elements (**FORUM** preprocessor), the following tabs are available: **Control**, **Model**, **Operations with Nodes**, **Operations with Elements**, **Loads**.

When you perform the analysis of results, including the check and the selection of reinforcement in the elements of reinforced concrete structures and of the sections of elements of steel structures, the following tabs are available **Control**, **Displacements**, **Force Diagrams**, **Stress Fields**, **Postprocessors**, **Groups**, **Reinforced Concrete**, **Steel**.

Icons

Toolbars have icons used for the initialization of operations. It is not always possible to make the icon adequate to the sense of the operation it symbolizes due to the limited space of the button. Various means are used to make mastering of the complex and working with it easier. Firstly, it is the information line where a brief description of the operation corresponding to the button is output when the cursor is within the button area. Secondly, it is a system of symbolic images in icons given below, that captures the essence of the operation and is common for the entire complex.

— perform the operation (**OK** button);

— cancel the given operation;

— restore the original model;

— remember the loading;

— operations for working with elements;

— letter “R” — stiffness properties;

— operations for working with nodes;

— sign “plus” — add nodes or elements;

— left arrow — restore the deleted nodes or elements;

— sign “minus” — delete nodes or elements;

— sign “division” — divide elements;

— letter “C” — create a model or a contour;

— “question mark” — display or obtain information.

If the lower right corner of the icon has an arrow pointing to the right, for example, , it means that when you press the button, a group of controls of the respective mode will be opened. If the arrow is pointing down, for example, , a drop-down menu will appear after pressing this button.

Buttons with Additional Features

Some buttons include an additional arrow on the right side. This arrow means that there are several options for performing the operation. Clicking the button will perform the "standard" action, and clicking the arrow will open a menu for selecting alternatives. Let’s take, for example, the **Save** operation – ; clicking the button will save the model under the current name, and clicking the arrow will open a menu where you can select the **Save As** item and save the model under a different name.

Graphs

In those cases when the data are displayed as graphs (see Fig. 4.1-1), special controls are provided, which include buttons for zooming the image and drawing the graph.

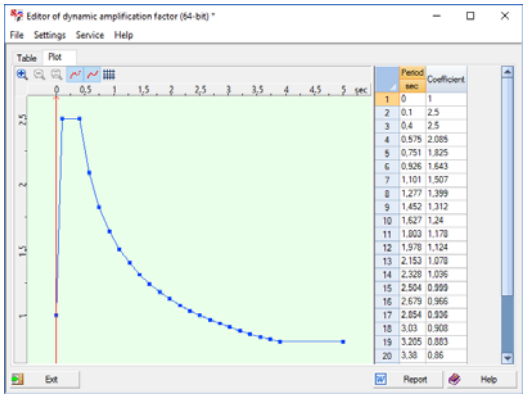


Figure 4.1-1. *Display graphs*

Button with the plus sign zooms in the image. Every click on it doubles the size of the graph. Button with the minus sign is used for the stepwise decrease of the image and works only after the “plus” button. In order to return to the original size of the graph, you just have to click the “equal” button.

Buttons for drawing enable to show or hide the grid, and to display the points on the basis of which the graph was plotted. As a rule, there is a table with the numerical representation of the same data next to the table.

Filters

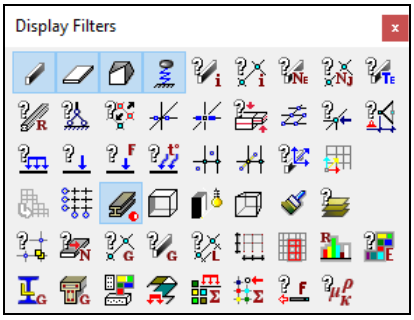
Filters are used to control the display of the design model. Two types of filters are provided in the program — control of the display of information on the design model (further — **display filters**) and control of the display of the actual design model (**visualization filters**). Filters of the first type enable not only to show different characteristics of the model on the design model, but also to specify the type and rules for representation of these characteristics, and also to display objects of the same type according to the specified characteristics. Moreover, the filtration can be performed simultaneously for different properties of the objects. For example, the filter can be used to select elements with the same stiffness properties, lying in the given plane and belonging to a certain group from all the elements.

Display Filters toolbar (Fig. 4.1-2, a) is designed as a special window containing a set of buttons for the selection of display parameters. You can show or hide this window by clicking the respective button—

, in the **Control** section of the toolbar and duplicated in the ***View*** menu (**Toolbars** section). **Display Filters** toolbar can change its sizes and can be located in any part of the screen.

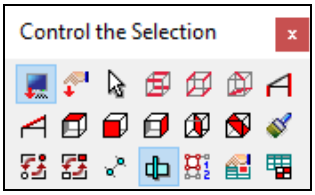
The multi-tab **Set the Information Display Filters** dialog box is used for a more detailed setting of the display filters. It appears after selecting with a cursor a button of the respective filter in the toolbar and clicking the ***right*** mouse button. If the user has changed the setting of a particular filter, the corresponding button will have an additional indicator, which indicates a change in the default settings. For example, the display filter of bar elements (in the standard and modified states) looks as follows: , .

As noted above, the filters are duplicated in the form of a tree structure located along the left edge of the window of the problem. Filter setting is performed with the help of a system of checkboxes (Fig. 4.1-3).

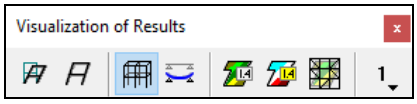


a)

b)



c)



d)

Figure 4.1-2. *Filter toolbars*
a) Display Filters, *b*) Visualization, *c*) Control the Selection, *d*) Visualization of Results

Controls of the **Visualization** toolbar enable to manipulate the image of the model. These operations include rotation and fragmentation, projection on the plane of the principal coordinate system and the selection of the technology of representation of the graphical information on the screen (raster or vector).

In terms of the size and location the **Visualization** toolbar (Fig. 4.1-2, b) has the same properties as the **Display Filters** toolbar. You can show or hide it by clicking the button .

Control the Selection toolbar (Fig. 4.1-2, c) is used to control the selection of nodes or elements in the process of working with the model. In addition to the buttons for specification of the type of cursor, it includes elements that provide interaction between the graphical and tabular information, as well as a series of buttons that can be considered as filters which enable to select objects occupying a certain position in the model, for example, vertical plates.

Visualization of Results toolbar (Fig. 4.1-2, d) is available only in the post-processor and is used to set the mode of display of isofields, and also for the control of output of values of the analyzed factor on diagrams, isolines and isofields.

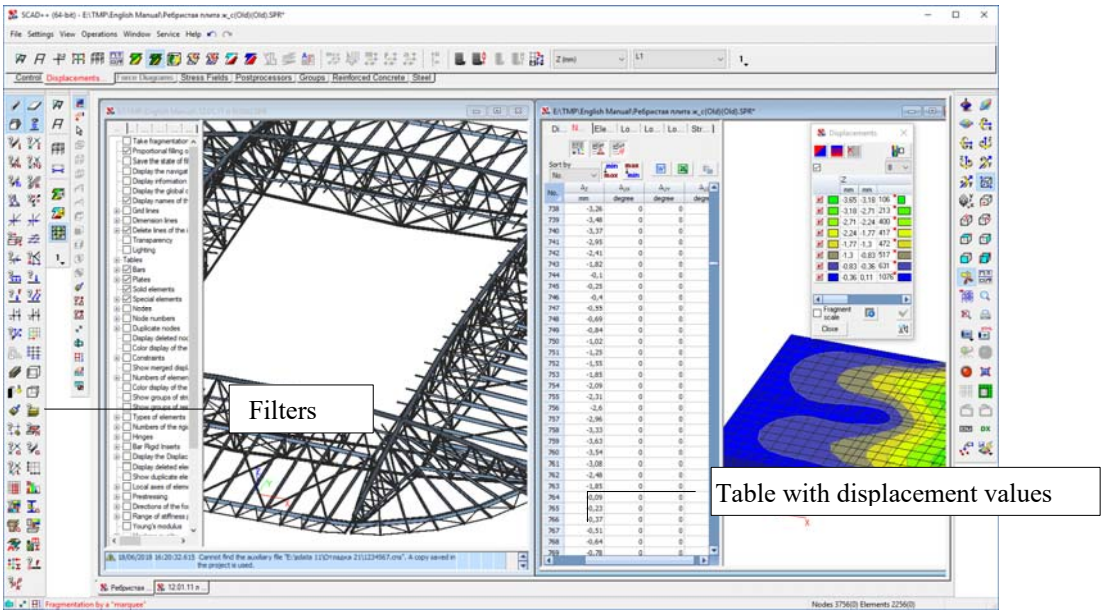


Figure 4.1-3. *Tables of results and filters toolbar*

The work with toolbars and filters is considered in more detail in the respective sections of this book.

Windows

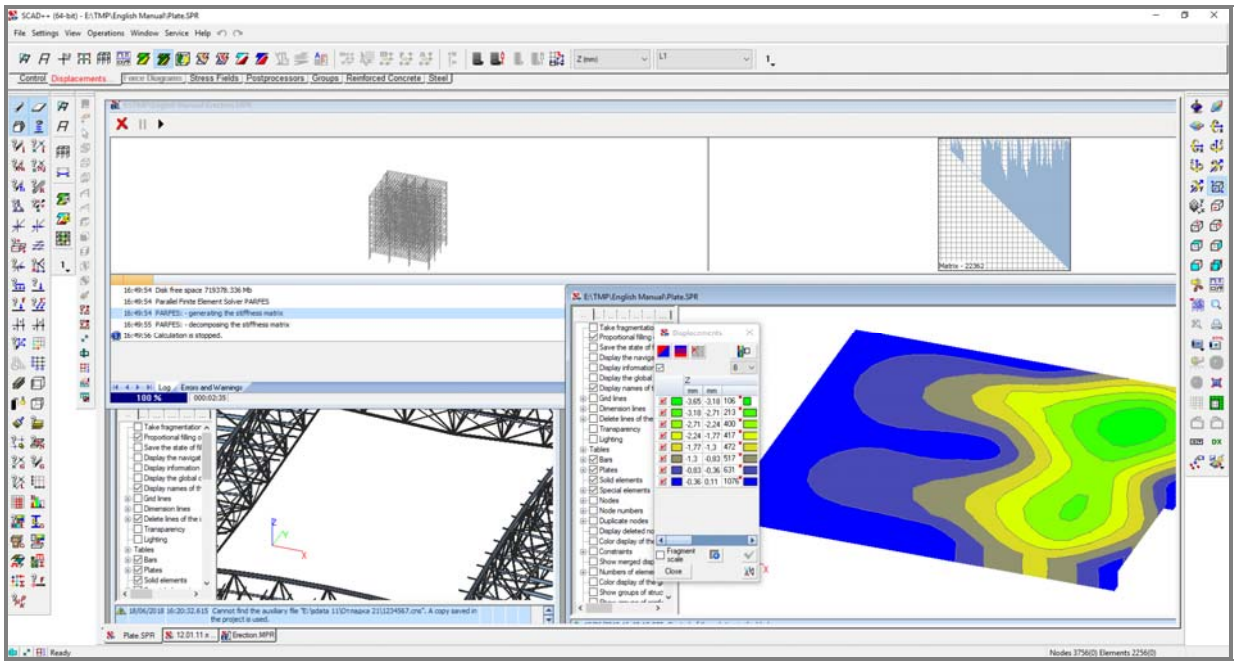


Figure 4.1-4. *Implementation of the* **MDI** *interface in the* **SCAD** *complex*

There are the following types of windows in the **SCAD** complex:

- start window — includes a menu and the **General Operations** toolbar, and is used to create and load the project;
- project control window — includes a project tree and is used to select a working mode;
- window with a design model— is used in the preprocessor and postprocessor;
- processor window — is used in the analysis;
- dialog boxes — are used to prepare the data and to specify the parameters controlling the process of information processing.

Detailed information on the rules of working with the windows is given below in the respective sections of the book. The windows are activated either by tabs in the bottom left corner of the work area or by clicking on the window.

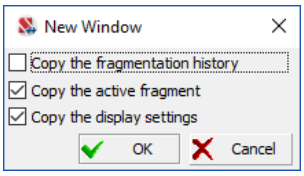


Figure 4.1-5. *The* **New Window** *dialog box*

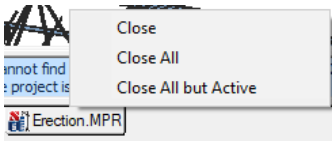


Figure 4.1-6. *Menu of tabs*

When working in the postprocessor the ***Window*** menu contains the **New Window** item, which can be used to create several windows with the information on the results of the analysis. This enables to analyze simultaneously in several windows different factors or fragments of the model, for example, displacements can be analyzed in one window and stresses – in the other one. Activating this menu item opens a dialog box (Fig. 4.1-5), where you can indicate the parameters of the existing window that have to be transferred to the new window (fragmentation history, active fragment, display settings).

Right-clicking on the tab of the window opens a menu (Fig. 4.1-6) which enables to perform the following operations:

- close the respective window;
- close all windows;
- close all windows, except for the one that corresponds to the tab;
- create a new window.

Mouse

Windows environment is designed for the work with a mouse; therefore the majority of operations are carried out with its help. The cursor moves over the screen field together with the mouse. To initiate the operation it is necessary to place the cursor over a control "responsible" for this operation, and click the left mouse button. Depending on the type of the control clicking the button will perform certain actions (for example, make a button shown on the screen "depressed").

“Double left click” or “double click” operation means clicking this button twice with a very short interval (interval can be set when setting the Windows environment).

Moreover, the following mouse controls are used in **SCAD**:

- right button (clicking when the cursor is in the work area of the window) — drops down a menu for the control of the selection of nodes and elements;
- right button (clicking when the cursor is placed over one of the display filter buttons) — opens a multi-tab dialog box for the setting of the display filters;
- middle mouse button (mouse wheel) — clicking and holding the button while dragging the cursor moves the image of the design model in the window;
- mouse wheel (scrolling the wheel) — changes the scale of the image of the design model in the window.

Cursors

Several types of cursors are used in the complex:

standard cursor — for selecting a mode or operation;

cross with a central target— for selecting one node or element;

cross with a rectangle — for selecting a group of nodes or elements using a rectangular marquee;

cross with an arbitrary polygon — for selecting a group of nodes or elements using a marquee of an arbitrary shape;

move the separator line of the window vertically (for example, increase or decrease the size of the message panel);

move the separator line of the window horizontally (for example, increase or decrease the size of the table panel);

move the separator lines simultaneously horizontally and vertically (for example, change the sizes of windows when displaying the model on projections);

mouse pointer for selecting buttons of the table control;

cursor for selecting data in tables.

In those cases when it takes a long time to perform an operation, the screen displays an «hourglass» cursor — . When the operation is completed, a previous cursor is displayed.

Dialog Boxes

Different dialog boxes are used in the complex. A multi-tab dialog box is shown in Fig. 4.1-7.

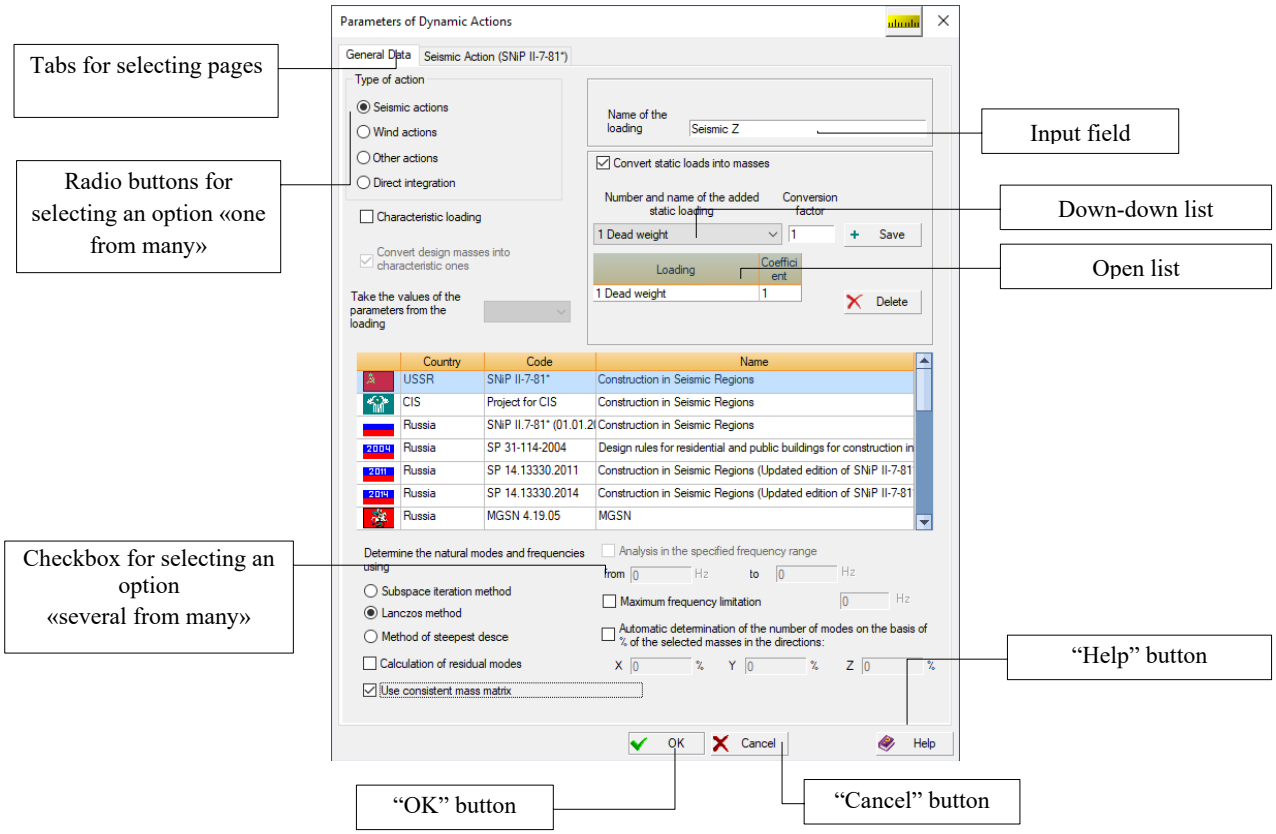


Figure 4.1-7. *Controls of dialog boxes*

Apart from the controls shown in figure 4.1-7, sliders and marker buttons are used in dialog boxes.

Titles of most dialog boxes have a button , clicking which invokes the **Setting of Units of Measurement** dialog box, where you can quickly change the units of measurement of values in the given dialog box.

If the user has opened several design models and the dialog boxes (for example, the color scale dialog box) are activated in one of them, then in order to find out which design model the information in the dialog box refers to, you just have to place the cursor over the title of the dialog box and a tooltip with the name of the file the information refers to will appear (see Fig. 4.1-8).

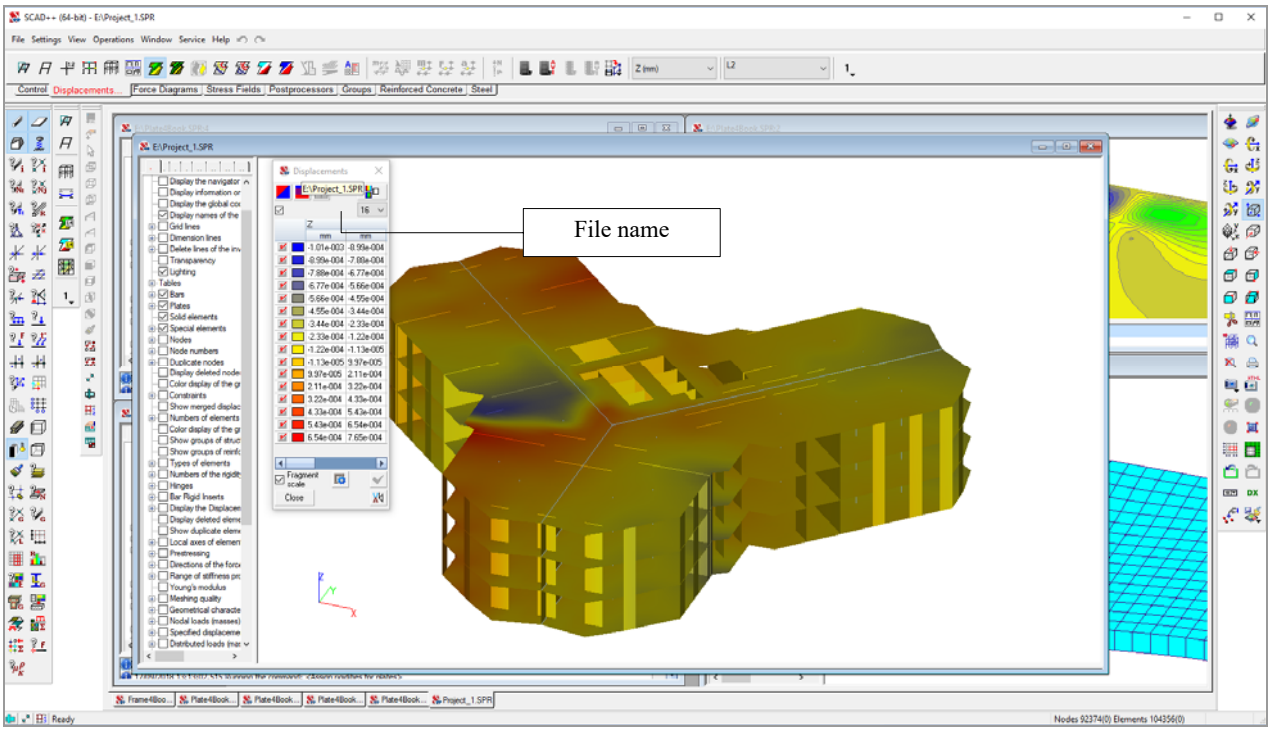


Figure 4.1-8. *Tooltip with the file name.*

Working with Tables

Universal tables are used to enter tabular information in dialog boxes. Operations with the data given in tabular form are performed with the help of a set of commands (Fig. 4.1-9) invoked by right-clicking when the cursor is in the table field (Fig. 4.1-10).

You must observe the following rules when entering the numerical data in tables:

- new rows can be added to the table by clicking the **Add** button;
- selected rows can be deleted by clicking the **Delete** button;
- the fractional and the integer parts of a number must be separated by a symbol assigned by the user during the customization of the operating system, for example, period (**Language and Regional Settings** section on the **Control panel**). Decimal point format is always available;
- numbers with a floating-point can be entered in scientific notation, for example, 3e-5;
- to switch between cells of the same row after entering a number, press the **Tab** key on your keyboard, and to switch between rows of the same column use buttons ↓ and ↑;
- control of the table can be performed with the help of the drop-down menu (Fig. 4.1-9), which appears after right-clicking (cursor must be in the table field);
- to select several successive cells or rows, place the cursor over the first one, click and hold the **left** mouse button, and drag the pointer to the necessary cell or row;
- to select some rows press the **Ctrl** key on your keyboard and holding it point at the numbers of the selected rows fixing each row by clicking the left mouse button.

The set of commands for the control of the table includes the following commands:

- **Copy** — copies the contents of the selected rows or cells (horizontally or vertically) to the **Clipboard**;
- **Paste** — inserts the information from the **Clipboard** into the table starting with the selected cell. If in the process of inserting it turns out that the existing rows are not enough, they are automatically added. Columns are not added, i.e. information on the right from the last column gets lost (disappears);

- **Insert row before selected row(s)** (a selected row is a row with at least one active cell) — a row is added before the selected row; all the cells of the new row will have default values (unless otherwise specified, they are equal to zero);
- **Insert row after selected row(s)** — a row is added after the selected row (the command is performed according to the same rules as the previous one);
- **Clear selected row(s)** — all cells of the selected rows are filled with default values (zero, as a rule);
- **Delete selected row(s)** — deletes all the selected rows;
- **Copy current rows** — the entire row is copied to the clipboard regardless of whether it is selected fully or partially;
- **Paste current rows** — pastes rows that were copied by the **Copy current rows** command.

It should be noted that the last two commands perform the check of the correctness of the information for compliance with the format of tabular data before copying and pasting.

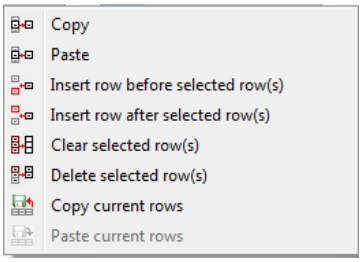


Figure 4.1-9. *Set of commands for working with tabular data*

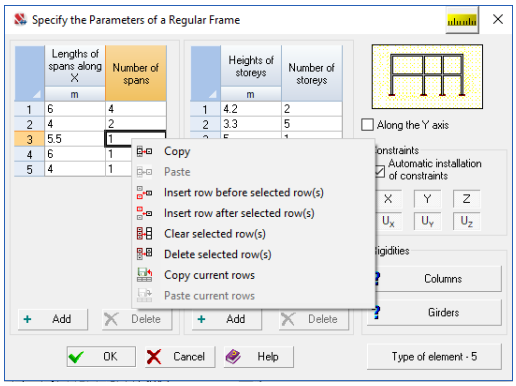


Figure 4.1-10. *Working with tables*

Commands (**Copy** and **Paste**) can be used not only for operations with the current table of the program but also to transfer the tabular data to other tables and applications, including those that are not a part of the **SCAD Office** system (for example, MS Excel).

Tabular Representation of the Data and Results

The left part of the window where the design model is displayed contains a multi-tab panel with the tabular representation of data (and in postprocessor – with the results of the analysis). Tabs enable to switch between pages and proceed to the analysis of particular types of data.

Each page displays information on the current fragment of the design model. Rows that refer to the selected nodes/elements are highlighted in pink. Marker on the selection control panel enables to switch to the mode where only the nodes/elements selected in the model are displayed in the tables. Marker activates the mode which enables to perform the selection of nodes/elements in the model by highlighting the respective rows in the tables.

Information in tables can be sorted according to a certain criterion which is selected from the drop-down list, and buttons , enable to select whether to sort ascending or descending.

Buttons , enable to obtain the contents of tables in the form of a report document or a spreadsheet respectively. Clicking the button copies the list of nodes/elements which are displayed in the table to the clipboard.

When preparing the data double-clicking on most of the cells invokes a dialog box where you can change the data for the node/element corresponding to the row of the table. If such a modification is impossible, the cell is filled in grey. When analyzing the results the dialog will also appear, but you will not be able to change the data (with the exception of the name of the node/element).

The first tab **Display Filters** actually contains information on the settings of the display filters states.

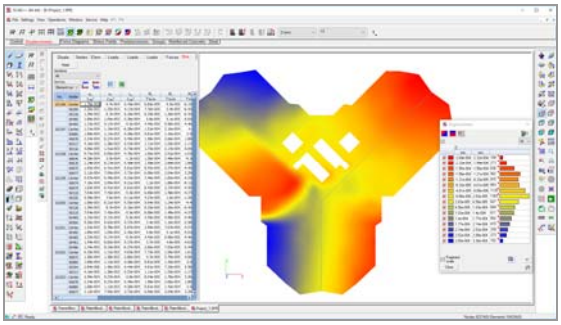


Figure 4.1-11. *The* **Nodes** *tab*

The **Nodes** tab (Fig. 4.1-11) contains information on the nodes of the design model (names of nodes, coordinates, constraints). Markers enable to show or hide the coordinates of nodes and constraints.

If the model has selected nodes, then clicking the button when preparing the data deletes them.

The analysis of results mode enables to obtain the tabular representation of the following information:

— displacements of nodes;

— reactions in constraints.

For dynamic loadings:

— weights of masses;

— accelerations of nodes;

— dynamic loads.

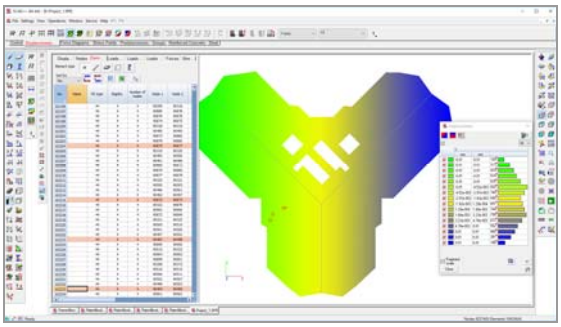


Figure 4.1-12. *The* **Elements** *tab*

The **Elements** tab (Fig. 4.1-12) contains information on the elements of the design model (name, type of the finite element, number of the rigidity type, ...).

The upper part of the table contains filters which enable to reject the display of finite elements of a certain type (bars, plates, ...) in the table, and also the button to delete the selected elements.

Loads in Nodes (Fig. 4.1-13), **Loads on Elements** (Fig. 4.1-14), and **Loads on Regions** tabs contain data on loads of the current loading. Filters in the upper part enable to perform the filtration of the data in the table. For example, for nodes — output only the loads of the *displacement of the constraint* type, and select the output only along the Z axis from the **Directions** drop-down list. Similarly, the filtration by the types of elements, types of loads, coordinate system (local or global), etc. can be performed for elements.

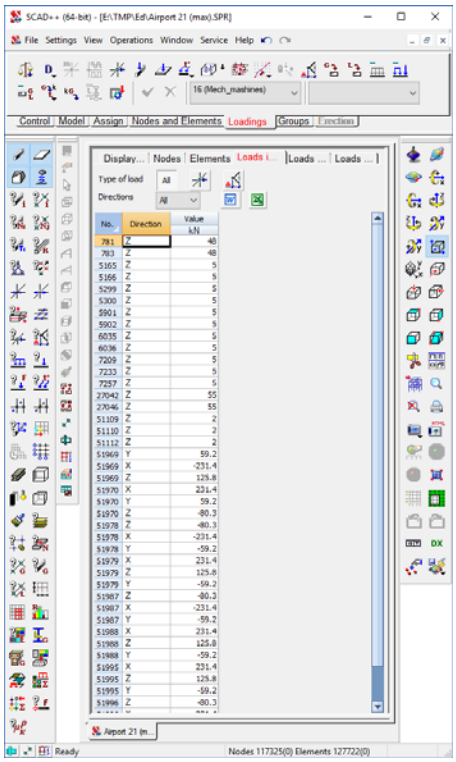


Figure 4.1-13. *The* **Loads in Nodes** *tab*

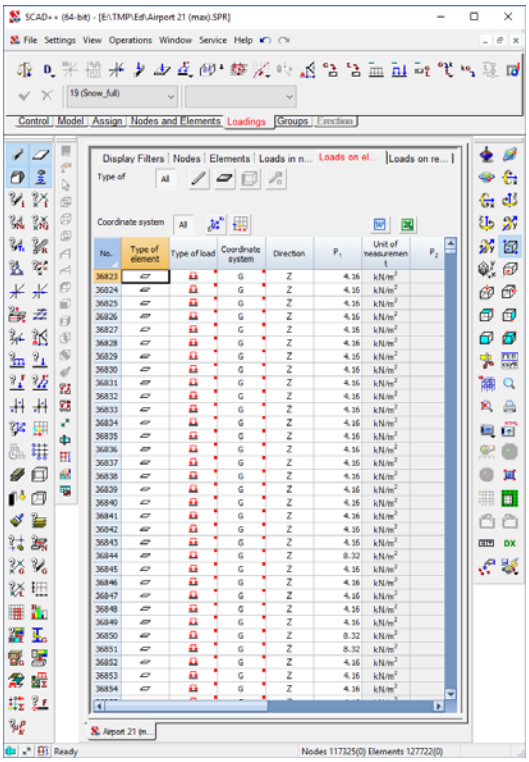


Figure 4.1-14. *The* **Loads on Elements** *tab*

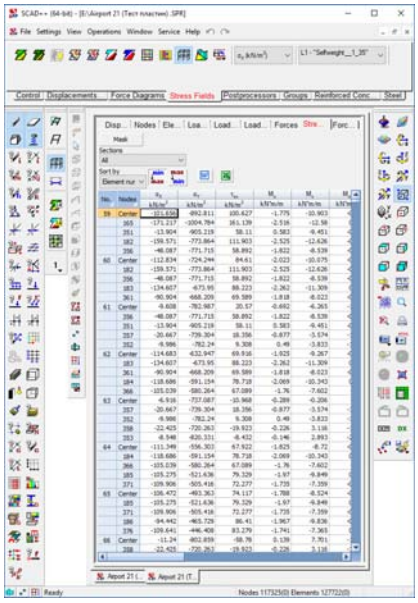


Figure 4.1-15. *The* **Stresses** *tab*

When analyzing the results the **Forces**, **Stresses**, **Forces in Special Elements** tabs (Fig. 4.1-15) appear, where the data on forces in bars, stresses in plates and solid elements, forces in special finite elements are output. The **Mask** button invokes a dialog box (Fig. 4.1-16) where you can specify the types of forces/stresses which should be displayed in the table.

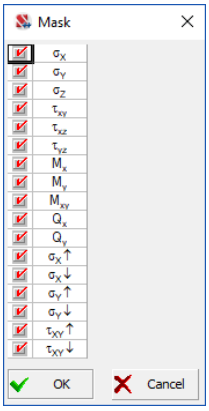


Figure 4.1-16. *The* **Mask** *dialog box*

Since the loading of data on forces/stresses is a relatively long procedure, tabular representation of the information in the **Forces** and **Stresses** tabs is performed only after the user has ordered the graphic display of diagrams or isofields.

List of Messages

When making any changes to the design model, the messages about these changes are recorded in a special log (workbook) located at the lower part of the window displaying the design model (Fig. 4.1-17).

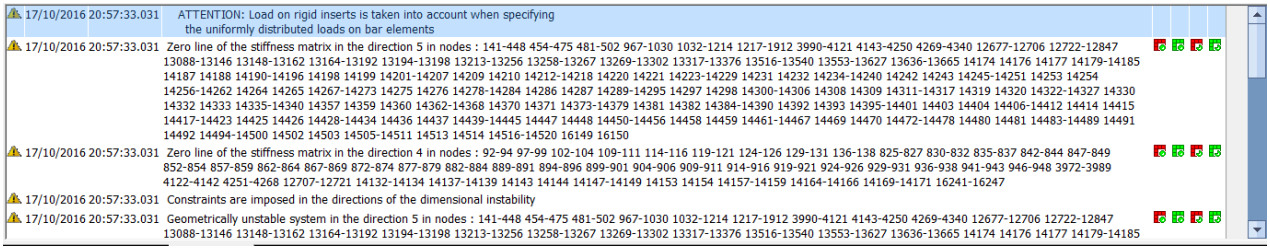


Figure 4.1-17. *List of messages*

This list can be cleared using the button of the **Control** toolbar.

Moreover, the log contains messages on the errors detected, for example, during the control of the design model. In many cases these messages contain buttons which enable to perform the following actions:

— display the entire design model and select elements/nodes the message refers to;

— display the entire design model and select elements/nodes the message refers to for the further fragmentation;

— select elements/nodes from the active fragment the message refers to;

— select elements/nodes from the active fragment the message refers to for the further fragmentation.

If the cursor is in the workbook field and the right mouse button is clicked, a menu appears (Fig. 4.1-18), the first two items of which enable to save the selected line in the **Clipboard** — **Copy line** command, if any line of the workbook has been selected, or all lines of the messages — **Copy all lines** command. Moreover, the buttons for the selection of objects specified in the workbook are duplicated in the menu.

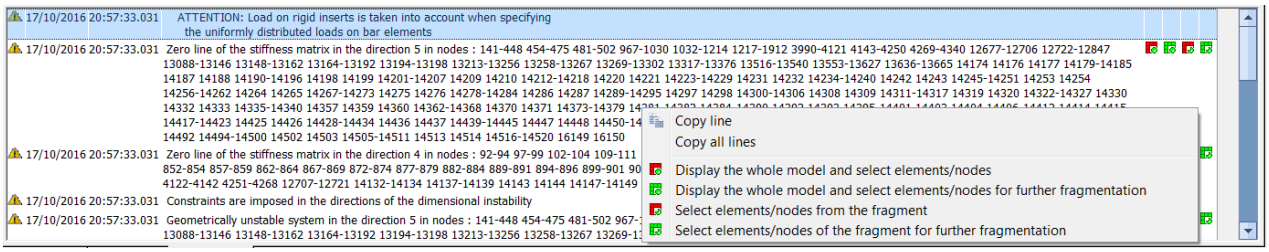


Figure 4.1-18. *Workbook menu*

4.2. Undo and Redo Operations

The ***Operations*** menu (Fig. 4.2-1) enables to return to the previous state of the design model, i.e. cancel the results of the work of the operations performed earlier (**Undo**) or restore the results of the operations after they were cancelled (**Redo**). The **Redo** command is available only after invoking the undo command.

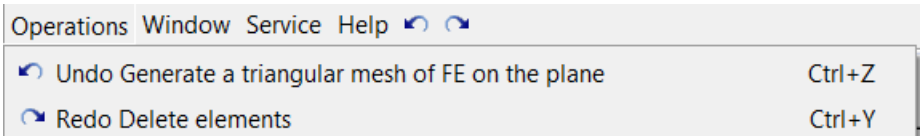


Figure 4.2-1. *Undo and redo operations*

Information in the menu includes the name of the last operation. The operation can be also undone or redone with the help of the combination of buttons **Ctrl+Z** and **Ctrl+Y** respectively. Moreover, the main menu also contains the **Undo** and **Redo** buttons.

4.3. Color Scales

Whenever there is a need to assess the overall picture of the distribution of a factor in the structure or its fragment, color representation of the initial data and calculation results can be used.

This applies, for example, to color marking of the rigidity types or the values of displacements in nodes, to isofields of stresses in plates or the total loads in nodes.

Depending on the type of data given in color, the scales can have a finite number of colors, for example, the number of colors in the scale of the types of rigidities will correspond to the number of the specified types of rigidities. When it comes to the results of the calculation, in most cases, the user himself determines the number of intervals (colors) of the factor representation. Such scales can be called ***interval***.

Color scales are given in the form of dialog boxes, each of which contains a table with the parameters of a factor and the controls of the scale. The dialog boxes of the scale also contain information on the number of objects (nodes or elements) which have the property corresponding to the row of the table, and a representation of the number in the form of a histogram (Fig. 4.3-1).

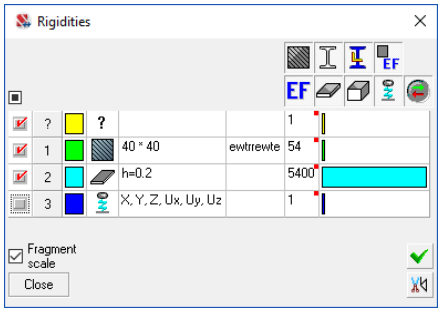


Figure 4.3-1. *Dialog box of the color scale*

Controls of the color scale enable to control the display depending on the type of information. Each interval of the results or the model parameter (e.g., type of rigidity or the value of the subsoil parameter) has a corresponding checkbox and a color field. If the checkbox is checked, the results included in the respective interval or objects with the respective characteristics will be displayed on the model in color (isofields, isolines, etc.) By unchecking the checkboxes you can cancel the color display, for example, of the part of the results with the values of the analyzed factor insignificant for the behavior of the structure.

You can decrease or increase the number of intervals and make them uneven. In order to display such settings use the button – **Apply** in the lower part of the dialog box with the scale. It is possible to simultaneously enable or disable all intervals using the checkbox on the left above the color scale table.

Right-clicking on the table invokes a menu which enables to check or uncheck all checkboxes, invert the selected checkboxes, select nodes and/or elements.

Control of the Interval Color Scales

A set of controls of interval color scales depends on the type of representation of the analyzed factor in the scale. There are the following types of scales:

- uniform scales of force factors, displacements, utilization factors of restrictions, etc. with the automatically adjusted values of intervals (the number of intervals is specified);
- scales of force factors, displacements, utilization factors of restrictions, etc. with the specified value of the interval step (constant step);
- uniform scales with the specified value of the “transition zone”.

The type of scale is selected in the toolbar of the analyzed factor, and the type of representation – directly in the dialog box of the color scale. For this purpose, there are four control buttons in the upper part of the dialog box (except for a scale with the specified value of the "transition zone", where only one button is used):

— specification of the value of the factor f, dividing the scale into two independent parts $f_{min} \div f$ and $f \div f_{max}$, each of which is divided into the respective number of equal intervals;

— uniform coloring of the specified scale interval;

— switching the type of representation of the factor values in the color scale table (in one or two columns);

— selection of the color scale palette.

In order to divide the scale into two parts click the button , and specify the value of the factor that divides the scale in the invoked dialog box. After closing the dialog box by clicking the **OK** button, the scale will be modified according to the new specifications.

Uniform coloring of the specified scale interval means that in the interval of the values of the factor $f_i \div f_n$ the colors in the scale will change smoothly from the color specified for f_i, to the color corresponding to f_n. In order to perform this operation you have to:

- uncheck checkboxes of all intervals of the color scale except for two checkboxes corresponding to the intervals f_i and f_n;
- specify color for one of the intervals:
  - place the cursor over the color field of the selected interval and double left-click;
  - the standard **Colors** dialog box will appear where you have to select the color for this interval;
  - repeat the same actions for the second interval;
- click the button .

Three fixed color scales are implemented in the program: "hot" – red-yellow-blue scale, "cold" – blue-yellow-green scale and "gray" – in the shades of gray. Switching between scales is performed in the drop-down menu that appears after clicking the button – **Select palette**.

The user can also prepare custom color scale palettes:

- specify colors for the intervals of one of the scales;
- click the button — **Select palette** in this color scale;
- select **Save palette** in the drop-down menu, the **Save Palette** dialog box will appear where you have to specify the name of the palette.

Once the new scale is saved, it can be used not only in the current problem but also when working with other models.

Select the Number of Intervals

When working with interval scales you can change the number of intervals of the color representation of the factor by selecting a new value from the list 16 — **Select the number of intervals**. The number of intervals can take any value in the range from two to sixteen, and also twenty four, thirty two, forty eight and sixty four.

Setting the Values of Intervals

When you invoke the color display of the calculation results in the form of an interval scale the entire range of values of the factor is automatically divided into the currently set number of equal intervals. In cases when the uneven division is preferable for the analysis, you can modify the scale by adjusting the values of boundaries of individual intervals. To do this, click the button – **Color scale settings**, and the **Settings** dialog box will appear. The values of boundaries of the intervals can be changed in the table, the filling of which corresponds to the state of the color scale.

This dialog box also contains the **Invert** button, clicking which changes the colors of intervals in the scale. The order of colors is reversed (for example, if in the scale the color changed from blue – for the minimum value to red – for the maximum one, then after inverting the colors will change from red to blue). Changes in the color scale will be performed after closing the dialog box by clicking the **OK** button.

Fragmentation of the Model

The model can be fragmented in such a way that the screen will display only the objects with the values of the analyzed factor within the range of active (with checked checkboxes) intervals. To do this make sure that only the intervals you are interested in are left active in the color scale and click the button – **Fragment**, located in the lower right corner of the color scale.

Control of the Range of the Factor Values in the Color Scale

If the current window displays only a part of the model (the fragmentation of the design model is performed), the values of the analyzed factor can be displayed in the color scale without fragmentation, i.e. in the variation range of the factor values in the whole model, or only in the range of this fragment. The **Fragment scale** checkbox is used to switch the scale.

To finish working with the color scale use the button Close.

Scale with a Constant Step of Intervals

Unlike the scales with an automatic step selection (depending on the number of intervals), the number of intervals in the scales with a constant step will depend on the specified step. Intervals for this type of scales are formed according to the following rule:

- value of factor f_{min} nearest to the minimum and multiple of the specified step is determined;
- the entire range of the factor values, starting with f_{min}, and to the value f_{max}, (number nearest to the maximum factor value, less than the maximum and multiple of the step) is divided into equal intervals;
- the first and the last values of the interval in the color range will be the values from the minimum factor value to f_{min} and from f_{max} to the maximum factor value.

The value of the step is specified in the **Settings** dialog box, which is invoked by clicking the **Color scale settings** button — .

If a certain scale has been changed (set), then these settings are saved in a file with the extension .SCADModelColor in the catalogue of the design model. This file with settings is deleted when you change the design model. If you want to transfer the initial data, the results of the calculation and the color scale settings to another computer, you should not forget to copy files of the SCADModelColor type.

Scales with the Specified Value of the "Transition Zone"

A characteristic feature of this scale is a zone, conventionally called "transition" (although it is possible that the user will be for some reason interested only in this range of the factor values). The number of intervals in such a scale is always odd, the minimum number is three, and the maximum one is fifteen. The settings of the zone are performed in the **Settings** dialog box, which is invoked by clicking the **Color scale settings** button — . This dialog box is used to specify the value of the factor – f, around which the transition zone is formed and the value that characterizes the size of this zone Δf. Interval of the factor values within the zone lies from $f - \Delta f$ to $f + \Delta f$. Intervals lying outside the transition zone are divided into equal parts depending on the specified number of intervals.

The upper part of the dialog box of the color scale contains only one setting button — for changing the representation of the factor values in the table of the color scale (in one or in two columns).

4.4. Solution of the Simplest Problem

Since, as noted above, the principles of control of the **SCAD** complex are not much different from the principles adopted for most applications implemented in the Windows operating environment, the given general information is enough to try to solve the simplest problem. To do this, perform the following actions:

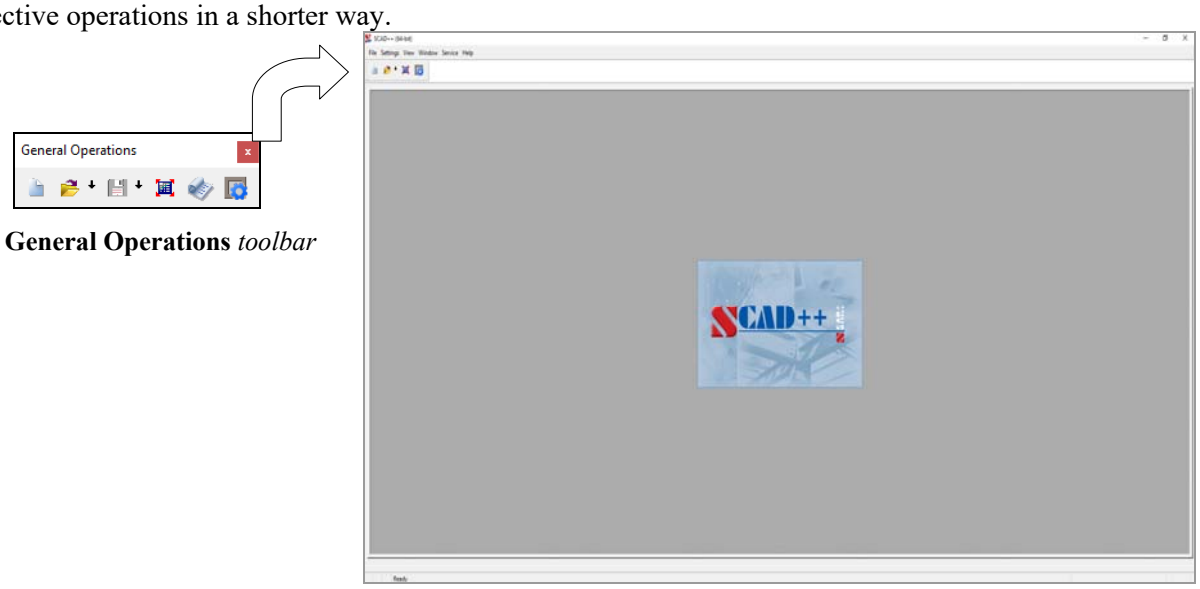
- load the software;
- create a new project;
- create a design model;

- perform the calculation;
- perform the analysis of the calculation results.

Loading the Software

In order to start working with the software, place the cursor over the **SCAD++** icon and double **left-click**. The startup image will appear against the background of the initialization window (Fig. 4.4-1). It takes a few seconds to load the complex. During this time, the screen displays the hourglass "waiting" cursor (unless, of course, it is not replaced on your computer by a more exotic one). The completion of loading is signalized by a change of the cursor from the waiting one to the arrow.

Initialization window contains a menu and the **General Operations** toolbar. The toolbar contains five buttons. The first two buttons duplicate the similar functions of the ***File*** menu, and enable to perform the respective operations in a shorter way.



The **General Operations** *toolbar*

Figure 4.4-1. *The* **Structure CAD** *initialization window*

— Create a new project;

— Open project;

— Save project;

— Switching to the full-screen mode enables to increase the size of the working window vertically by hiding the title bar and the taskbar. Clicking the button for the second time returns the window to its original size;

— View the analysis log in the text editor specified in the settings;

— the **Settings** button invokes the **Settings** dialog box where you can specify the settings of the software (units of measurement, fonts, assortments of rolled profiles, etc.), assumed for the current problem.

If one or more buttons are in gray shades, it means that the respective operation or option is currently not available.

Creating a New Project

In order to create a new project, place the cursor over the respective button of the toolbar and left-click. The **New Project** dialog box will appear on the screen (Fig. 4.4-2), where you have to specify information on the project (name, company, object), select the type of the design model (standard, design model of the variation of models mode, erection or a model from enlarged elements created in the **FORUM** preprocessor).

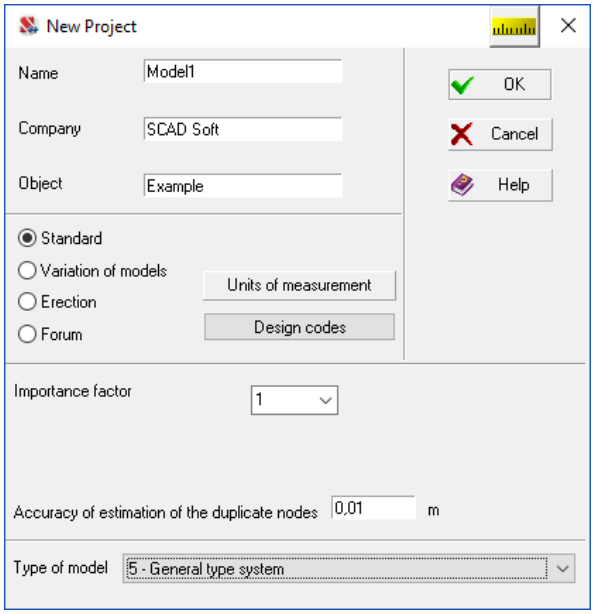


Figure 4.4-2. *The* **New Project** *dialog box*

The **Units of measurement** and **Design codes** buttons invoke the respective dialog boxes, which enable to set the respective parameters. The default units of measurement are meters for the sizes of the design model, centimeters for the sizes of sections, and tons – for forces. Let's agree that when creating a design model of the simplest problem, these settings will not be changed, as well as the default design codes. The description of the respective dialog boxes is given below.

Since the importance factors depend on the design codes, their values are selected from the respective list after specifying the design code (when DBN B.1.2-14-2009 is used — there are two factors, — one for the ultimate and one for the serviceability limit state).

Type of the design model is selected from the list and determines the content and the maximum number of the degrees of freedom in the nodes of the model, and it also describes the features of its stress-strain state. The selected type of model must include all the necessary degrees of freedom for the finite elements used in the model. The default type of model is **type 5** — the general type system where each node has six degrees of freedom: linear displacements X, Y, Z and rotation angles U_X, U_Y, U_Z.

This dialog box is also used to specify the parameter determining the accuracy of the assessment of duplicate nodes (the maximum distance between nodes at which the nodes are considered to be duplicate).

Almost all parameters specified in this dialog box can be changed in the course of creating the design model.

The **New Project** dialog box like most other dialog boxes includes three command buttons: **OK**, **Cancel** and **Help**.

OK — closes the dialog box automatically saving all the input information.

Cancel — closes the dialog box without saving the input information.

Help — provides access to the help information.

Thus, the **New Project** dialog box is used to customize the software before starting to work with the project.

Clicking the **OK** button closes the dialog box and the screen will display a standard **Save As** dialog box, where you have to specify the name of the file of the new project (let's name the file of the simplest

problem, for example, Project 1). This name will be given to the actual file of the project, and also to all service files and files with the calculation results. The file with the **.SPR** extension will be automatically placed in the directory specified during the installation of the complex for saving the initial data.

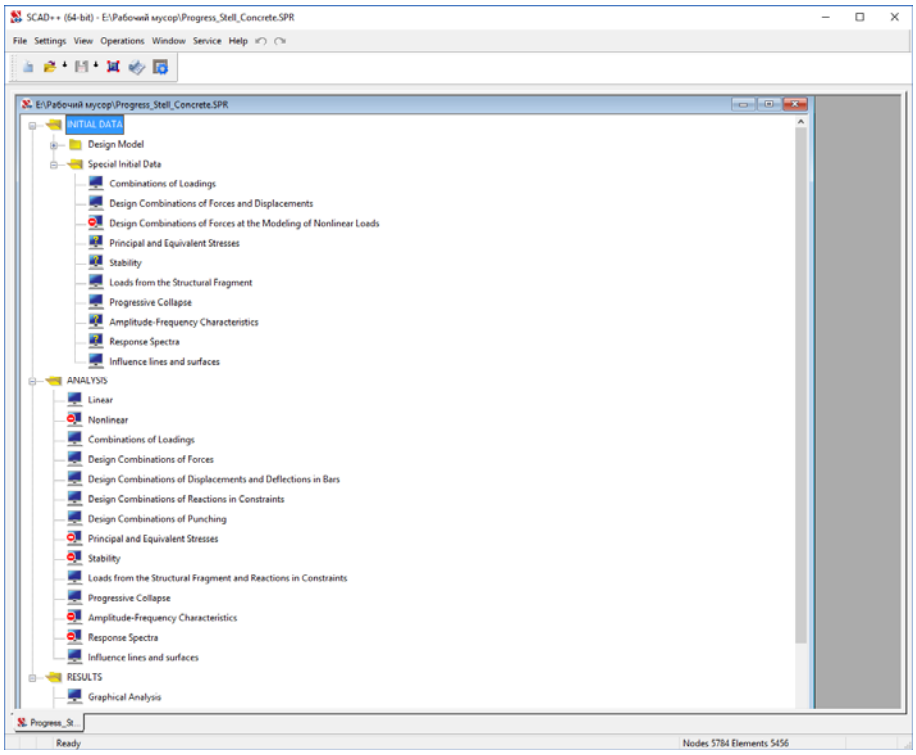


Figure 4.4-3. **Project tree**

After creating the file of the project the control is given to the **Project tree** (Fig. 4.4-3), which contains three sections of the first level that are actually the stages of the solution of the problem: **Initial Data**, **Analysis** and **Results**.

These sections contain links to the sections of the second level, which detail the performed operations, for example, **Design Model**, **Linear Calculation** or **Print Tables**.

Sections **Design Model** and **Special Initial Data** contain subsections with a list of initial data.

The **Project tree** reflects the state of the project, i.e. the presence or absence of the respective type of data in the project, and the availability of functions, such as calculation and the analysis of results. The **Tree** is used to initialize all the functions – data input, calculation or the graphical analysis of the results. In order to do it, just place the cursor over the **name** of the corresponding branch and left-click. Moreover, the **Project tree** illustrates the structure of the software.

"Control" Section of the Preprocessor Toolbar

The work on a new project begins with creating a design model. To do this, place the cursor over the **Design Model** section of the **Project tree** and left-click. The control will be transferred to the graphical preprocessor, where the design model is created. The preprocessor toolbar contains various operations of creating the geometry of the model, specifying the boundary conditions, loads, etc. After the start of the preprocessor the **Control** section is active in the toolbar (Fig. 4.4-4).

Figure 4.4-4. *The* **Control** *section of the preprocessor toolbar*

— **Exit** — finish the work with the program;

— **New project** — create a new project;

— **Open project** — load a previously created project;

— **Save** — save the current project without changing the name;

— **Print** — print the current image of the model;

— **Exit to project control screen** — transfer control to the **Project tree**;

— **Pack data** — the deleted nodes and elements are removed from the project, and the remaining ones are renumbered; it is used in cases when the work on creating the model is finished, and there is no need to save information on the deleted nodes and elements;

— **Show/hide visualization toolbar** — hide or display the **Visualization** toolbar on the screen;

— **Show/hide filter toolbar** — hide or display the **Display Filters** toolbar on the screen;

— **Save the screen image** — the current image of the model is saved as a Windows metafile (in a file with extension **.emf** or **.wmf**) or raster image (.**bmp**, .**png**, ...) and can be used, for example, when preparing a report;

— **Save as HTML** — the current image of the model is saved as an HTML-file and can be used for example to put it on the Internet;

— **Save the fragment of the model** — the fragment of the design model visible on the screen is saved as an individual project;

— **Express control of the model** — perform the formal control of the design model in the process of preparation of the initial data;

— **Check the availability for analysis** — perform the extensive control of the design model, check the availability of the model for the analysis;

— **Correct finite elements** — delete or replace invalid (in terms of shape) finite elements;

— **Determine the area, perimeter, and angles of polygons** — determine the area, perimeter, and angles of the polygon specified on a plane fragment of the design model;

— **View the analysis log** — the analysis log is output in the text editor specified in the settings;

— **Settings** — specify the settings of the software (units of measurement, fonts, assortments of rolled profiles, etc.), assumed for the current problem;

— **Clear the list of messages** — clear the workbook with messages on the process of formation and solution of the current problem.

Creating the Design Model, Calculation and Analysis of Results

This section provides a quick description of the operations of creating a design model. After performing step by step the actions described below the simplest model will be created. Operations related to its modification are not considered here, since the main goal is to master the basic principles of the software control.

Operations of creating the design model and analyzing the calculation results are implemented in a single graphical environment. The location of controls and, in fact, the control of the software is the same for these modes. They only have different sets of functions.

Input the Parameters of the Design Model

Let's use an example given in Fig. 4.4-5 – a frame with three spans and two storeys, as a training model. To create a design model of such an object, you can use the standard operation **Generate a frame prototype** – . This operation, like most other operations of creating design models based on parametric prototypes, is invoked from the **Model** section of the toolbar (Fig. 4.4-6).

Place the cursor over an icon with the image of a multi-storey and multi-span frame in the **Select Frame Configuration** dialog box (Fig. 4.4-7) and left-click. After clicking the **OK** button this dialog box will close and the **Specify the Parameters of a Regular Frame** dialog box will appear (Fig. 4.4-8).

This dialog box is used to specify the geometric sizes of the frame and the rigidities of elements, and to impose constraints on nodes. Two tables are used to specify the sizes: the left one — to describe the spans and the right one — to describe the storeys.

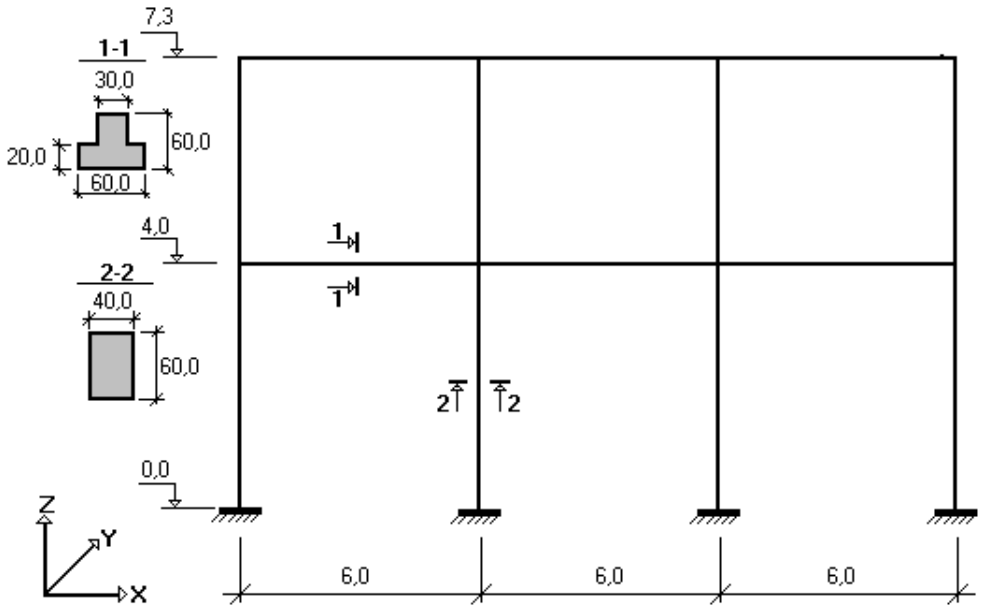


Figure 4.4-5. *Design model*

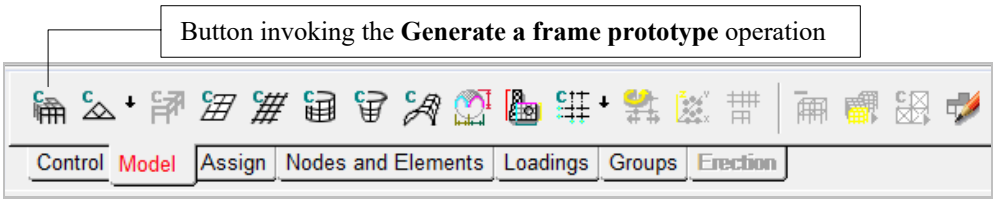


Figure 4.4-6. *The* **Model** *toolbar*

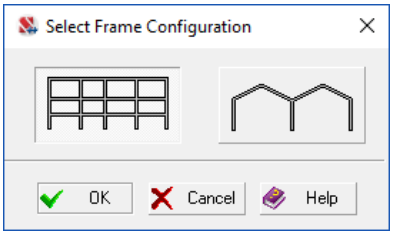


Figure 4.4-7. *The* **Select Frame Configuration** *dialog box*

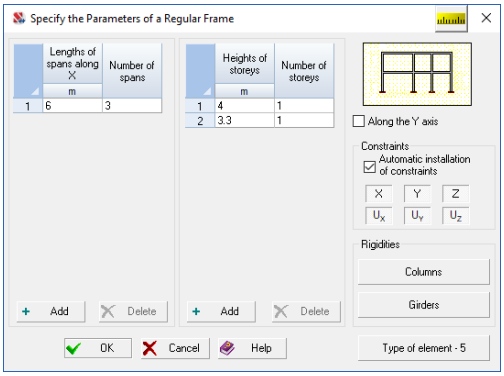


Figure 4.4-8. *The* **Specify the Parameters of a Regular Frame** *dialog box*

Let's specify three spans of 6 m each into the table with the description of spans, and one storey with the height of 4 m and another with the height of 3.3 m into the table with the description of storeys.

The checked **Automatic installation of constraints** checkbox means that constraints in the directions corresponding to the active (pressed) marker buttons with the name of the degrees of freedom will be installed at the base of the columns of the generated frame. Markers corresponding to the degrees of freedom of the type of model selected for the problem are activated by default.

Columns and **Girders** buttons are used to specify the stiffness properties for the respective elements of the model, and the **Type of element** button is used to specify the types of finite elements.

Specification of the Stiffness Properties of the Elements

When specifying the parameters of the frame it is assumed that the input values of the stiffness properties of columns are assigned to all columns, and those of girders are assigned to all girders.

The stiffness properties of columns can be specified in the **Rigidity of Bar Elements** dialog box (Fig. 4.4-9), which appears after clicking the **Columns** button.

Sections of elements selected for the frame (Fig. 4.4-5) belong to the so-called parametric sections, and in order to specify them you have to enable the respective radio button in the first tab (**General Data**) of the dialog box. Stiffness properties of parametric sections are calculated automatically depending on the material, shape and sizes of the section. These data can be specified in the **Parametric Sections** tab (Fig. 4.4-10).

It is recommended to perform the actions in the following sequence:

- select the class of concrete from the **Material** list, for example, B25; the fields with the properties of the selected material are filled automatically;
- press the marker-button with the image of a column section, in this case — rectangle;
- specify the sizes of the section in the text fields;
- to check the specified data you can use the **Characteristics of the section** button, clicking which invokes the **Section Properties** dialog box with the characteristics of the section;
- if the specifications are correct – click the **OK** button of the **Rigidity of Bar Elements** dialog box (the dialog box will close).

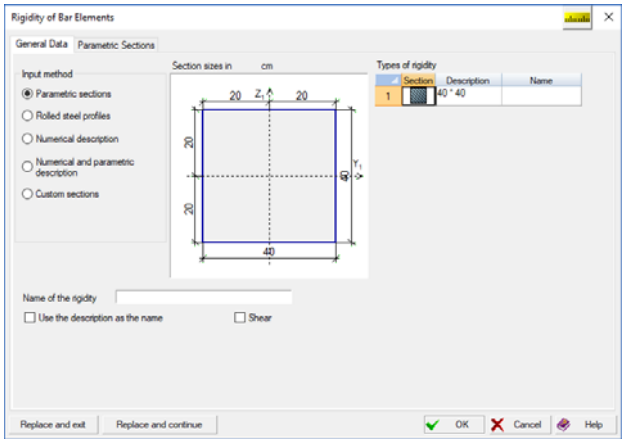


Figure 4.4-9. *The* **Rigidity of Bar Elements** *dialog box*

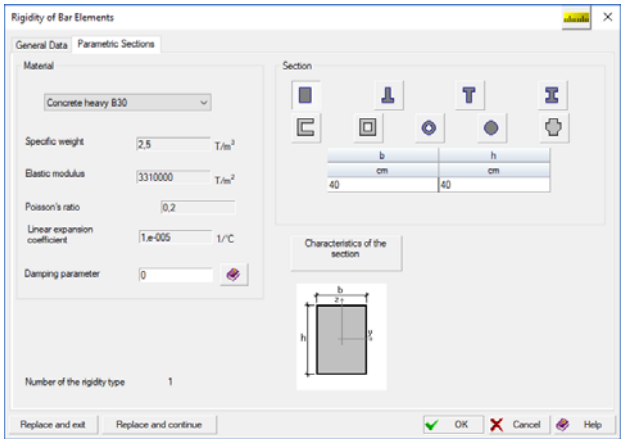


Figure 4.4-10. *The* **Parametric Sections** *tab*

After the last operation the control goes back to the dialog box for the specification of the parameters of the frame. Click the **Girders** button and perform the above actions for the specification of the stiffness properties of girders. Pay attention to Fig. 4.4-5 – unlike columns, girders have a T-section.

Generation of the Model

Since all parameters of the frame are determined (types of finite elements are assigned automatically based on the selected type of the model), the design model is generated after clicking the **OK** button and its image appears in the window (Fig. 4.4-11).

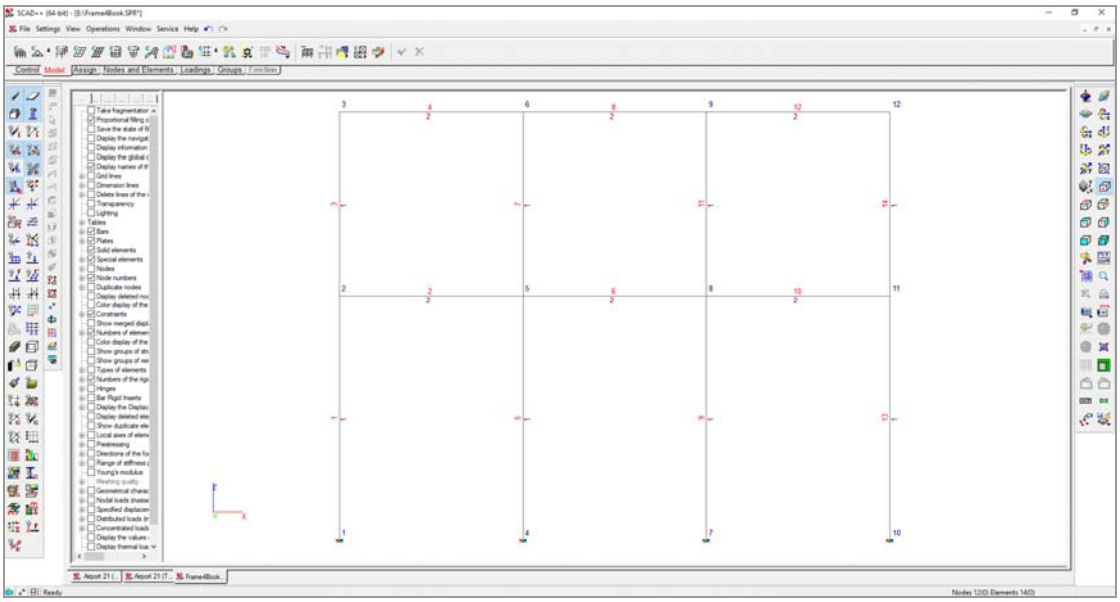


Figure 4.4-11. *Design model of the frame*

If you use the filters of the display control (Fig. 4-1), information on the numbers of nodes, elements, and types of rigidity can be obtained directly on the design model.

In order to do it, activate the respective buttons of the filters toolbar:

— numbers of elements;

— constraints;

— numbers of nodes;

— types of rigidity;

— display nodes.

A design model with nodes, their numbers, numbers of finite elements, numbers of types of rigidity and designations of constraints is given in Fig. 4.4-11. In order to perform the calculation, it is necessary to specify the loads.

Specification of Loads

Operations of the specification of loads are invoked from the **Loadings** section of the toolbar (Fig. 4.4-12).

These operations provide an automatic generation of the self-weight of the structure, specification of different types of dynamic and static loads on nodes and elements of the model, saving of the specified loads as systems of loadings or groups of loads.

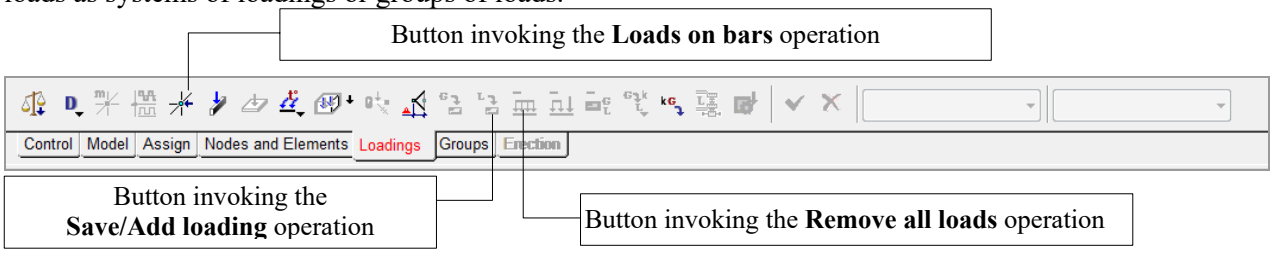


Figure 4.4-12. *The* **Loadings** *section of the preprocessor toolbar*

Let's specify loads on the design model in the form of two loadings:

Loading 1 — describes the permanent load on floors and roofs and the self-weight of columns, the first of which is specified as a uniformly distributed load of 4,36 T/m on girders, and the second one — on columns — 0,66 T/m. Loads are specified in the direction of the Z axis of the global coordinate system;

Loading 2 — describes the temporary load with a reduced characteristic value on girders. It is specified in the same way as **loading 1**, but with another value of the load — 3,57.

When performing the calculation it is assumed that the design values of loads are given, i.e. characteristic values are multiplied by the safety factor for load. Thus, when specifying loads on nodes and elements of the model, all loads must be first reduced to their design values.

Use the following buttons of the toolbar to specify these loads:

— Loads on bars;

— Remove all loads;

— Save/Add loading.

It is necessary to perform the following operations to specify the loads:

- click the button for the specification of loads on bar elements , the **Specify Loads on Bar Elements** dialog box will appear (Fig. 4.4-13), where you have to specify the type, direction and value of the first load;
- click the **OK** button in the dialog box;
- using one the cursors select elements of the model the load is applied to (the selected elements will be displayed in red);
- click the button in the toolbar of the **Loadings** mode, or press the **Enter** button on the keyboard;
- repeat the above actions for the second type of load of the first loading.

Please pay attention to the actions listed above. The vast majority of the operations with nodes and elements of the design model are performed exactly in this order. Shortly this sequence can be written as follows: select an operation, specify the parameters, select objects in the model, click the OK button in the toolbar the selected operation belongs to, or press the Enter key on the keyboard.

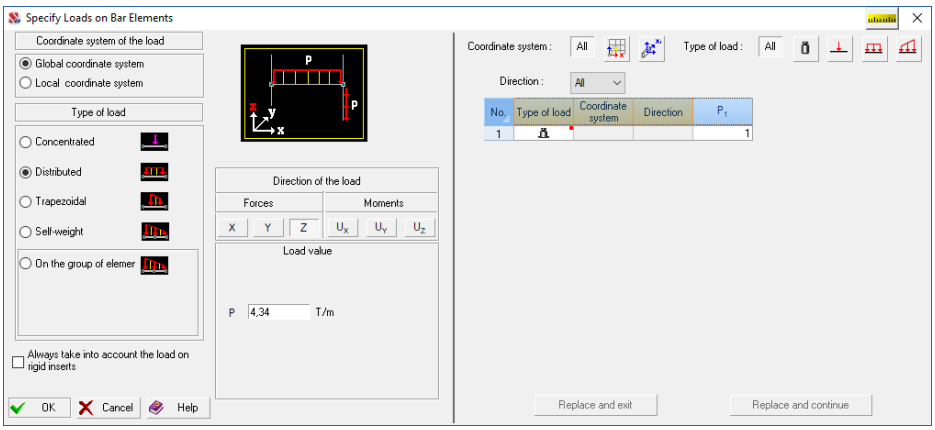


Figure 4.4-13. *The* **Specify Loads on Bar Elements** *dialog box*

If you activate the display filter of distributed loads , the specified loads will be shown on the design model. If the visualization filter of the load values is enabled , the load value will be shown next to the load. A design model with the display of the specified loads is shown in Fig. 4.4-14.

In order to save the loading click the **Save/Add loading** button — in the toolbar, and the **Save Loading** dialog box will appear (Fig. 4.4-15).

This dialog box is used to specify the name of the loading, select the type of loading and the type of load from the respective lists, and to specify the safety factor for load. You also have to indicate whether the specified loads are characteristic or design. When all the data is specified, click the **Save as new** button and the new loading will be saved. Information on the loading will appear in the loadings table of the dialog box.

A number is assigned to the loading automatically. You do not necessarily have to specify the name of the loading, but this information helps in the analysis of the initial data and the calculation results. The type of loading, the type of load and the safety factor for load will be used as the initial data of design combinations of forces (DCF).

In order to complete the specification of the current loading, it has to be saved (in the project). Otherwise the loading will not be taken into account when performing the calculation.

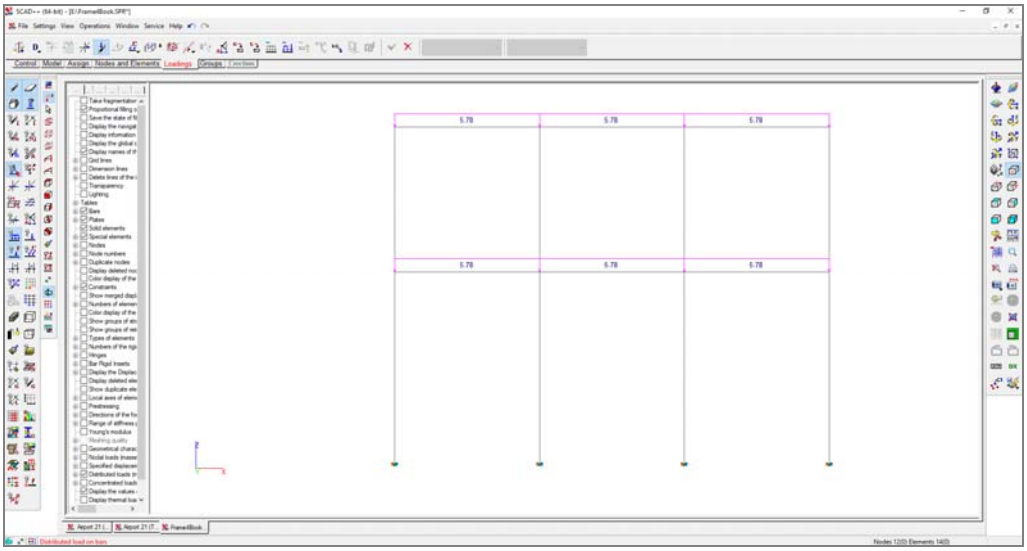


Figure 4.4-14. *Design model with displayed loads*

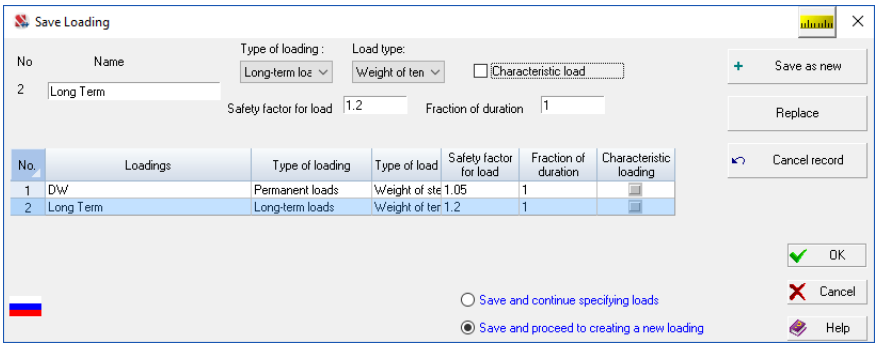


Figure 4.4-15. *The* **Save Loading** *dialog box*

Specification of the Next Loading

In cases when after saving the current loading you want to create a new one, it is necessary to remove the specified loads from the design model. This will be done automatically if before closing the **Save Loading** dialog box you enable the **Save and proceed to creating a new loading** option. If for some reason you do not do it, then after saving the loading in the project it will still be active with all the loads specified in it.

In this situation to proceed to creating a new loading use the **Remove all loads** operation — .

Create a second loading including a temporary load with a reduced characteristic value. Use the above sequence of actions and specify the loads on the elements of the design model, and then save the loading, giving it a name, type, and indicating the respective type of load.

Saving and Loading Files of the Project when Working with the Model

If when working with a model you need to save its current state on the disk, use the **Save** operation. If it is necessary to save the current state of the model under another name, use the **Save Project As …** operation available in the **File** menu. In the last case a model with a new name will be available for working.

When loading a design model from the disk the software may generate the following message:

"The document has been changed since it was last saved. To return to the saved version close the current document and reopen it".

This message appears when a user modifies the design model and attempts to load the same model from the disk. The software suggests to first save the modified model on the disk, and then to load it. If for some reason the modified model has to be replaced by a model saved on the disk without saving the current state of the model, it is necessary to close the active model using the **Close Project** operation in the **File** menu, and then load the model from the disk.

4.5. Calculation

Mathematics may be compared to a mill of exquisite workmanship, which grinds your stuff to any degree of fineness; but, nevertheless, what you get out depends on what you put in; and as the grandest mill in the world will not extract wheat flour from peas cods, so pages of formulae will not get a definite result out of loose data

T. HUXLEY

The input data is sufficient to perform the analysis of the created design model. Most likely, the simplest problem will be set correctly. However, when preparing more complex design models it is advisable to perform a control of the generated model before the calculation. Special operations in the **Control** section of the preprocessor toolbar are used for this purpose. A formal control of the design model in the process of the preparation of the input data is performed by the **Express control of the model** operation (Fig. 4.5-1) — , the **Check the availability for analysis** operation (Fig. 4.5-2) — , is used for the extensive control similar to that performed during the calculation, and the **Correct finite elements** operation (Fig. 4.5-3) — , is used to control the correctness of the shape and to correct invalid finite elements.

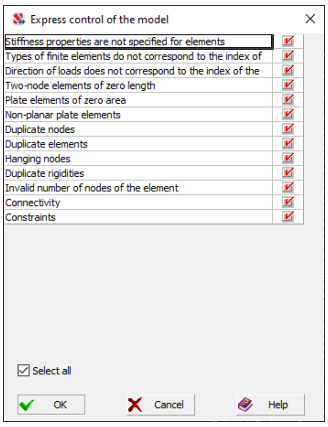


Figure 4.5-1. *The* **Express Control of the Model** *dialog box*

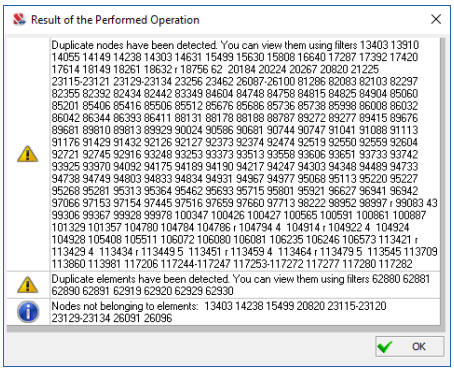


Figure 4.5-2. Message window with the results of the extensive control

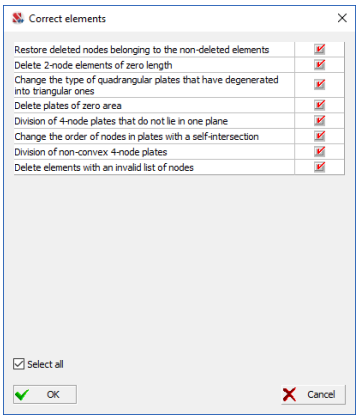


Figure 4.5-3. *The* **Correct Finite Elements** *dialog box*

Before performing the express control it is necessary to indicate the parameters of the design model that have to be checked with the help of the checkboxes in the **Express Control of the Model** dialog box. The list of these parameters is quite obvious. We should probably only comment on the concept of *connectivity of the model*.

A model is considered to be connected if for every pair of elements (E_1,E_2) we can find a sequence of elements beginning with the element E_1 and ending with the element E_2 in which each successive element has a common node (or common nodes) with the previous one. Moreover, merged displacements are taken into account in the analysis of connectivity.

If it turns out that the model is not connected, i.e., consists of several fragments, the information on the number of elements included in each of these fragments is output in the list of messages (see Fig.4.1-17). You can perform the selection or fragmentation of the respective part of the model using the following controls of the list of messages — .

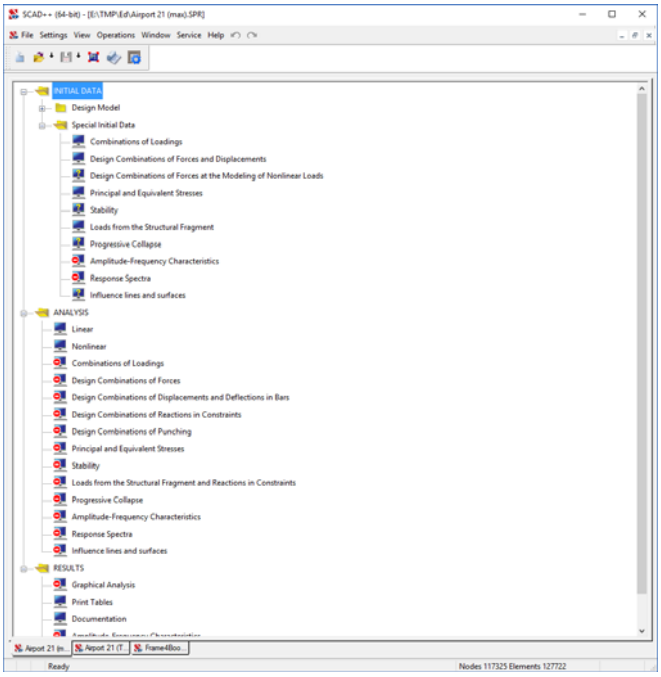


Figure 4.5-4. *Project tree*

If the *Constraints* checkbox is checked, the software will additionally check whether the design model is constrained (whether constraints, elastic constraints or elastic subgrade are specified). If the model is not

connected, i.e., consists of several fragments, the check is performed for each fragment. The absence of constraints means that the design model is dimensionally unstable (note that constraints do not guarantee its dimensional stability).

To perform the calculation it is necessary to return to the **Project tree** by activating the **Control** tab in the toolbar and clicking the button — **Exit to project control screen** (**Project tree**).

If there is an icon next to the **Linear** item of the project tree, it means that the design model is ready for the calculation. Icon means that the calculation is not available. The reason for this, as a rule, is a complete or partial absence of the required initial data. These data include loads, stiffness properties and, of course, the description of the geometry of the design model. In cases when the optional data is missing (for example, the description of hinges the model might not have), the icon of the respective branch of the **Project tree** in the **Design Model** section contains a question mark .

Place the cursor over the **Linear** item in the **Calculation** group of functions and left-click (Fig. 4.5-4). The **Calculation Parameters** dialog box will appear (Fig. 4.5-5), where you can set the calculation parameters. A detailed description of the settings is provided in the following chapters. Let's assume the default values of the parameters for the calculation, and click the **OK** button.

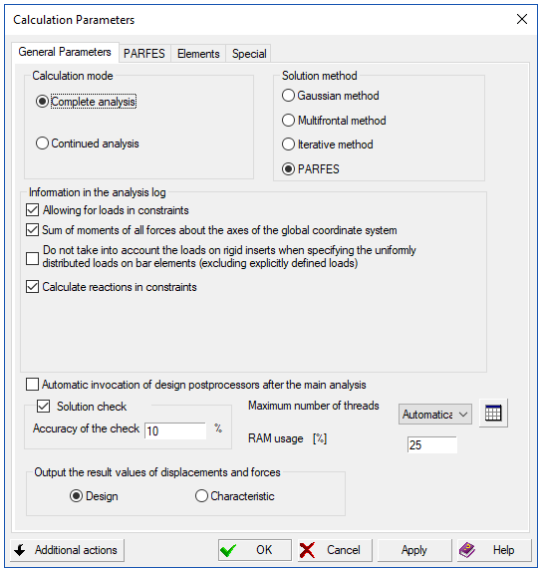


Figure 4.5-5. *The* **Calculation Parameters** *dialog box*

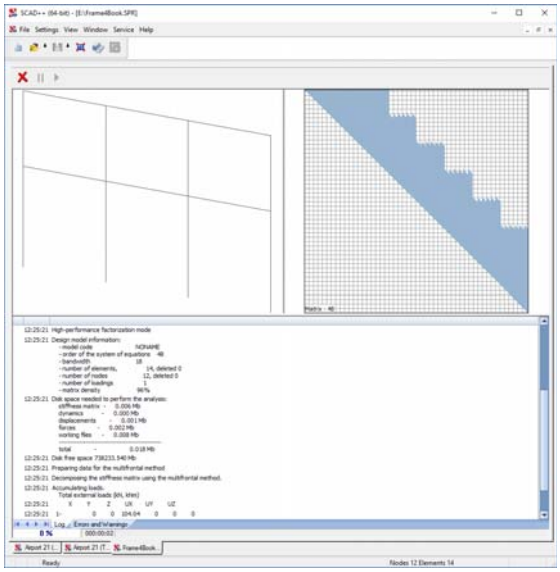


Figure 4.5-6. *Window displaying the calculation progress*

The calculation progress is displayed in a special window (Fig. 4.5-6), the title of which contains the name of the problem. The window consists of three fields, the first one displays the design model, the second one contains the profile of the stiffness matrix, and the third one provides the analysis log. If the cursor is in the first field (design model), you can rotate the model by holding the left mouse button down and moving the mouse (a double click will restore the image to its original state).

Buttons in the toolbar of this window invoke the following operations:

— **Close the calculation window** — aborts the current calculation;

— **Stop the calculation** — stops the calculation;

— **Continue the calculation** — continues the calculation after the stop.

The analysis log contains all the information on the calculation progress, errors that have occurred and warnings, as well as the data on the design model. To view the log use the respective operation , which can be invoked from the **Control** toolbar of the pre- and post-processor or the **Project tree** window.

4.6. Graphical Analysis of the Calculation Results

After completing the calculation and clicking the **Close the calculation window** button the control goes to the project tree. If the calculation was successful, you can proceed to the analysis of the obtained solution.

Place the cursor over the **Graphical Analysis** item of the **Results** section of the project tree and left-click. This will activate the window of the graphical postprocessor (Fig. 4.6-1), the toolbar of which contains tabs of the visualization panel for the calculation results (displacements, forces and stresses), as well as the results of work of calculation and design postprocessors – tabs **Postprocessors** (analysis of loads from the structural fragment, the principal and equivalent stresses, etc.), **Steel** (check of elements of steel structures), **Reinforced Concrete** (check of elements of reinforced concrete structures).

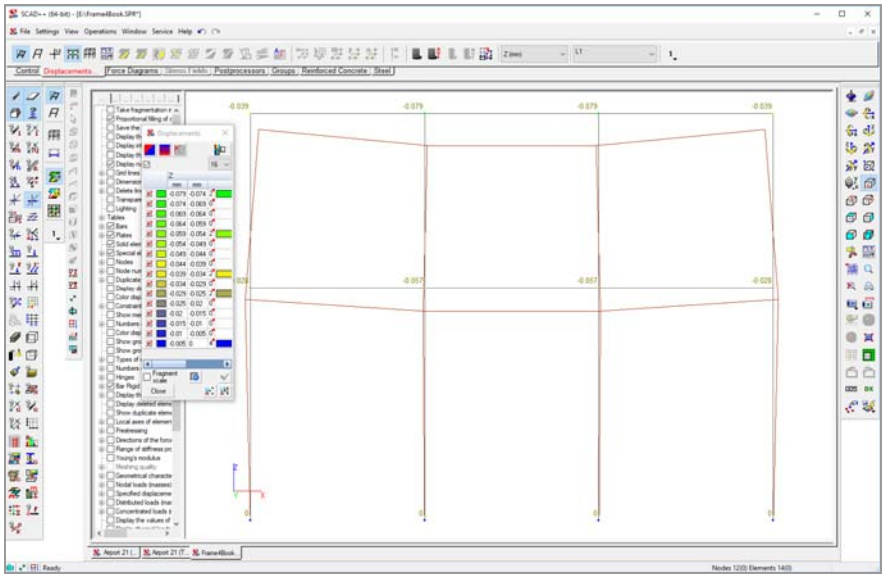


Figure 4.6-1. *Window of* the graphical **postprocessor**

Analysis of Displacements

Analysis of displacements is performed with the help of operations of the respective section (Fig. 4.6-2) of the toolbar. In order to do it, perform the following actions:

- activate the mode of the analysis of displacements by clicking the tab;
- select the analyzed loading from the **Select loading** list;
- select the direction of the displacement from the **Type of displacements** list;
- click the button invoking the operation of display of the calculation results (for example, **Display the initial model together with the deformed one** — Fig. 4.6-1).

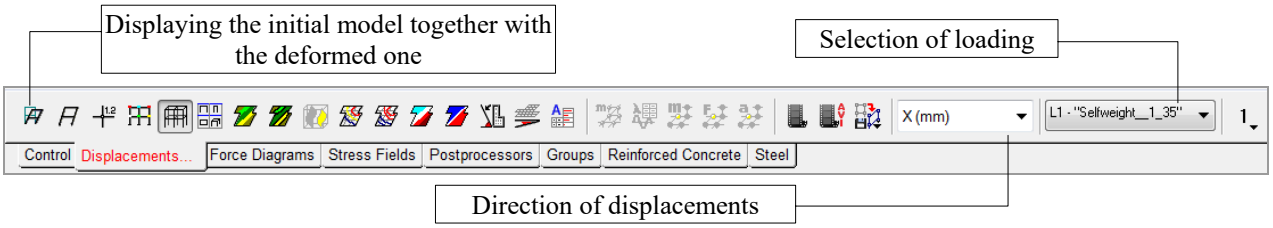


Figure 4.6-2. *Toolbar of the analysis of displacements mode*

The set of display operations enables to obtain different representations of the results of the calculation of displacements. Each representation has a corresponding button in the toolbar. You can use the following buttons when analyzing displacements from static loads in bar structures:

— displays the initial model together with the deformed one;

— displays the deformed model;

— outputs values of displacements in nodes;

— color indication of the displacement value in nodes;

— displays the model.

A deformed model can be displayed together with one of the types of color indication, for example, an indication in the form of color markers in the nodes of the original model and the deformed state of the structure are shown in Fig. 4.6-1.

Analysis of Forces

Since regardless of the mode the consistency of control functions is maintained in the software, the analysis of force factors (in this case – the generation of diagrams) is performed according to the same rules as the analysis of the displacements.

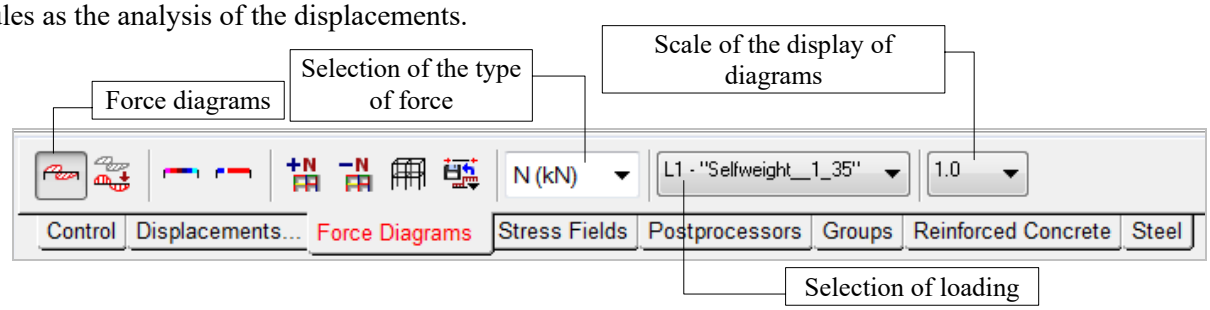


Figure 4.6-3. *Toolbar of the analysis of forces mode*

You can use the following operations in the respective toolbar (Fig. 4.6-3) to display the force diagrams:

— force diagrams;

— color indication of the positive values of forces;

— color indication of the negative values of forces;

— displays the model.

An example of display of forces in frame elements using various means (diagrams and color indication) and in various windows is given in Fig. 4.6-4. New windows can be created with the help of the respective command in the ***Window*** menu. The rules on using the multi-window mode are given in the respective section of this book.

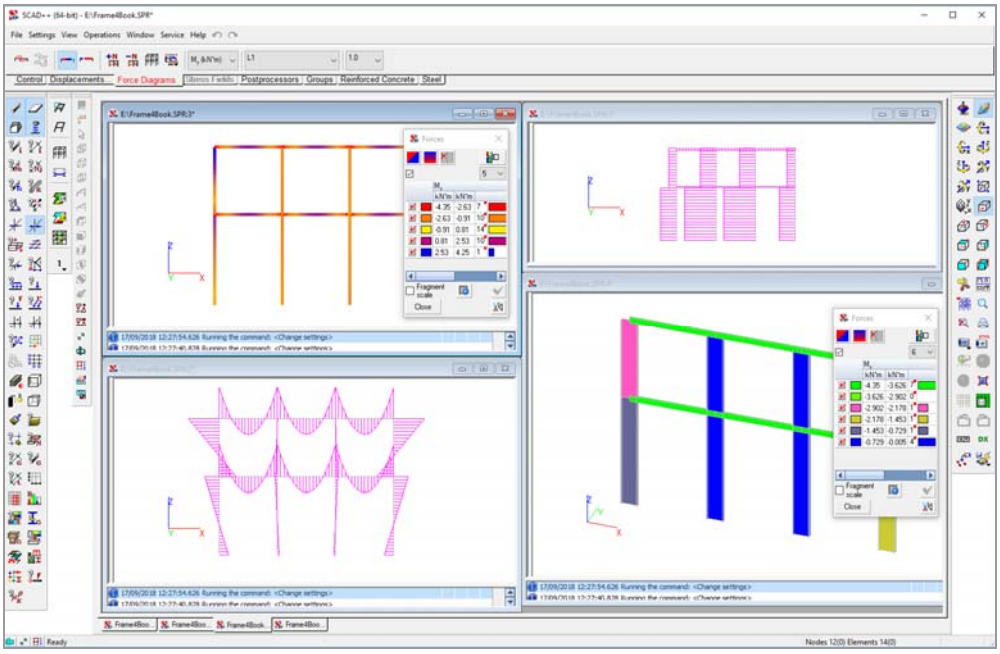


Figure 4.6-4. *Example of display of force factors in the multi-window mode*

4.7. Creating a Design Model from 2D Elements

This section provides an example of the generation of a design model from shell elements. Let's consider a horizontal rectangular shell, supported on its three sides (Fig. 4.7-1).

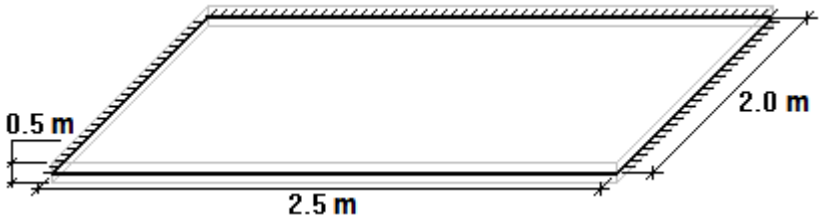


Figure 4.7-1.

Similarly to the previous example, a parametric model invoked by clicking the button — **Generate a rectangular mesh of elements on the plane**, in the **Model** section of the toolbar (Fig. 4.7-2), is used when creating the model.

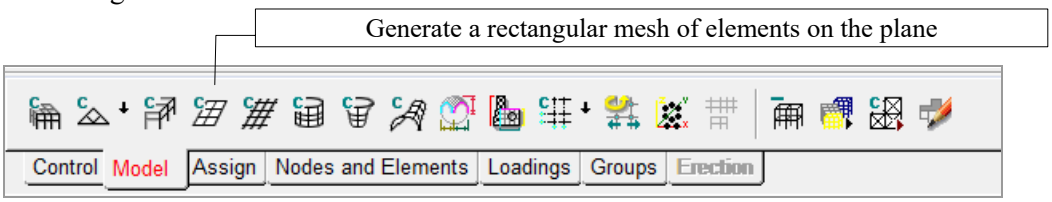


Figure 4.7-2.

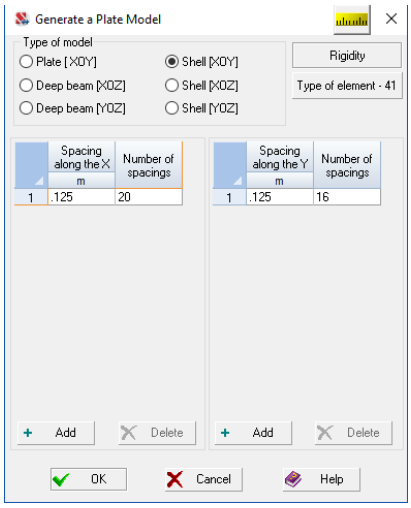


Figure 4.7-3. *The* **Generate a Plate Model** *dialog box*

The **Generate a Plate Model** dialog box (Fig. 4.7-3) is used to specify the parameters of the model which include:

- type and position of the model in space — select **Shell [XOY]**;
- spacing of finite elements along the X axis and the number of spacings — specify the spacing of 0.125 m and the number of spacings — 20;
- spacing of finite elements along the other axis depending on the position of the model (in our case — the Y axis), and the number of spacings — specify the spacing of 0.125 m and the number of spacings — 16.

This dialog box can be also used to invoke operations to specify the rigidities of finite elements (FE) and to select their type (the **Rigidities of Plates** (Fig. 4.7-4) and **Specify the Element Type** (Fig. 4.7-5) dialog boxes respectively) by clicking the respective buttons.

In order to specify the stiffness properties of shell elements you just have to select the name of the material from the list in the **Material** group and specify the thickness of the plates (for example, concrete B20 and 0.3 m) in the **Rigidities of Plates** dialog box. The element type, for example, FE 44 (quadrangular shell FE) is selected from the list of the **Specify the Element Type** dialog box.

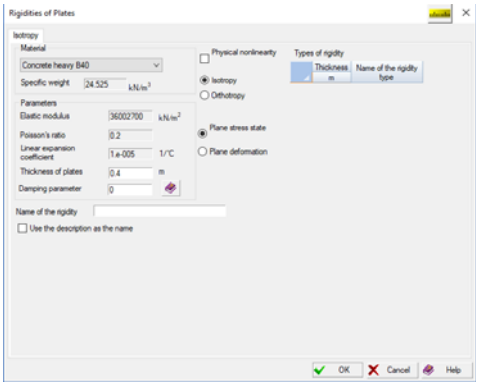


Figure 4.7-4. *The* **Rigidities of Plates** *dialog box*

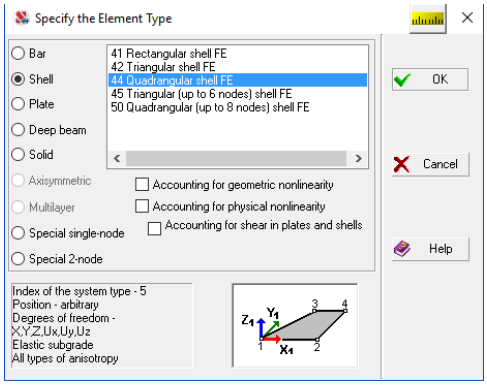


Figure 4.7-5. *The* **Specify the Element Type** *dialog box*

After specifying all the parameters and closing the **Generate a Plate Model** dialog box (the **OK** button) a design model of a rectangular shell from square 4-node finite elements will be generated (Fig. 4.7-6).

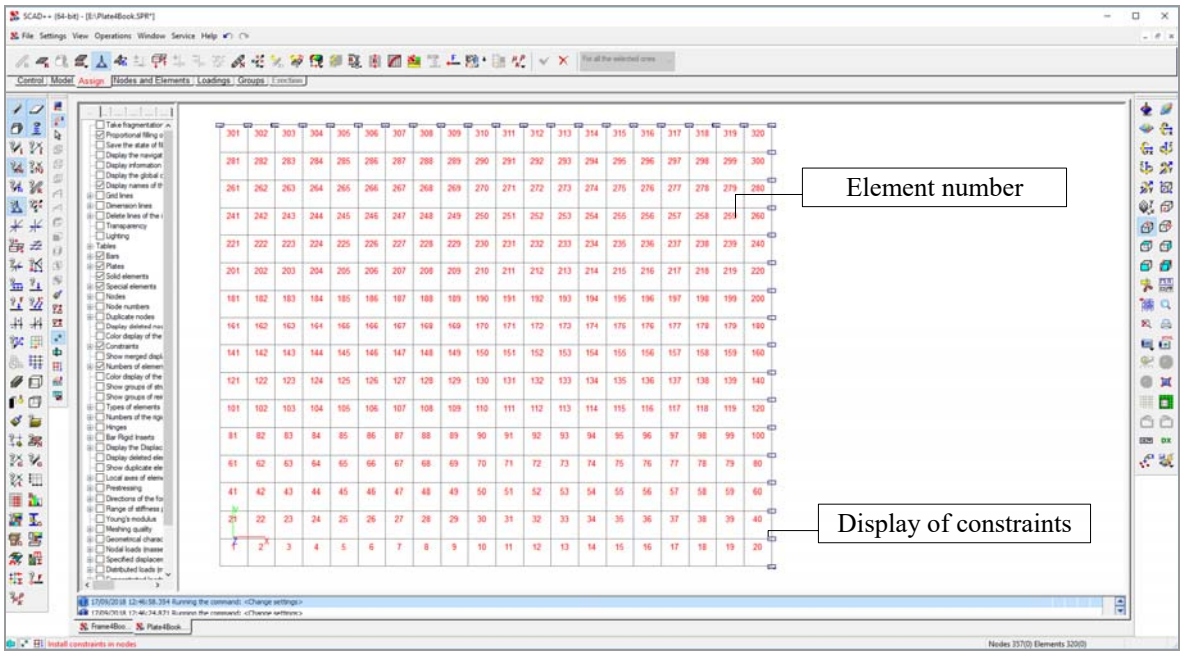


Figure 4.7-6. *Design model of the shell*

Specification of the Boundary Conditions

Let's assume that the model is rigidly clamped on three sides. Such boundary conditions can be applied by installing constraints in nodes along the contour.

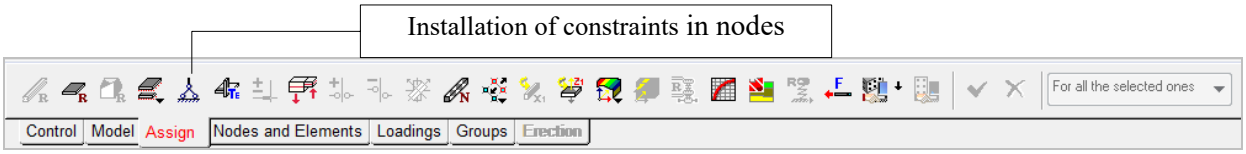


Figure 4.7-7. *The* **Assign** *toolbar*

Constraints can be installed with the help of the respective operation in the **Assign** section of the toolbar (Fig. 4.7-7). Clicking the **Install constraints in nodes** button invokes the **Constraints** dialog box, where you can specify the directions of displacements on which the constraints are imposed by checking the checkboxes (Fig. 4.7-8). The following procedure is standard – close the dialog box (the **OK** button), select the nodes in which the constraints are installed by one of the cursors, and click the **OK** button in the **Assign** toolbar or press the **Enter** key on the keyboard.

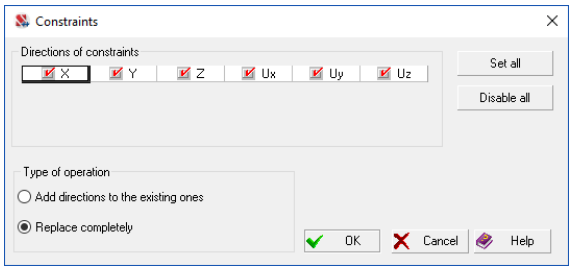


Figure 4.7-8. *The* **Constraints** *dialog box*

Specification of Loads

As in the previous example, the control of the input of loads and creation of loadings is performed from the **Loadings** toolbar.

When performing the calculation it is assumed by default that the design values of loads are given, i.e. characteristic values are multiplied by the safety factor for load.

Let's take the self-weight of the slab as the first loading. The respective operation in the toolbar is used to specify it (Fig. 4.7-9). Clicking the **Self-weight** button invokes the dialog box, where you can specify the factor with which the load from the self-weight will be included in the loading (Fig. 4.7-10).

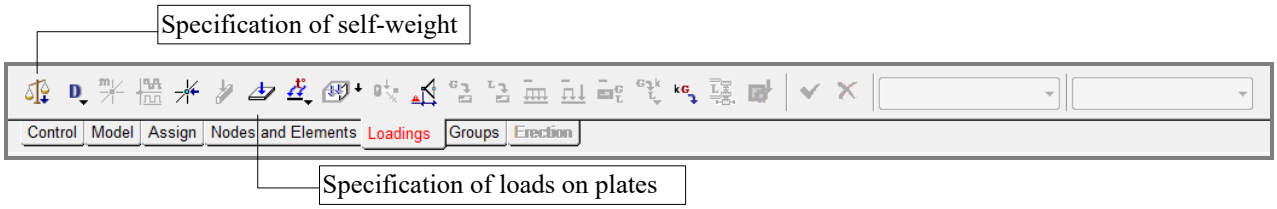


Figure 4.7-9. *The* **Loadings** *toolbar*

If you want the self-weight to be specified automatically, before activating this operation you have to make sure that the stiffness properties of elements, including the description of the section (for bars) or the thickness of the element (for plates and shells), as well as the characteristics of the material (specific weight) are specified.

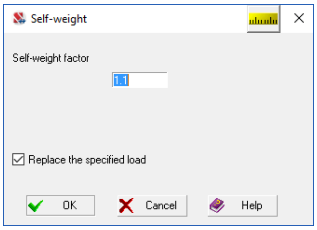


Figure 4.7-10. *The* **Self-weight** *dialog box*

The load will be specified for the elements of the model after closing the dialog box (the **OK** button). As mentioned above, in order to save the loading you have to click the **Save/Add loading** button — in the toolbar, and the **Save Loading** dialog box will appear (Fig. 4.4-15), where you can specify the type of loading and the type of load, and save the loading in the project.

In order to prepare the model for creating a new loading, it is necessary to remove all the specified loads from it.

Let's apply a local load (square 0.5m × 0.5m) in the center of the slab in the second loading. To do this, perform the following actions:

- click the **Loads on plates** button in the **Loadings** toolbar (Fig. 4.7-9);
- check the **Distributed** checkbox in the **Type of load** group in the **Specify Loads on Plate Elements** dialog box (Fig. 4.7-11);
- enable the Z marker in the **Direction of the load** group;
- specify the load, for example, 50 t/m^2 in the **Load value** group;
- select the elements in the model for which the load is applied by one of the cursors (the selected elements will be displayed in red);
- click the button in the toolbar of the **Loadings** mode or press the **Enter** key on the keyboard (if you activate the **Distributed loads** filter, the specified load will be shown on the design model — Fig. 4.7-12);
- save the loading (see the description of the first example).

Figure 4.7-11. *The* **Specify Loads on Plate Elements** *dialog box*

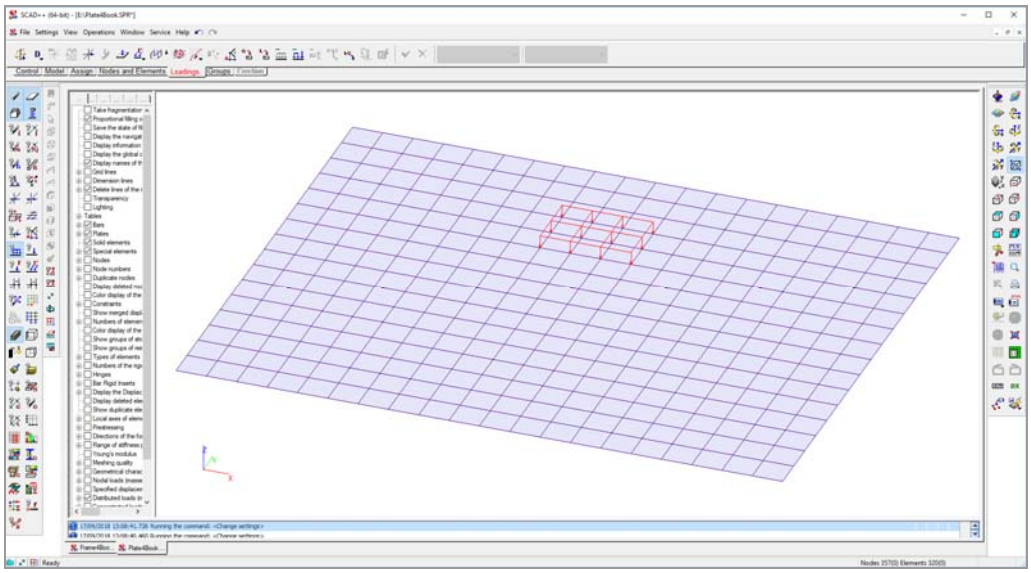


Figure 4.7-12. *Load on the slab*

Graphical Analysis of the Calculation Results

Having performed the calculation according to the rules given in the description of the previous example, switch to the mode of the graphical analysis of the results. The following types of representation of the results are available for the models from 2D elements (plates and shells): a deformed model, color markers of displacements and force factors, isolines and isofields of displacements and force factors (Fig. 4.7-13).

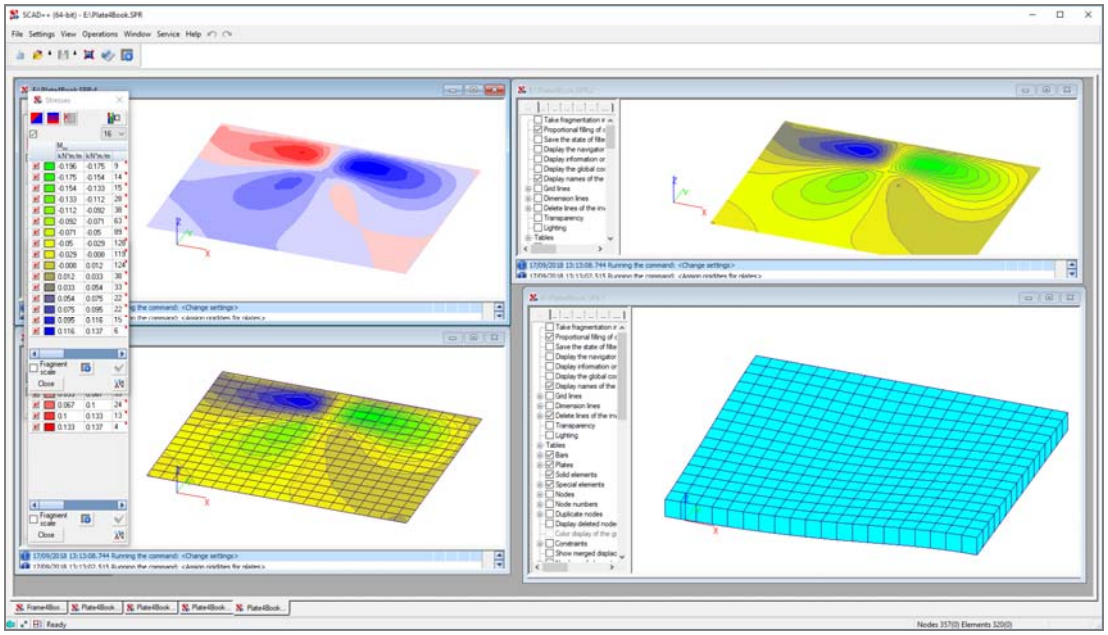


Figure 4.7-13. *Different types of representation of the results of calculation of 2D elements*

Unlike bar structures for which the color scale of displacements can only be represented by color markers in nodes, the isofields and isolines can be plotted in addition to the markers for structures from two-dimensional elements. The control of the output of isofields and isolines of displacements is performed in the **Displacements** toolbar (Fig. 4.6-2). To do this, use the following buttons:

— display isofields of displacements;

— display isofields and isolines of displacements;

— display isofields of displacements with a constant (specified) step;

— display isolines of displacements;

— display isolines of displacements with a constant (specified) step.

The control of representation of the results of calculation of the force factors for 2D elements is performed in the **Stress Fields** toolbar (Fig. 4.7-14). To do this, use the following buttons:

— display isofields of stresses;

— display isofields and isolines of stresses;

— display isofields of stresses with a constant (specified) step;

— display isolines of stresses;

— display isolines of stresses with a constant (specified) step;

— plot stress diagrams along a secant line.

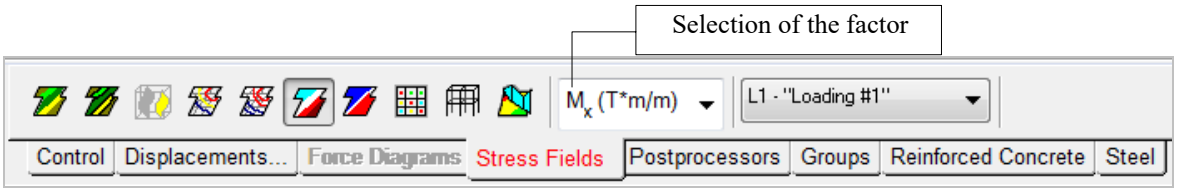


Figure 4.7-14. *The* **Stress Fields** *toolbar*

4.8. THUS …

The Red Queen shook her head,
"You may call it nonsense if you like," she said, "but I've heard nonsense, compared with which that would be as sensible as a dictionary!"

L.CARROLL "Alice in Wonderland"

… simple and as a rule the most used in practice techniques of working with the software have been considered in the **Structure CAD for DUMMIES** chapter. A sequence of actions from creating a design model to the analysis of the calculation results is illustrated on the example of the simplest problems. Despite the minimum of the used functions, it reflects the general technology of working with the program.

The following basic requirements to a design model and the principles of control of the software can be formulated for novice users:

1. The obligatory initial data (the minimum requirement for performing the calculation) include: geometry of the design model (description of nodes and elements), stiffness properties of the elements, description of the boundary conditions (e.g. constraints) and loads.
2. Any function of the software can be activated from the **Project tree**.
3. Most operations of creating a design model can be performed by the following sequence of actions:
   - activate the respective section in the toolbar by clicking on its tab;
   - click the button with the icon of the performed operation;
   - if additional information is required to perform the operation, specify the necessary data in the invoked dialog box and close it by clicking the **OK** button;
   - select (by the cursor) the respective nodes and elements (they will be displayed in red);
   - click the **OK** button (**Confirmation**) — in the section of the **Toolbar** the operation has been selected from or press the **Enter** key on the keyboard (**Enter** and **Esc** keys are called "hot", they can be used instead of the **Confirmation** and **Rejection** buttons of the toolbar);
   - if the operation has been completed successfully, the elements or the nodes involved in the operation will be displayed again in the color of the design model; otherwise a message with the reasons of the rejection of the operation will appear.
4. A system of display filters is used to reflect the results of the operations on the model.
5. The calculation will be available in the project tree, if the prepared data on the design model are sufficient to perform it.
6. The calculation progress is provided in the log, which contains information on the performed actions, and on the warnings and errors detected during the calculation.
7. If the analysis has been completed successfully (in this case we mean only that the files with the calculation results have been generated, irrespective of the quality of the results), the **Graphical Analysis** function will be available in the **Results** section of the project tree.
8. The following procedure is recommended for the graphical analysis of the results:
   - select a section in the toolbar with the information you are interested in by clicking on its tab (**Displacements**, **Force Diagrams**, **Stress Fields**);
   - select a loading from the list of loadings;
   - specify the analyzed factor in the list of factors (direction of displacements or the type of the force factor);
   - click the button corresponding to the selected representation of the result (deformed model, diagrams, isofields, etc.).
9. In order to compare different results, for example, displacements from different loadings, you can display different analyzed factors in separate windows.

5. Creating a Design Model

The **Model** section of the toolbar contains operations which enable to create a design model or its part on the basis of parametric structural prototypes. Moreover, the operations of this section are used for the modification of the model using the mechanisms of affine transformations, copying, assembling, etc.

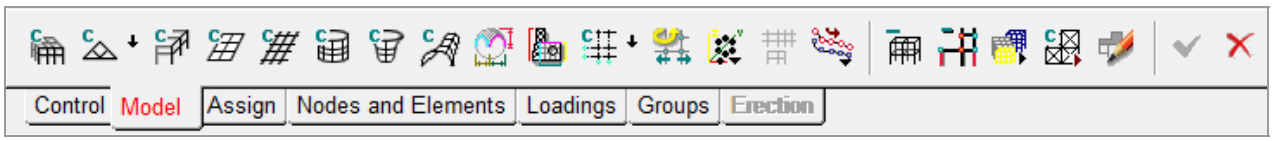


Figure 5-1. *The* **Model** *section of the preprocessor toolbar*

The following operations can be invoked from the **Model** section:

— generate a frame prototype;

— generate a truss prototype;

— duplicate the frame along the axis;

— generate a rectangular mesh of elements on the plane;

— generate a grade beam;

— create surfaces of revolution;

— create surfaces of revolution by the specified formula;

— create surfaces by the specified formula;

— create a standard analytic surface;

— create standard structures;

— specify grid lines;

— geometric transformations;

— copy the model;

— copy the fragment of the model;

— creation and modifications of **trajectories**;

— delete model;

— delete selected objects;

— assembling a model from several models (**Assembling mode**);

— generate a mesh of an arbitrary shape on the plane;

— invoke a graphics editor.

Since most operations of the **Model** section are performed immediately after specifying the parameters in the dialog boxes and do not require special confirmation, this section of the toolbar does not contain the **OK** — and **Cancel** — buttons.

5.1. Design Models of Frame Bar Structures

Generate a Frame Prototype Operation

This operation enables to create the following types of bar frame structures on the basis of parametric prototypes – multi-storey multi-span frames, single-storey multi-span frames of industrial buildings, and single-span variable I-section frames. A frame configuration can be selected in the **Select Frame Configuration** dialog box (Fig. 5.1-1) by clicking on the respective icon.

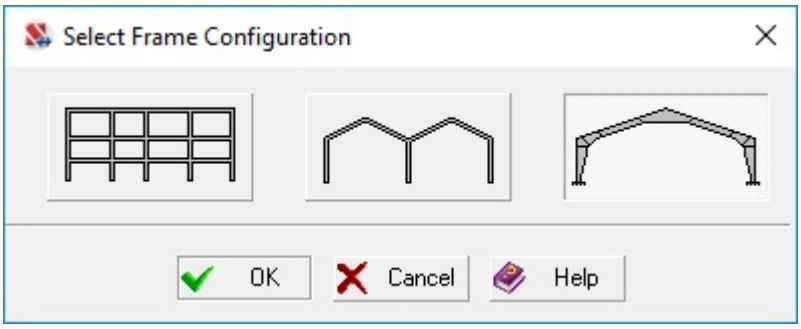


Figure 5.1-1. *The* **Select Frame Configuration** *dialog box*

Multi-Storey Multi-Span Frames

Data necessary to create a model of a multi-storey multi-span frame is input in the **Specify the Parameters of a Regular Frame** dialog box (Fig. 5.1-3). This dialog box is used to specify sizes of the frame, rigidities of the elements, and types of finite elements, and to impose constraints in nodes. Two tables are used to input the sizes: the left one — to describe the spans, and the right one — to describe the storeys.

The checked **Automatic installation of constraints** checkbox means that constraints in the directions corresponding to the active (pressed) marker buttons with the name of the degrees of freedom will be installed at the base of the columns of the generated frame. Markers corresponding to the degrees of freedom of the type of model selected for the problem are activated by default.

Columns and **Girders** buttons are used to specify the stiffness properties for the respective elements of the model, and the **Type of element** button is used to specify the types of finite elements. These data are specified with the help of the respective operations in the **Assign** toolbar and are described in Section 7 (Specifying Characteristics of Nodes and Elements).

By default, the frame will be located in the XOZ plane. Checkbox **Along the Y axis** allow to place the structure in the plane YOZ.

The program has almost no restrictions on the number of specified spans and storeys (in this case “almost” means 32000). Moreover, the degrees of freedom opened in this dialog box by default the constraints are imposed on are limited by six. If more degrees of freedom are used in the created bar model (for example, index of the system type 8 or 9), the missing constraints in nodes have to be specified using the respective operation from the **Assign** section of the toolbar.

In those cases when the sections of individual girders or columns differ from those specified when creating a frame (for example, some girders have a different section), you should use the **Assign rigidities for bars** operation — from the **Assign** section of the toolbar and specify all the necessary characteristics for these elements. However, in order to generate the geometry of a design model it is usually enough to fill only the data in the tables.

Let’s consider a frame (Fig. 5.1-2) with five spans (2x6 m, 3 m, 2x4.5 m) and 10 storeys (4 m, 8x3.3 m, 2.5 m) as an example of a multi-storey multi-span frame which will be used to describe the technology of creating a design model.

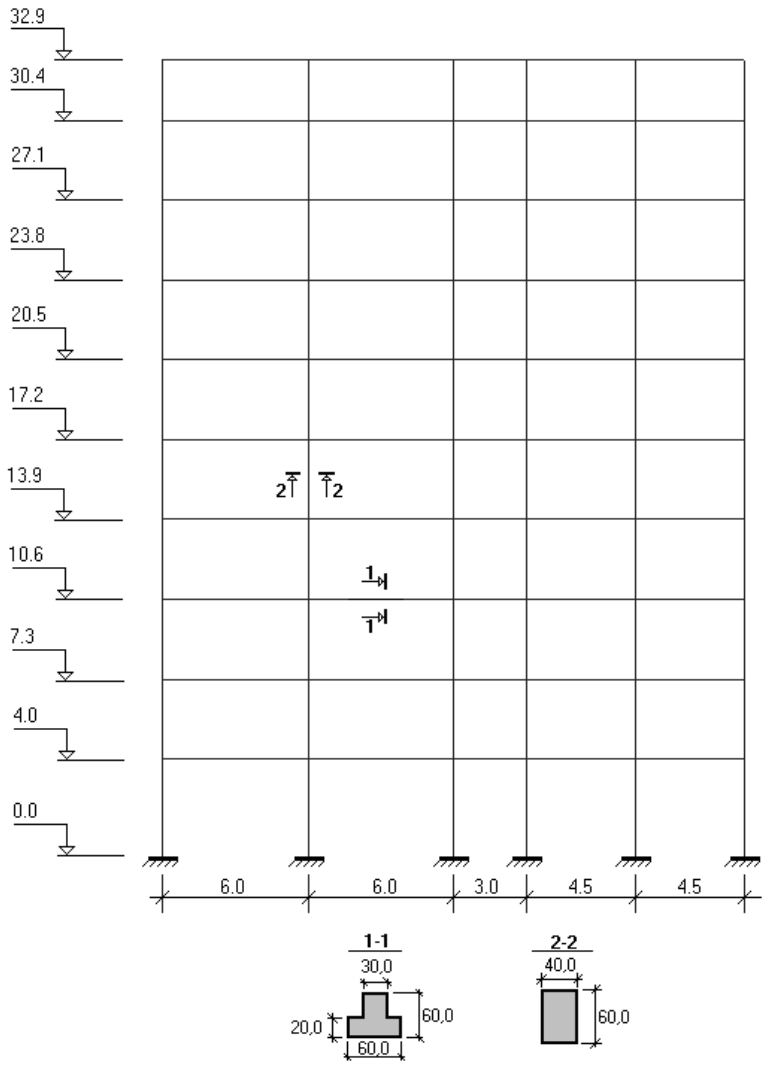


Figure 5.1-2.

A dialog box with the initial data for the generation of this frame is shown in figure 5.1-3.

If all the necessary parameters of the frame are determined, then after clicking the **OK** button the model will be generated (the result for the above example is shown in Fig. 5.1-4).

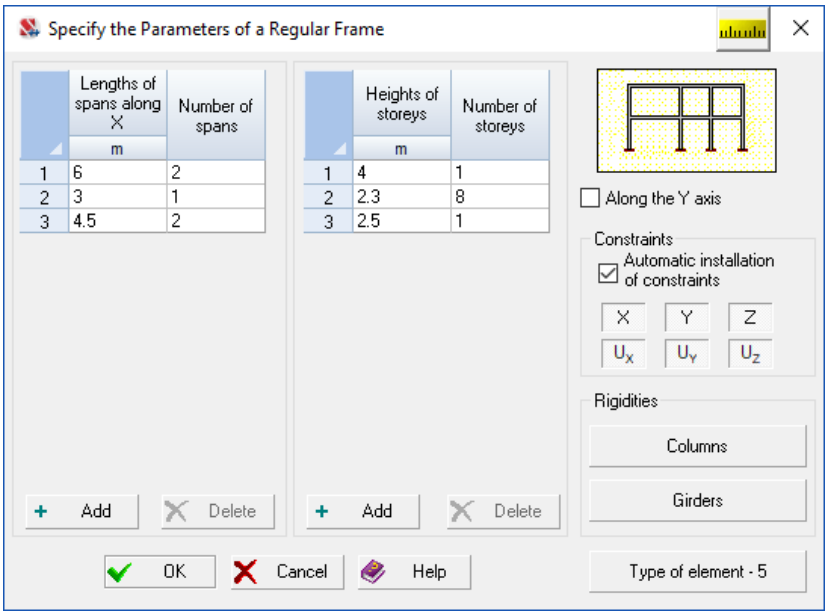


Figure 5.1-3. *The* **Specify the Parameters of a Regular Frame** *dialog box*

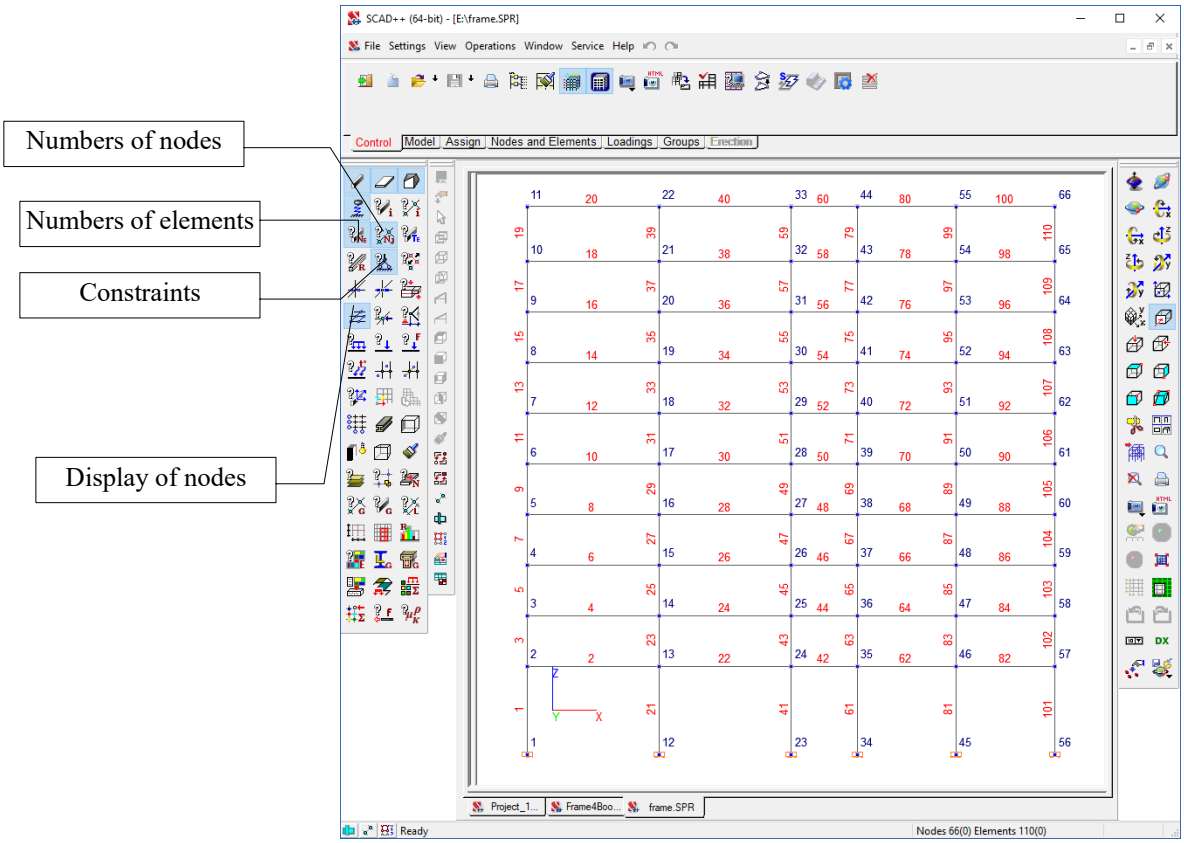


Figure 5.1-4. *The design model of the multi-storey multi-span frame*

Single-Storey Multi-Span Frames of Industrial Buildings

This parametric prototype enables to generate a design model of a single- or multi-span single-storey frame with different configuration of a girder (dual-pitched symmetric, dual-pitched asymmetric, mono-pitched), and also to take into account the presence of crane beams. Initial data necessary to generate a frame are specified in the **Single-Storey Frame** dialog box (Fig. 5.1-6) and include:

- lengths of spans and their number (are specified in the table);

- column height — H;
- girder slope — p (in percent, is assumed the same in all spans);
- position of the ridge joint (is defined by the ratio of the distance from the joint to the left column — L_c to the length of the span — L_p).

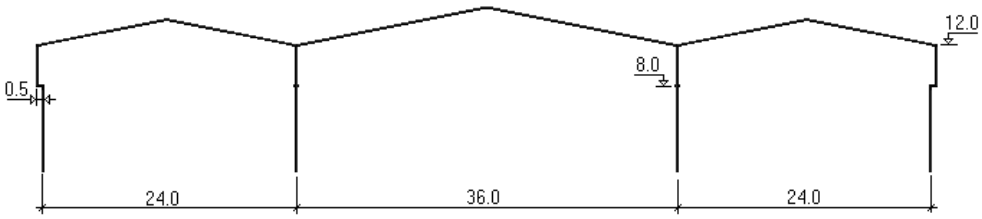
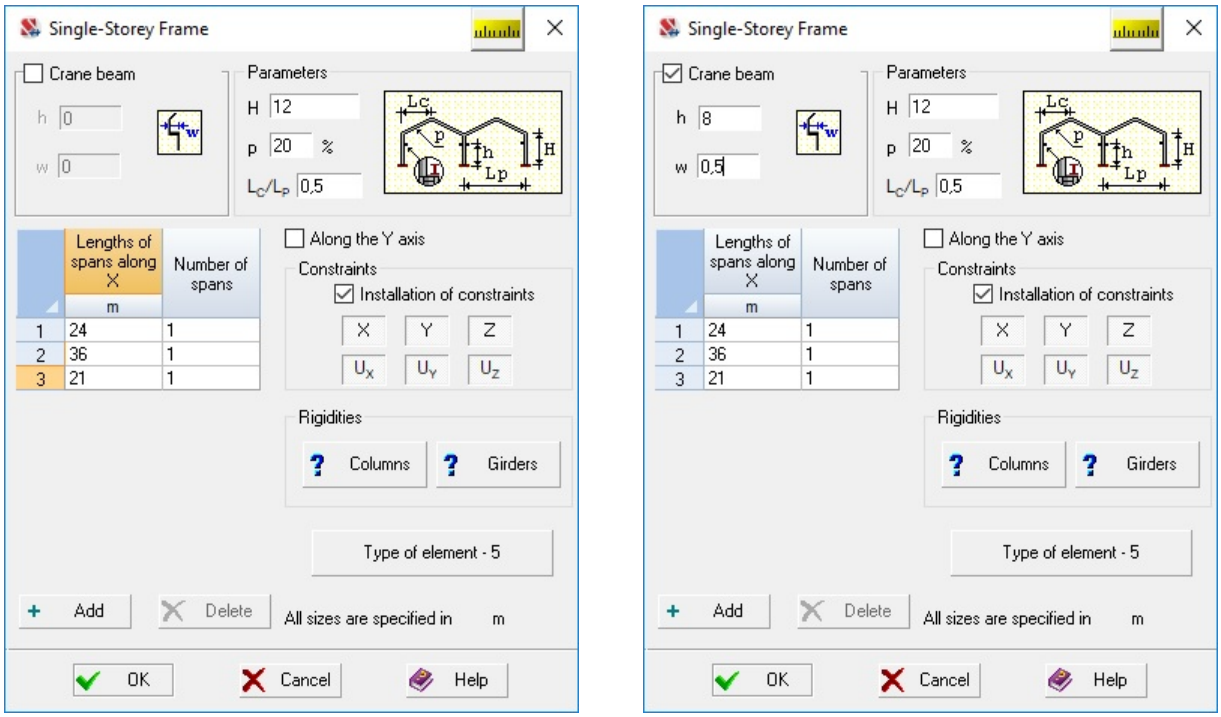


Figure 5.1-5. *Single-storey multi-span frame*



a) *b*)

Figure 5.1-6. *The* **Single-Storey Frame** *dialog box with the initial data necessary to create a design model: a) without a crane beam and b) with a crane beam*

Moreover, the dialog box enables to:

- automatically install the constraints in the column bases (if the **Installation of constraints** checkbox is checked) and specify the directions of the degrees of freedom of displacements the constraints are imposed on;
- model structures of crane beams (if the **Crane beam** checkbox is checked) and specify elevation — h and horizontal location — w of the supports of crane beams (Fig. 5.1-6,b);
- specify the stiffness properties of columns and girders — the respective buttons are used (it is assumed that all girders have the same section and all columns have the same section);
- specify the type of finite elements (it corresponds to the type of the model by default, i.e., if type 2 is selected in the identification data of the project, then the elements will be of type 2, if the model is of type 5, then the elements will be of type 5 as well).

By default, the frame will be located in the XOZ plane. Checkbox **Along the Y axis** allow to place the structure in the plane YOZ.

Let's consider a frame with dual-pitched symmetric girders as an example of a single-storey multi-span frame (Fig. 5.1-5). It has three spans (24 m, 36 m and 24 m) and the column height of 12 m, girder

slope – 20%, elevation of the crane beam – 8 m and the offset value of the axes of the part under the crane with respect to that above the crane — 0.5 m.

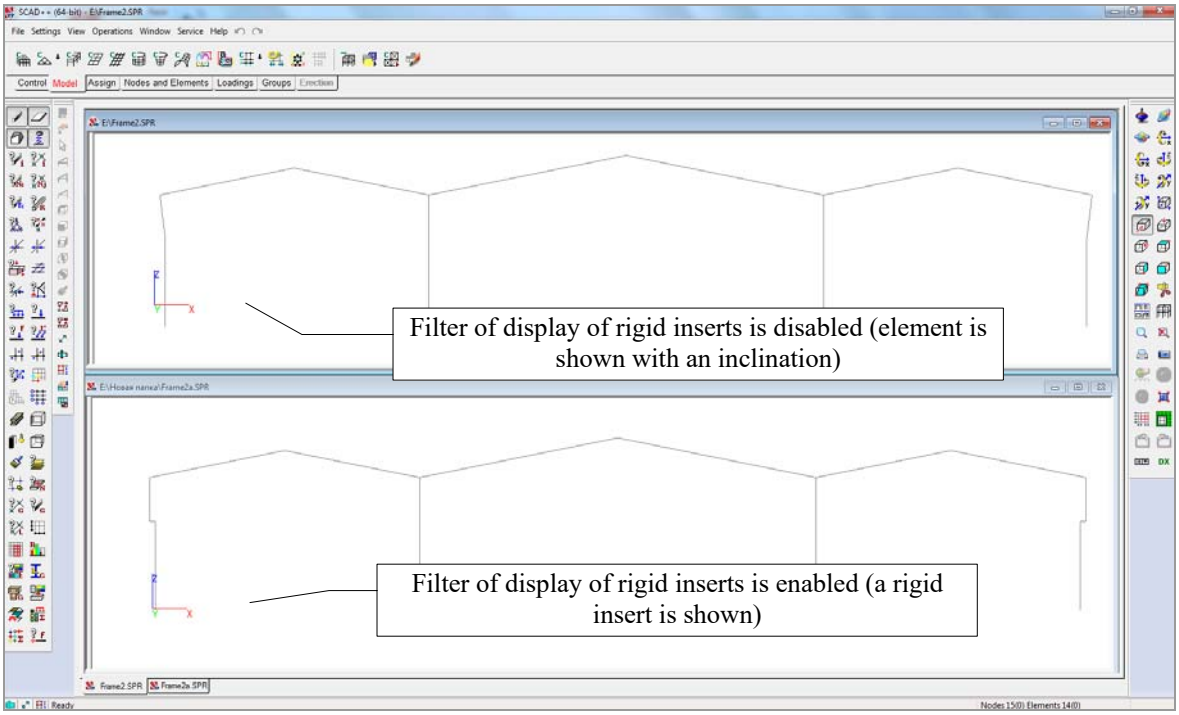


Figure 5.1-7. Display of rigid inserts on the design model

In order to model the bearing of crane beams on the edge columns, an offset of the axes of the column part under the crane with respect to that above the crane using rigid inserts is provided in the design model. Intermediate nodes are created in the middle columns and the user has to select the modeling method for the bearing of crane beams (creating cantilevers, applying loads to the central node, etc.). To display the rigid inserts on the model use the respective filter button . Since in this case the rigid inserts are created in the direction of the local Z_1 axis of the element, then when the filter button is disabled the elements modeling the column part above the crane will be displayed at an angle (Fig. 5.1-7).

You can obtain different configurations of a girder by changing the position of the ridge joint. If $L_c/L_p = 0.5$, the ridge joint will be in the middle of the span, i.e. a model with a symmetric dual-pitched girder will be generated. If $L_c/L_p = 1$ — the girder will be mono-pitched with a ridge joint near the right column, and if $L_c/L_p = 0$ — near the left column. For values $0 < L_c/L_p < 1$, different from 0.5, a model with an asymmetric dual-pitched girder will be generated.

It should be noted that the parameters specified in the **Single-Storey Frame** dialog box are the same for all spans. If some spans are different from the others, then after the generation of the model you should use the respective operations and change their characteristics (for example, to change the sections of elements or their orientation angle – use the respective operations of the **Assign** toolbar, and to change the geometry of the frame use the operations of the **Nodes and Elements** toolbar).

Single-span Variable I-section Frames

The parametric model enables to create a design model of a single-span frame from welded I-beams of variable cross-section with a dual-pitched girder. Frames with the following boundary conditions of columns and girders are considered:

- pinned-end columns, rigid beam-to-column and ridge joints;
- pinned-end columns, rigid beam-to-column joints and pinned ridge joint;
- clamped columns, rigid beam-to-column and ridge joints;
- clamped columns, pinned beam-to-column joints and rigid ridge joint;
- all the joints are rigid.

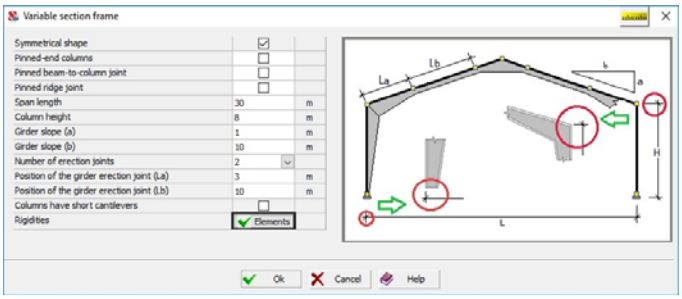


Figure 5.1-8. *The* **Variable Section Frame** *dialog box*

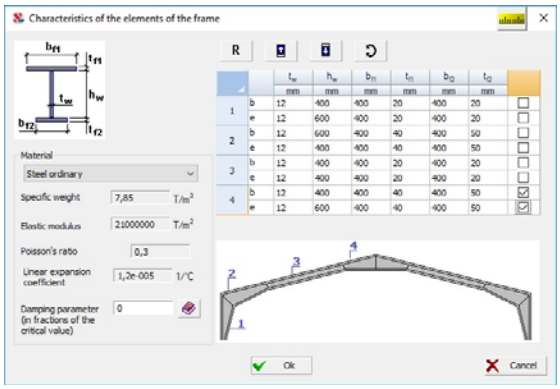


Figure 5.1-9. *The* **Characteristics of Frame Elements** *dialog box*

The girders can have intermediate erection joints (in addition to the ridge) and consist of a combination of elements of variable and constant cross-section. Columns can be made of welded I-beams of both constant and variable cross-section. Initial data necessary to generate a design model of a frame are specified in the **Variable Section Frame** dialog box (Fig. 1) and include:

- column boundary conditions (pinned-end/clamped);
- type of beam-to-column joint (pinned/rigid);
- type of ridge joint (pinned/rigid);
- frame span;
- column height (eaves level);
- girder slope (defined by parameters *a* and *b*);
- number of erection joints within half the frame span (0, 1, 2);
- distance from the eaves joint to the first erection joint (if there is one);
- distance between erection joints (if there are two joints);
- the presence of short cantilevers;
- distance from the column base to the short cantilever (if there is a cantilever).

The characteristics of columns and girder elements are specified depending on the selected type of frame in the **Characteristics of Frame Elements** dialog box (Fig. 5.1-9). The dialog box is invoked by clicking the **Elements** button, and includes the description of the material (steel) and cross-sectional dimensions of elements at the beginning and end of each structural member. The rules for specifying sections are as follows:

- the first node is considered the "beginning" of the element (the origin of the local coordinate system of the bar);
- cross-sectional dimensions of the column (S1) and the girder (S2) are defined by the boundary conditions at the ridge joint (Fig. 5.1-10);
- webs must have the same height at the beam-to-beam joints;
- the upper edges of the webs of the girder lie on one straight line, the angle of which is defined by the slope value;
- regardless of the flange thickness of different parts of the girder (when there are erection joints) the constant slope along the upper edge of the flanges is provided automatically by introducing rigid inserts;
- only columns of constant cross-section can have short cantilevers.

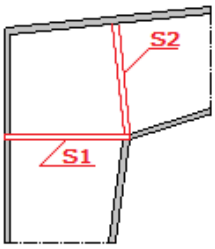


Figure 5.1-10

Cross-sectional dimensions are specified in tabular form. Control of the table is performed with the help of a set of buttons which enable to copy a line or to use the previously specified stiffness properties of the elements. The rules for working with the control buttons are as follows:

- clicking the "Down" button enables to copy the data from the first selected line into the lower ones;

- clicking the “Up” button enables to copy the data from the last selected line into the upper ones;
- clicking the “Swap” button enables to swap the contents of two selected lines;
- pressing the **R** (Read) button invokes the **Rigidity of Bar Elements** dialog box and enables you to access previously specified stiffness properties of the elements. The lines selected in the table of frame element sections will be filled with the cross-sectional dimensions corresponding to the type of rigidity selected in the table in this dialog box (after closing the dialog box by clicking the **OK** button).

You should keep in mind that only the stiffness properties of variable cross-section bars can be transferred into the table.

5.2. Creating Models of Plane Hinged-Bar Systems

A library of parametric prototypes available in the software can be used to create a design model of a truss. A truss prototype is selected according to two parameters — configuration of the chords and the type of the lattice.

Clicking on the arrow on the right from the **Generate a truss prototype** button , drops down a menu where you can select the material of the truss (**Generate a steel truss prototype** or **Generate a timber truss prototype**). It is assumed that the steel truss is made from rolled angles or round and rectangular (square) pipes, and the timber one – from round or rectangular beams. In the case when during the generation the stiffness properties do not have to be assigned to the truss elements, the button is used.

Generating a Steel Truss Prototype

The two-tab **Generate a Truss Prototype** dialog box (Fig. 5.2-1) enables to select the configuration of the truss chords, type of the lattice and to specify the parameters of the truss. Trusses with the following configurations of the chords can be generated in the software:

- parallel chord trusses;
- triangular trusses;
- trapezoid trusses;
- trusses with polygonal top chords;
- one slope roof trusses;
- double slope roof trusses.

The following procedure for specifying the data in this dialog box is recommended:

in the **General** tab:

- select the configuration of the truss chords from the respective list;
- click on the icon with the image of the respective lattice;
- specify the sizes of the truss and the number of elements of the chord in the table (to check the geometry of the truss you can use the **Preview** button — , clicking which invokes a dialog box with a drawing of this truss);
- switch to the **Section** tab (Fig. 5.2-2);

in the **Section** tab:

- select a marker-button with the image of a profile of the truss elements;
- enable a radio button in the **Cross-section** group, for example, **Top chord** (the selected element will be displayed in red in the drawing of the truss);

- select the section of this truss element in the list of assortments of rolled profiles (the name of the selected profile will be shown in the **Profile** table);
- select the thickness of the gusset plate g from the list in the **Profile** table;
- repeat the last three operations for other elements of the truss;
- close the dialog box by clicking the **OK** button.

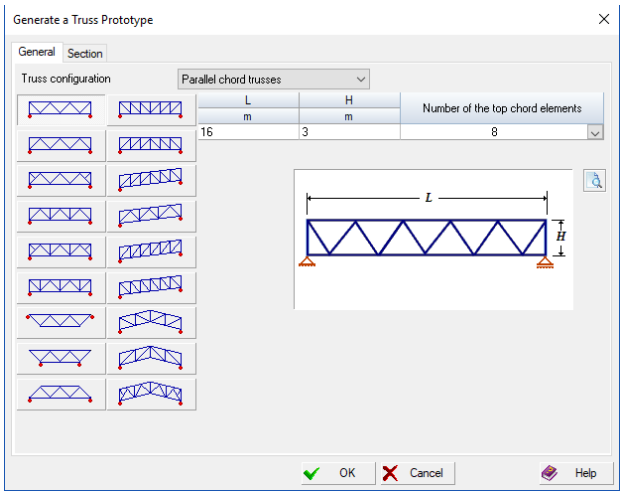


Figure 5.2-1. *The* **General** *tab of the* **Generate a Truss Prototype** *dialog box*

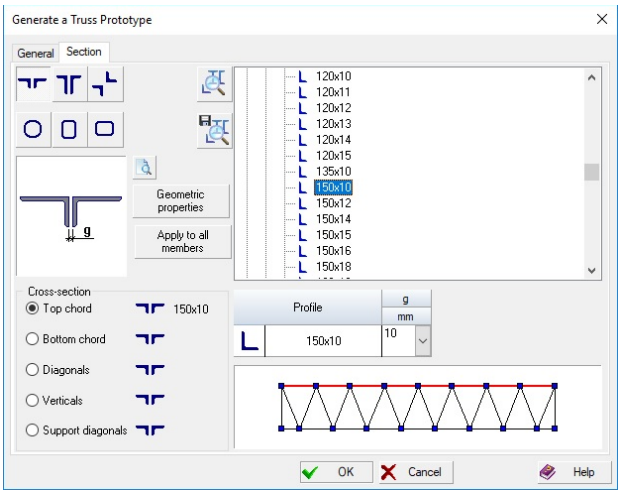


Figure 5.2-2. *The* **Section** *tab of the* **Generate a Truss Prototype** *dialog box*

If the parameters of the truss are specified correctly, its design model will be displayed on the screen (Fig. 5.2-3).

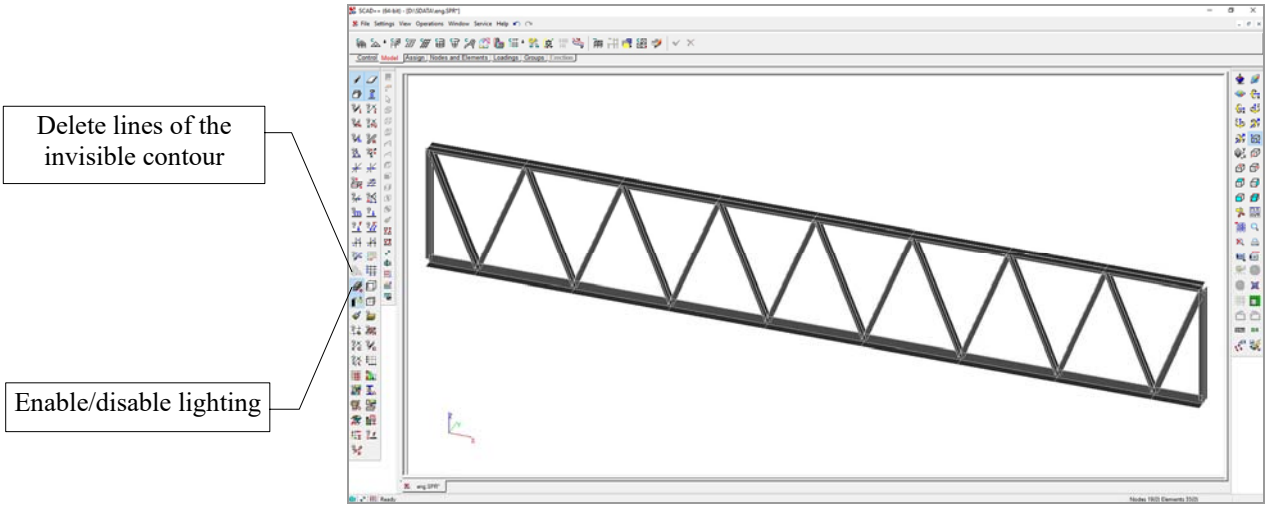


Figure 5.2-3. Design model of the truss

Work in the Section Tab (Additional Features)

Sections of truss elements are selected with the help of the following marker buttons:

— double angles arranged as a tee connected on short legs;

— double angles arranged as a tee connected on long legs;

— equal angles arranged as a cross;

— round pipes;

— rectangular pipes with a long vertical side or square;

— rectangular pipes with a long horizontal side or square.

After selecting the type of section only the items corresponding to the selected section will be available in the lists of assortments of rolled profiles.

In order to remember the frequently used profiles or combinations of double angles the software enables to create a library of custom sections. The buttons **Save cross-section in the user database** — and **Load cross-section from the user database** — enable to use sections created not only in **SCAD**, but also in other applications of the **SCAD Office** system (for example, in **Kristall**). File of the database of custom sections ***UserSectionStorage.uss*** is saved in the **Application Data** section of the **Documents and Settings** system directory.

Clicking the **Preview** button — opens the **Preview** dialog box with the image of the selected section (Fig. 5.2-4).

Clicking the **Geometric properties** button invokes the **Section Properties** dialog box which provides the drawing of the section together with its geometric properties given in the tabular form (Fig. 5.2-5).

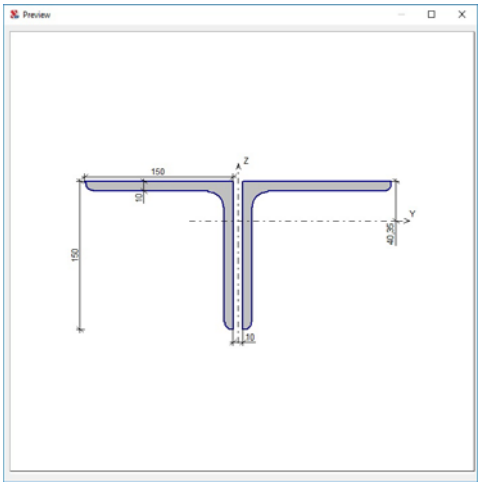


Figure 5.2-4. *The* **Preview** *dialog box*

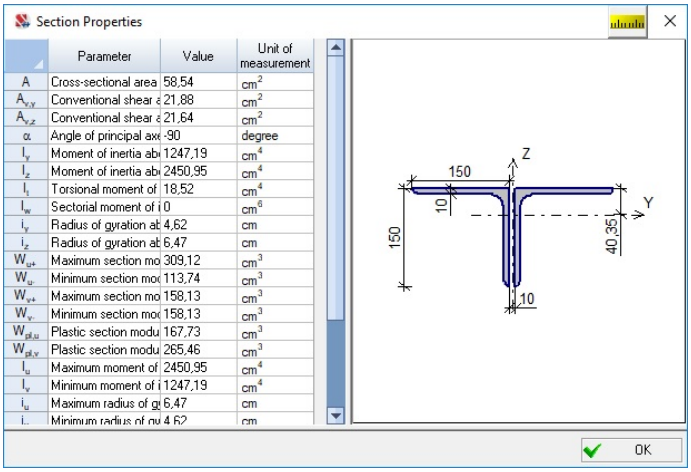


Figure 5.2-5. *The* **Section Properties** *dialog box*

The **Apply to all members** button is used in those cases when all or most of the truss members have the same cross-section. In order to use this button assign a section to one of the elements and without changing the active radio button click the given button. All the members of the model will have the specified section. If the model has members with a different section, it has to be assigned after performing the considered operation.

Generating a Timber Truss Prototype

A timber truss prototype is generated according to the same rules as the steel one. The number of timber truss prototypes is much less and all of them are given in the **General** tab (Fig. 5.2-6).

Two types of sections are provided in the program for the members of timber trusses — rectangular (square) and round from non-glued timber. Controls and the rules of working in the **Section** tab (Fig. 5.2-7) are the same as those considered above for the steel trusses. The elastic modulus and the Poisson's ratio for timber are specified in the stiffness properties of elements according to the selected design codes.

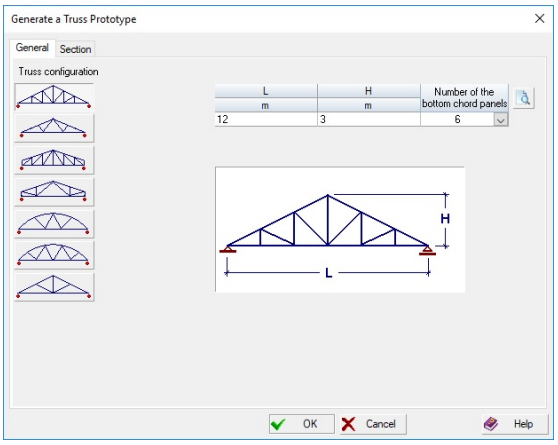


Figure 5.2-6. *The* **General** *tab when generating a timber truss prototype*

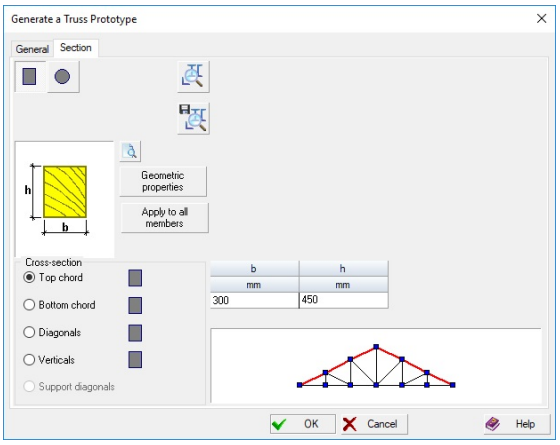


Figure 5.2-7. *The* **Section** *tab when generating a timber truss prototype*

Generating Trusses with Elements with a Custom Section

This mode is used in cases when the sections of elements differ from those provided in the modes of the generation of trusses from rolled steel and timber. In this case only the geometry of the model is generated, and the stiffness properties of the elements are specified after clicking the **Assign rigidities for bars** button — in the **Assign** section of the toolbar. Truss configuration is selected in the **General** tab (Fig. 5.2-1) according to the same rules as for the trusses from rolled steel.

5.3. Spatial Frames and Trusses

In cases when a prototype of a bar system is a plane structure lying in a plane parallel to one of the coordinate planes, it can be used to create a spatial bar system with the help of the **Duplicate along the axis** operation — .

This operation is based on the principle of duplicating a transverse plane frame structure or a truss lying in the plane parallel to one of the coordinate planes with the given step in the direction of the axis selected by the user in the right-hand Cartesian coordinate system XYZ. Longitudinal structures are created by adding bars in the direction of the selected axis in each node of the transverse structure (except for the support nodes of the columns of the frame).

When the operation of duplicating a frame or a truss is invoked, the **Model of the Longitudinal Frame** dialog box appears (Fig. 5.3-1). This dialog box enables to specify the direction of copying, the step and the number of repetitions (the step may be variable), and to specify the stiffness properties of bars created in the direction of the selected axis (the **Profile** button). The sections of all elements of the longitudinal structure are assumed to be the same. Sections of elements of the longitudinal frame can be modified with the help of the **Assign rigidities for bars** operation — in the **Assign** section of the toolbar.

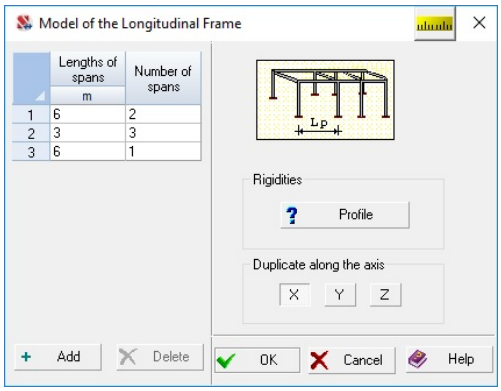


Figure 5.3-1. *The* **Model of the Longitudinal Frame** *dialog box*

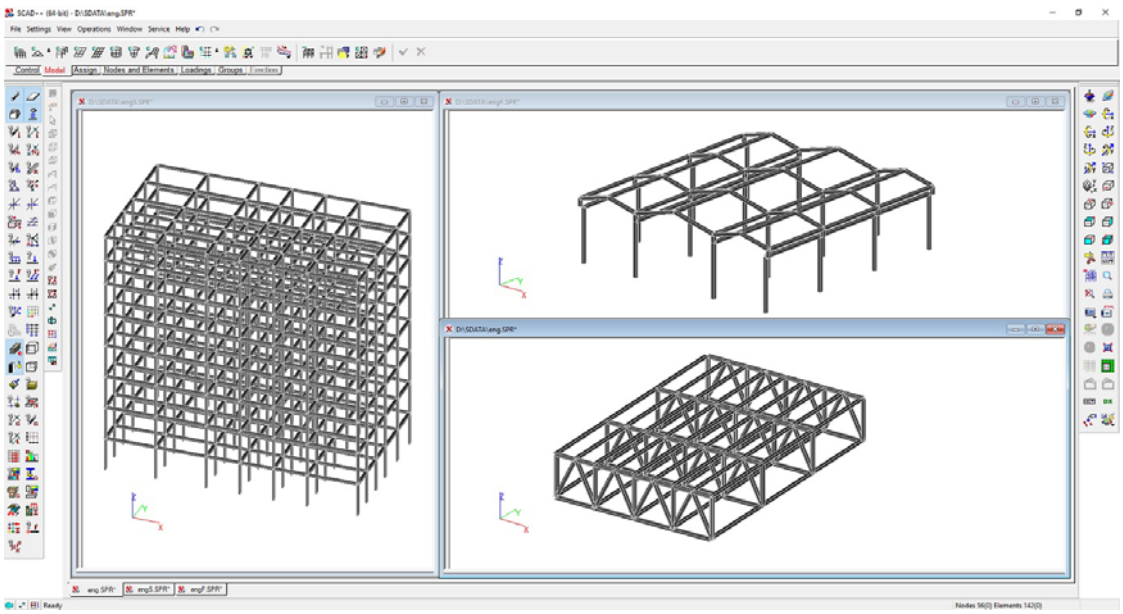


Figure 5.3-2. *Examples of using the* **Duplicate along the axis** *operation for different types of bar systems*

Copy operations considered below are used to duplicate arbitrary bar systems.

Design models of a multi-storey multi-span frame, single-storey frame and a truss considered earlier in the current chapter are given as examples of performing the duplication operation (Fig. 5.3-2).

5.4. Creating a Design Model of a Grade Beam

The respective operation in the **Model** section is provided to generate a design model of a grade beam. Initial data for this type of model are specified in the **Generate a Grade Beam** dialog box (Fig. 5.4-1) in the same way as for the parametric prototypes considered earlier. Spacing and the number of spacings of the grade beam along the X and Y axes respectively are specified in two tables of the dialog box. Stiffness properties of elements (the **Rigidity** button) and their type (the **Type of element** button) are specified similarly to the respective operations when generating parametric prototypes of frames.

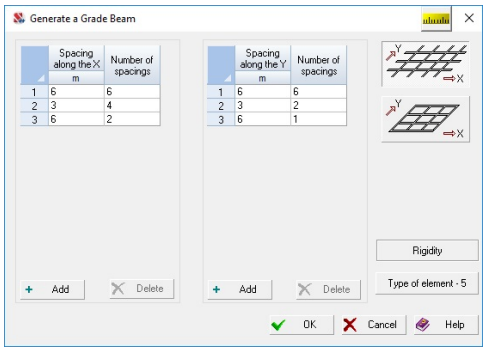


Figure 5.4-1. *The* **Generate a Grade Beam** *dialog box*

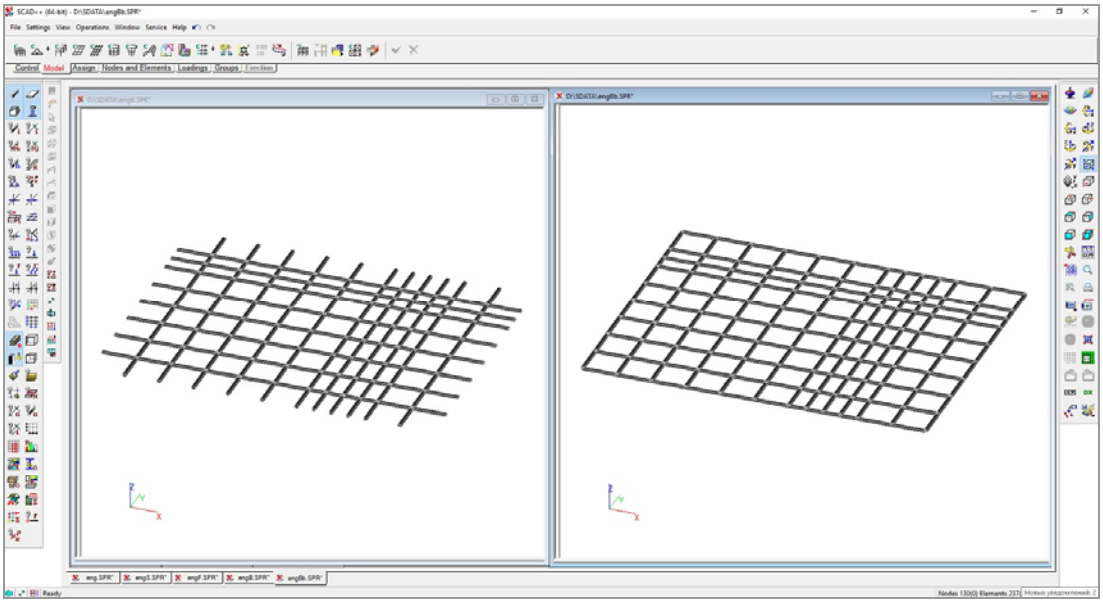


Figure 5.4-2. *Design models of grade beams open and closed along the contour*

When generating a model of a grade beam, you can create models of two configurations – open along the contour and closed (Fig. 5.4-2). A configuration of the model is selected by clicking on the icon with an image of the respective grade beam.

5.5. Generating a Rectangular Mesh of Finite Elements on the Plane

This operation enables to create a rectangular mesh with a variable or constant spacing lying in the XOY or XOZ plane. Parameters of the mesh can be input in the **Generate a Plate Model** dialog box shown in Fig. 5.5-1. Type of the model and its position in space are specified in the respective group with the help of the radio buttons.

Type of the finite elements is assigned automatically depending on the selected type of the model. If the type of finite elements is different from the default one, it can be changed with the help of the respective operation which can be invoked by clicking the **Type of element** button. Plates are assigned type 11 by default, grade beams — 21, shells — 41. If another type of finite element is preferable for a particular structure, it can specified with the help of the **Specify the type of finite elements** operation in the **Assign** toolbar.

The **Rigidity** button invokes an operation of specifying the stiffness properties of elements.

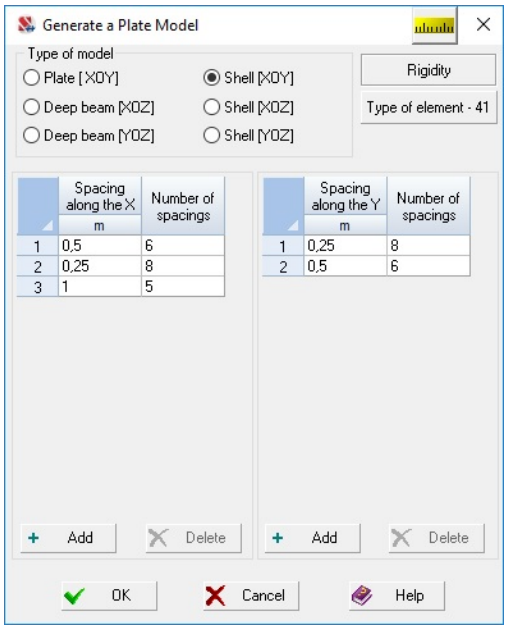


Figure 5.5-1. *The* **Generate a Plate Model** *dialog box*

Let's consider the generation of a slab, 10×5 m with the spacing of the mesh of 1 m in both directions as an example. Specify the spacing of 1 m in the X direction, number of spacings — 10, and the spacing of 1 m in the Y direction, number of spacings — 5. A model shown in Fig. 5.5-2,A will be generated after clicking the **OK** button.

A model with a variable spacing of the mesh of finite elements created according to the data given below is shown in Fig. 5.5-2,B.

Spacing along the X axis	Number of spacings	Spacing along the Y axis	Number of spacings
0.5	6	0.25	8
0.25	8	0.5	6
1	5		

When you specify a different spacing of the mesh, you should keep in mind that the most accurate solution is obtained when the aspect ratio of 4-node finite elements is close to one. Square is perfect in this sense.

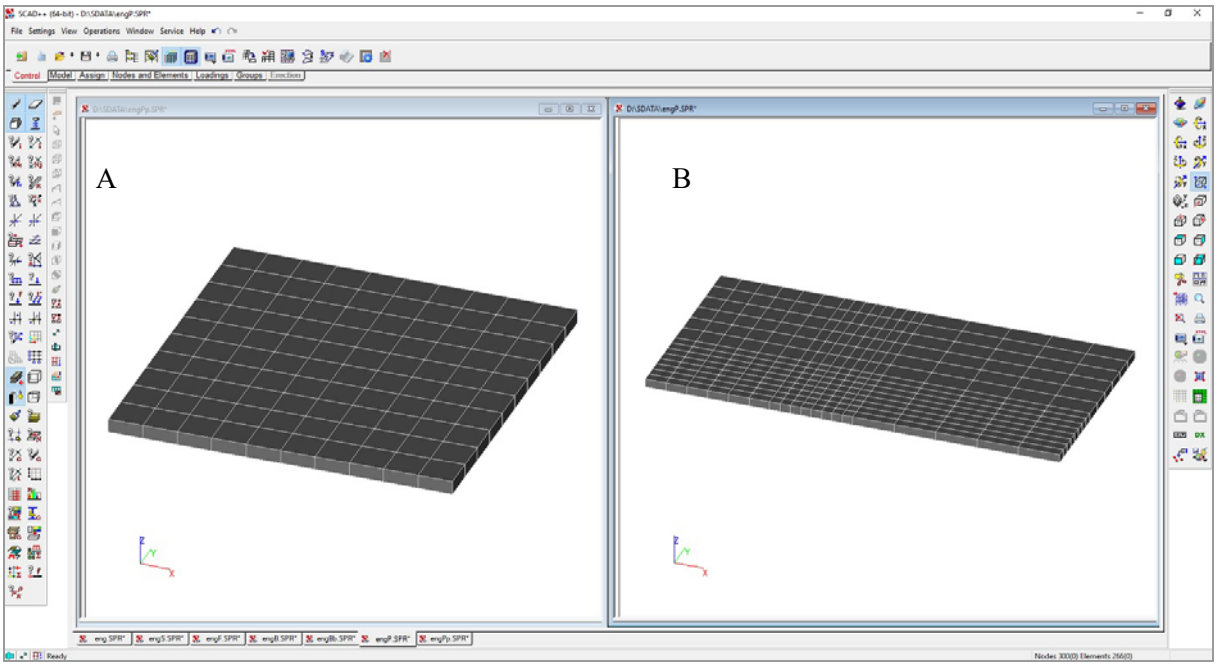


Figure 5.5-2. *Examples of design models from rectangular finite elements*

5.6. Surfaces of Revolution

This operation is used to create design models or their fragments which are modeled using standard surfaces of revolution. You can create models in the form of a cylinder, cone, sphere and torus. The term "surface" is conventional, since models can be also made from bar elements, the nodes of which lie on this surface.

Initial data necessary to perform this operation are specified in the **Create the Surface of Revolution** dialog box (Fig. 5.6-1) and include:

- type of surface (cylinder, cone, sphere, torus);
- kind of finite elements (bars, plates) and their type;
- shape of the finite element mesh (for bar elements — lattice);
- geometric sizes;
- parameters of division;
- stiffness properties of elements.

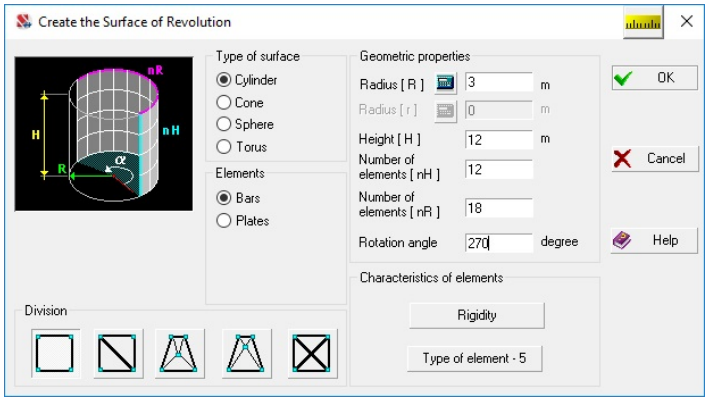


Figure 5.6-1. *The* **Create the Surface of Revolution** *dialog box*

Stiffness properties of elements (the **Rigidity** button) and their type (the **Type of element** button) are specified similarly to the respective operations when generating parametric prototypes of frames. Type 5 is selected by default for bars, and type 42 (triangles) or 44 (quadrangles) – for plates.

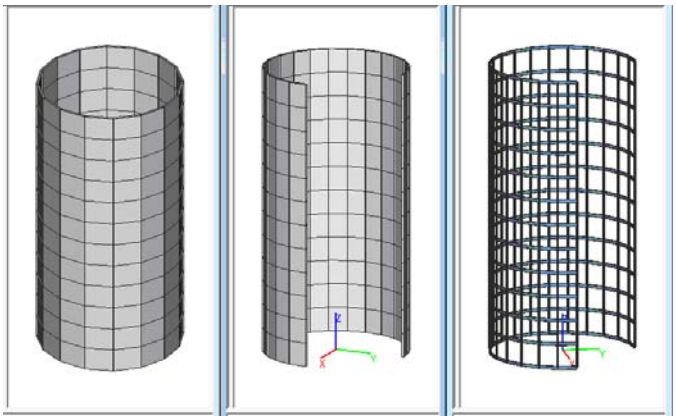


Figure 5.6-2. *Design models of cylinders from shell and bar elements*

If the rotation angle is less than 360°, an open surface is generated. Moreover, the design models in the form of a cylinder, cone or sphere can be truncated.

The following procedure is recommended for creating a surface of revolution:

- select the type of surface;
- select the kind of elements of the model;
- select the type of division;
- specify the geometric properties;
- specify the stiffness properties of elements (this step should be performed if all or most of the elements have the same properties);
- click the **OK** button.

Examples of various variants of design models of cylinders are given in Fig. 5.6-2.

In those cases when in the description of a cylinder, cone and sphere a defining parameter is the length of the face (chord) instead of the radius of the surface, you can calculate the radius as a function of the number of faces and the chord length. It is necessary to perform the following actions to calculate the radius:

- specify the number of faces (nR — number of elements in the base of a cylinder or a cone, and for a sphere – along the equator);
- invoke the **Calculate the Radius from a Chord** dialog box — , (the button is on the left from the radius text field);
- specify the chord length in the respective text field of this dialog box (Fig. 5.6-3) and click the **Calculate** button (the calculated value of the radius of the surface of revolution will be shown in the **Radius** information window);
- click the **Apply** button, the dialog box closes, and the result is automatically input in the respective text field of the **Create the Surface of Revolution** dialog box.

For example, in order to generate a design model of a triangular (nR = 3) pyramid with a base side of 4 m and a height of 4 m, you can use the above technique (Fig. 5.6-4).

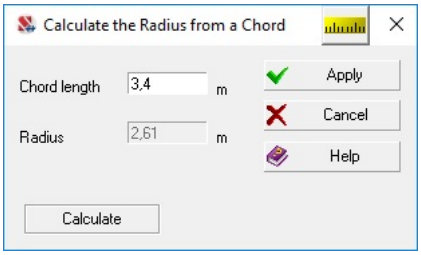


Figure 5.6-3. *The* **Calculate the Radius from a Chord** *dialog box*

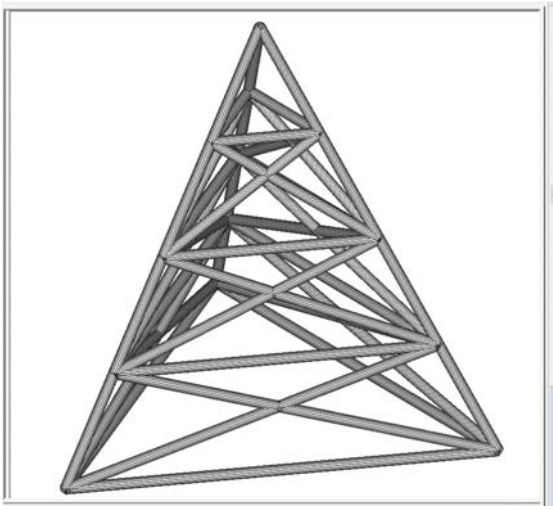
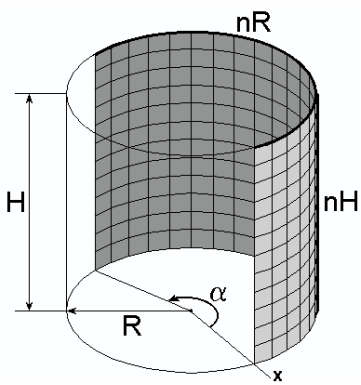


Figure 5.6-4. *Design model of the triangular pyramid*

Description of Parametric Prototypes of Surfaces of Revolution



Cylinder

The following parameters have to be specified to create a cylinder :

R — radius of the cylinder;

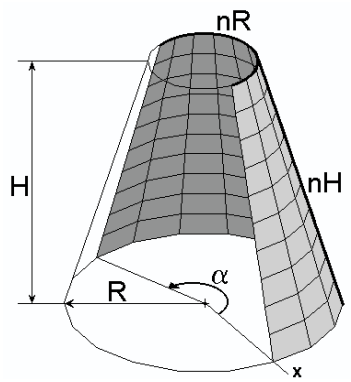
H — height of the cylinder;

nH — number of elements along the height of the cylinder;

nR — the same in the base of the cylinder;

Rotation angle (α) — central angle in degrees ($0° < \alpha \leq 360°$). If the rotation angle is less than 360°, an open surface is generated.

Any shape of the division of the mesh or lattice can be selected for the cylinder from those suggested in the **Create the Surface of Revolution** dialog box. Examples of design models of a cylinder are given in Fig. 5.6-2.



Cone

A cone can be full or truncated. If a truncated cone is generated, r > 0. The following parameters have to be specified to create a cone:

R — radius of the cone base;

r — radius of the top of the truncated cone;

H — height of the cone;

nH — number of elements along the height of the cone;

nR — the same in the base of the cone;

Rotation angle (α) — central angle in degrees (0° < α ≤ 360°). If the rotation angle is less than 360°, an open surface is generated.

Any shape of the division of the mesh or lattice can be selected for the cone from those suggested in the **Create the Surface of Revolution** dialog box. Examples of design models of a cone are given in Fig. 5.6-5.

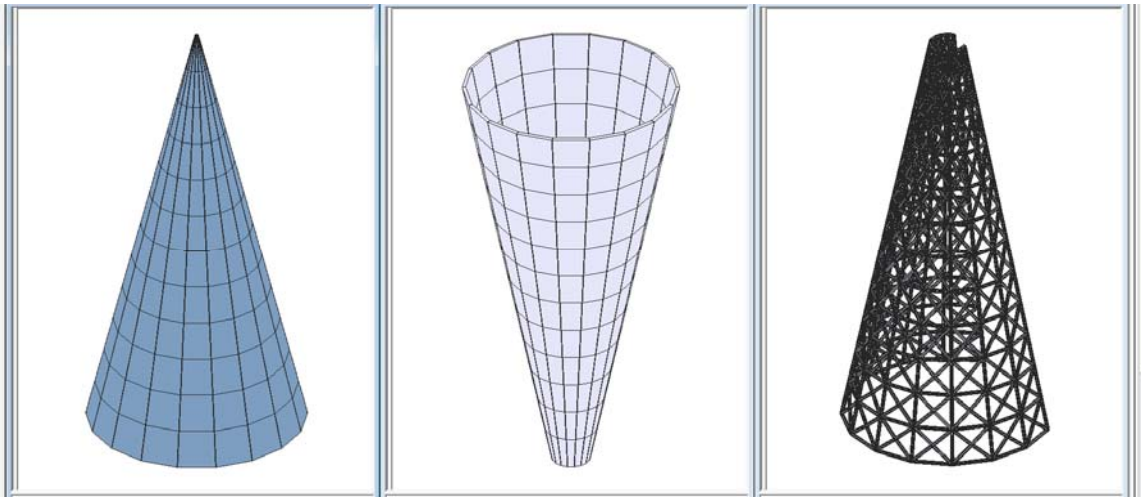
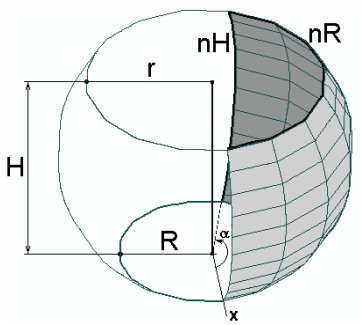


Figure 5.6-5. *Design models of a cone from shell and bar elements*



Sphere

The following parameters have to be specified to create a sphere:

R — radius of the lower cutting plane;

r — same of the upper one;

H — distance between the upper and lower cutting planes;

nH — number of elements along the height of the sphere;

nR — the same along the equator of the sphere;

Rotation angle (α) — central angle in degrees ($0° < \alpha \leq 360°$). If the rotation angle is less than 360°, an open surface is generated. In order to create a full sphere, the values of radiuses of cutting planes are specified as zero, and the height **H** must be equal to the diameter of the sphere.

Any shape of the division of the mesh or lattice can be selected for the sphere from those suggested in the **Create the Surface of Revolution** dialog box. Examples of design models of a sphere are given in Fig. 5.6-6.

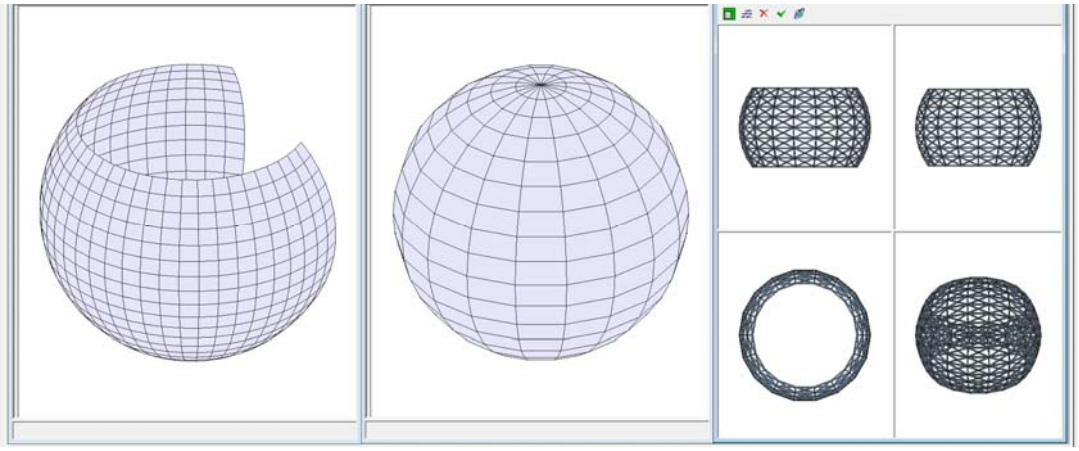
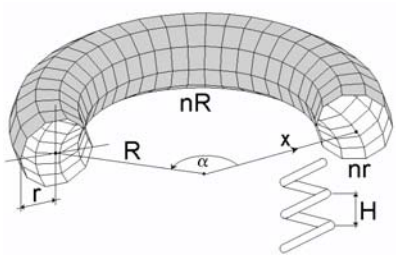


Figure 5.6-6. *Design models of a sphere from shell and bar elements*



Torus

The following data have to be specified to create a torus:

R — radius of the torus measured from the center of rotation to the axis passing through the center of the section of the torus;

r — radius of the section;

H — lead of the helix;

nr — number of elements in the section of the torus;

nR — the same along the length of the torus;

Rotation angle (α) — central angle in degrees.

If α > 360° and **H** > 0, a spiral is generated (Fig. 5.6-7).

There is a restriction on the choice of the shape of elements when a spiral is created from plate elements. For example, 4-node elements cannot be used in this case, since non-planar elements may appear. Unacceptable shapes of the mesh are blocked automatically. Examples of design models of a torus are given in Fig. 5.6-7.

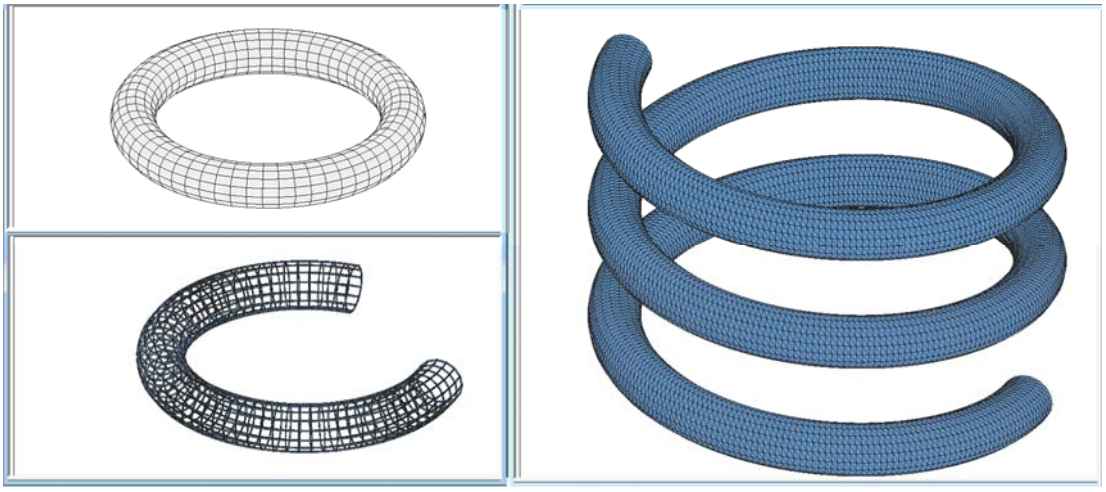


Figure 5.6-7. *Design models of a torus from shell and bar elements*

5.7. Creating an Analytically Defined Surface of Revolution

— Creating a surface of revolution by the specified formula.

This operation enables to create analytically defined surfaces of revolution. Data for performing the operation are specified in the **Generate the Surface of Revolution** dialog box (Fig. 5.7-1).

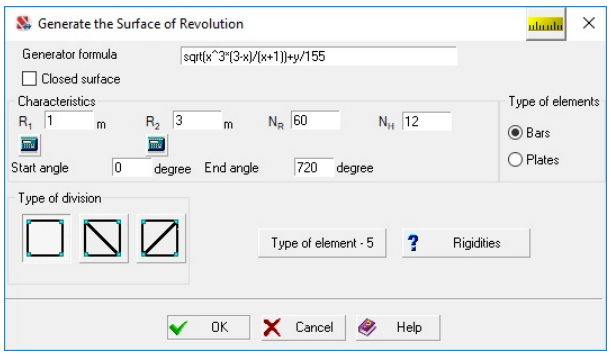
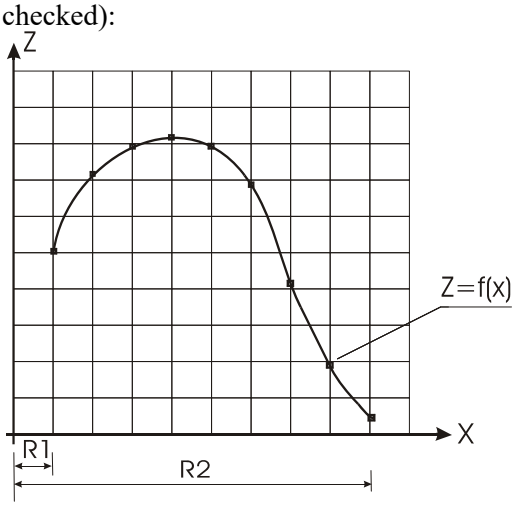


Figure 5.7-1. *The* **Generate the Surface of Revolution** *dialog box*

The operation enables to create closed, open and helical surfaces from bar and shell elements obtained by moving a generator z = *f*(x) along a circular arc.

The following parameters are specified to generate a closed surface (the respective checkbox is checked):



R_1 — initial circle radius (distance from the start of the generator to the Z axis);
R_2 — final circle radius (distance from the end of the generator to the Z axis);
N_r — number of elements of the surface of revolution along the circular arc;
N_h — number of elements of the surface of revolution along the generator.

There is a restriction on the use of meshes of 4-node elements for models from shell elements, since non-planar elements may appear.

Types of elements and their rigidities can be assigned when specifying the parameters of the model. *By default*, type 5 is selected for bars, 42 (triangles) or 44 (quadrangles) – for plates.

Figure 5.7-2,A shows an example of a closed surface obtained by specifying the generator

$$z = \sqrt{x^3\left(3-x\right)/\left(x+1\right)},$$

when the **Closed surface** option is active and the parameters have the following values: $R_1 = 1$ m; $R_2 = 3$ m; $N_r = 30$; $N_h = 20$.

In order to obtain open surfaces you also have to specify the start and end angles of the arc. Thus, for the start and end angles of 0° and 270° respectively a surface shown in Fig. 5.7-2,B will be obtained.

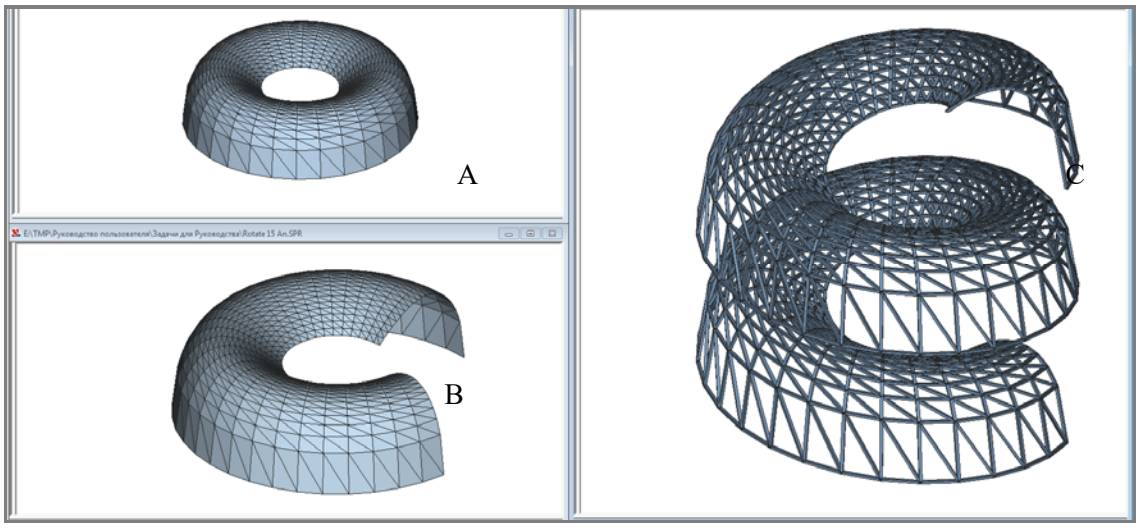


Figure 5.7-2.

Finally, in order to obtain a helical surface, it is necessary to introduce a second variable – Y into the formula, the variation of which describes the process of "twisting" the helix as a function of angle. For example, by adding the second variable — y/155 into the formula

$$z = \sqrt{\left(x^3\left(3-x\right)\right)/\left(x+1\right)} + y/155$$

at the values of the start and end angles 0° and 720° respectively, we will obtain a new model (Fig. 5.7-2,C). Here the number of elements along the arc is $N_R = 60$ (a bar model is used).

In those cases when a defining parameter is the length of the face (chord) instead of the radius of the surface, you can calculate the radius as a function of the number of faces (N_R) and the chord length. It is necessary to perform the following actions to calculate the radius:

- specify the number of faces (N_R — number of elements along the circular arc);
- invoke the **Calculate the Radius from a Chord** dialog box — ;
- specify the chord length in the respective text field of this dialog box and click the **Calculate** button (the calculated value of the radius of the surface of revolution will be shown in the **Radius** information window);
- click the **Apply** button, the dialog box closes, and the result is automatically input in the respective text field of the **Create the Surface of Revolution** dialog box.

Rules for Entering Mathematical Formulas

The following rules should be observed when entering mathematical formulas :

- names of functions must be entered in lowercase Roman letters;
- the fractional and the integer parts of a number must be separated by a decimal point;
- Roman letters **x** and **y** must be used as arguments;
- arithmetic operations must be specified with the symbols +, –, *, /, raising to a power — ^ (for example, x^3 can be written as x^3).

  The following functions can be used in the formulae:

  floor — the greatest integer not greater than the argument;
  tan — tangent;
  sin — sine;
  cos — cosine;
  asin — arc sine;
  acos — arc cosine;
  atan — arc tangent;
  exp — exponent;
  ceil — the least integer greater than the argument;
  tanh — hyperbolic tangent;
  sinh — hyperbolic sine;
  cosh — hyperbolic cosine;
  log — natural logarithm;
  log10 — decimal logarithm;
  abs — absolute value;
  sqrt — square root.

Example. Radio Telescope Reflector

Radio telescope reflector (Fig. 5.7-3,A) in the form of a paraboloid of revolution with the diameter of 60 m and the height of 6 m is defined by the generator formula $z=x^2/150$ in the cylindrical coordinate system.

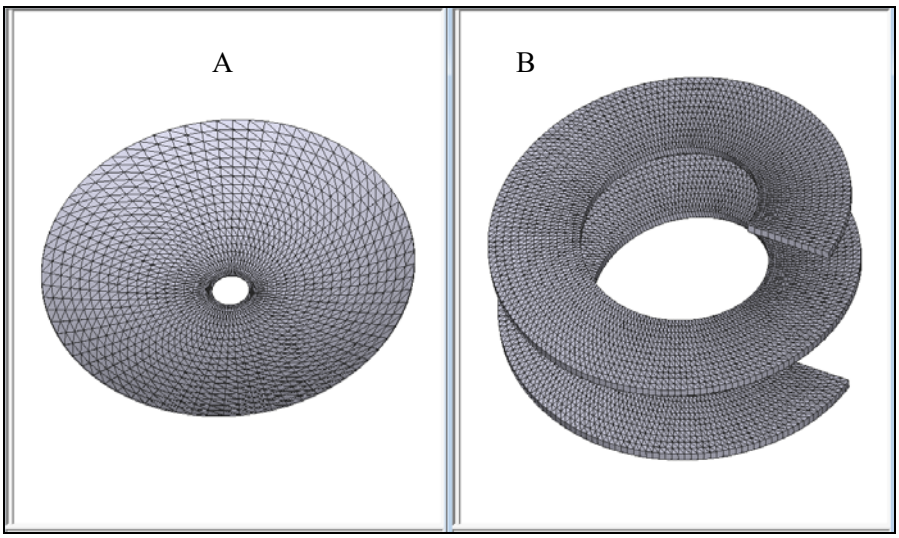


Figure 5.7-3. *Examples of design models of surfaces of revolution*

Example. Helical Ramp

In the case of a helical ramp (Fig. 5.7-3,B) the generator is based on the formula H / 360 * *y*, where H — rise of the stairs per one turn (360° degrees).

Characteristics of a helical ramp — formula 3.2 / 360*y (rise per one turn 3.2 m), inner radius of 3 m, outer radius of 6 m, number of elements along the length — 360, number of elements along the width — 12, start angle 0°, end angle 720º (two full turns).

5.8. Creating an Analytically Defined Surface

— Creating a surface by the specified formula

This operation enables to create a design model in the form of an analytically defined surface obtained by tabulating a function of two variables $z = f(x,y)$ with an interval and a step specified for each variable. Data for performing the operation are input in the **Surface Given by the Formula** tab of the two-tab **Analytically Defined Surfaces** dialog box (Fig. 5.8-1).

The model can be created from bar or from plate elements. In addition to the formula of the surface (rules for entering mathematical formulas are given above) it is necessary to specify the following characteristics:

$\mathbf{X_{beg}}$ — start value of X;
$\mathbf{X_{end}}$ — end value of X;
$\mathbf{N_x}$ — number of elements in the X direction;
$\mathbf{Y_{beg}}$ — start value of Y;
$\mathbf{Y_{end}}$ — end value of Y;
$\mathbf{N_y}$ — number of elements in the Y direction.

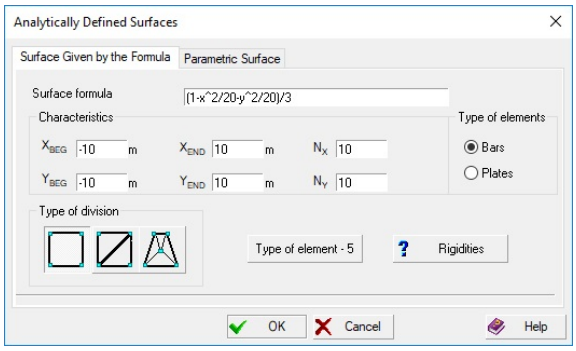


Figure 5.8-1. *The* **Analytically Defined Surfaces** *dialog box*

This tab also enables to select finite elements of the model (bars or plates) and the type of division of the surface. If the type of finite elements is different from the default one, it can be changed with the help of the respective operation which can be invoked by clicking the **Type of element** button. Type 5 is assigned to bar elements by default, and types 44 (4-node) and 42 (3-node) — for shells depending on the selected shape of division. The **Rigidities** button invokes an operation of specifying the stiffness properties of elements.

Example. Shell (Paraboloid of Revolution)

The model is described by the relationship $z = (1 - x^2/20 - y^2/20)/2$ at the following values of the characteristics: $X_{beg} = -10$ m; $X_{end} = 10$ m; $N_x = 10$; $Y_{beg} = -10$ m; $Y_{end} = 10$ m; $N_y = 10$. Design models obtained in the result (on the left – from bar elements, on the right – from shell elements) are shown in Fig. 5.8-2.

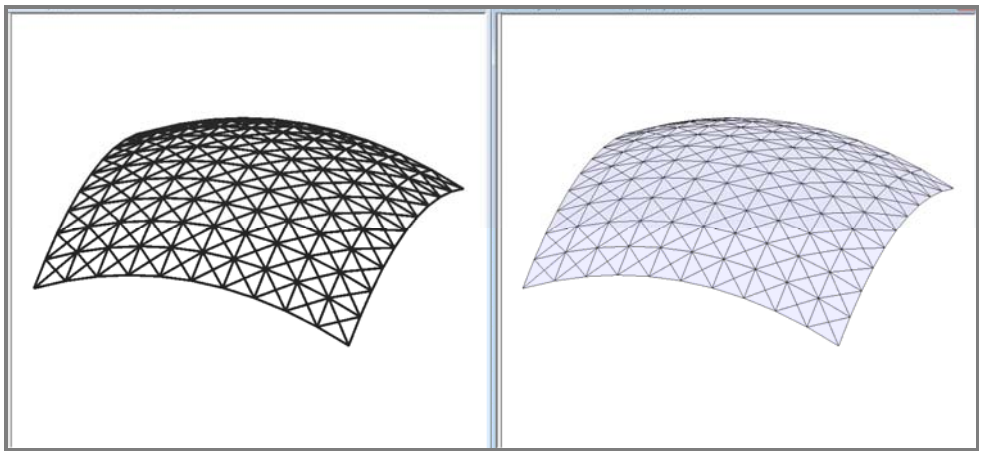


Figure 5.8-2. *Example of a design model of a shell*

5.9. Creating Parametric Surfaces

In addition to the operation of creating analytic surfaces (by the specified formula) the software also enables to generate parametric surfaces. Initial data for this operation are specified in the **Parametric Surface** tab of the **Analytically Defined Surfaces** dialog box (Fig. 5.9-1).

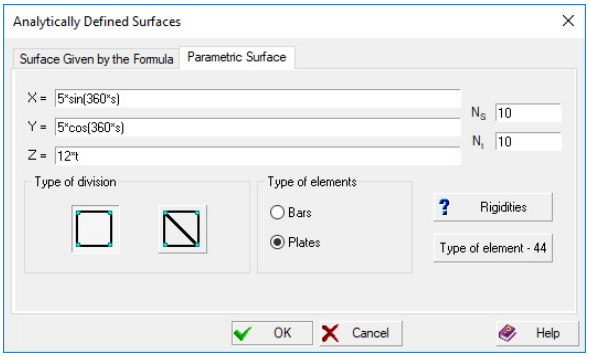


Figure 5.9-1. *The* **Parametric Surface** *tab of the* **Analytically Defined Surfaces** *dialog box*

The following relationships are considered $x = f(s,t)$, $y = f(s,t)$, $z = f(s,t)$. It is assumed that each variable s and t changes in the interval [0, 1], and the variable s successively takes the values 0, $1/N_s$, $2/N_s$, … 1, and the variable t — respectively 0, $1/N_t$, $2/N_t$, … 1, where N_s and N_t — number of steps of tabulation of the respective variables.

For example, formulas describing the surface of a cylinder with a radius 5 and height 12, will be as follows:

$x = 5 \sin\alpha$;
$y = 5 \cos\alpha$;
$z = 12\ h$

and must be input in the respective text fields of the **Parametric Surface** tab as follows:

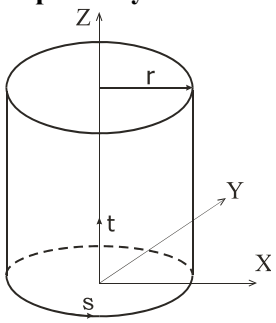
$x = 5*\sin(360*s)$;
$y = 5*\cos(360*s)$;
$z = 12*t$.

This tab also enables to specify the values of N_s and N_t., select finite elements (bars or plates) and the type of division of the surface. If the type of finite elements is different from the default one, it can be changed with the help of the respective operation which can be invoked by clicking the **Type of element** button. Type 5 is assigned to bar elements by default, and types 44 (4-node) and 42 (3-node) — for shells depending on the selected shape of division. The **Rigidities** button invokes an operation of specifying the stiffness properties of elements.

Note: **if the formula involves trigonometric or inverse trigonometric functions, it should be taken into account that their arguments (or calculated values) must be in degrees.**

Examples of Creating Design Models Using the Operation of Generating Parametric Surfaces

Example 1. Cylinder



Initial data:

Radius r = 3 m;
Height h = 12 m;
Number of steps of the variable s — N_s = 10;
Number of steps of the variable t — N_t = 10;

Formulas for the generation of the model will be as follows:

Variable	Formula	Record
x	$r\cos(s)$	3*cos(360*s)
y	$r\sin(s)$	3*sin(360*s)
z	t	12*t

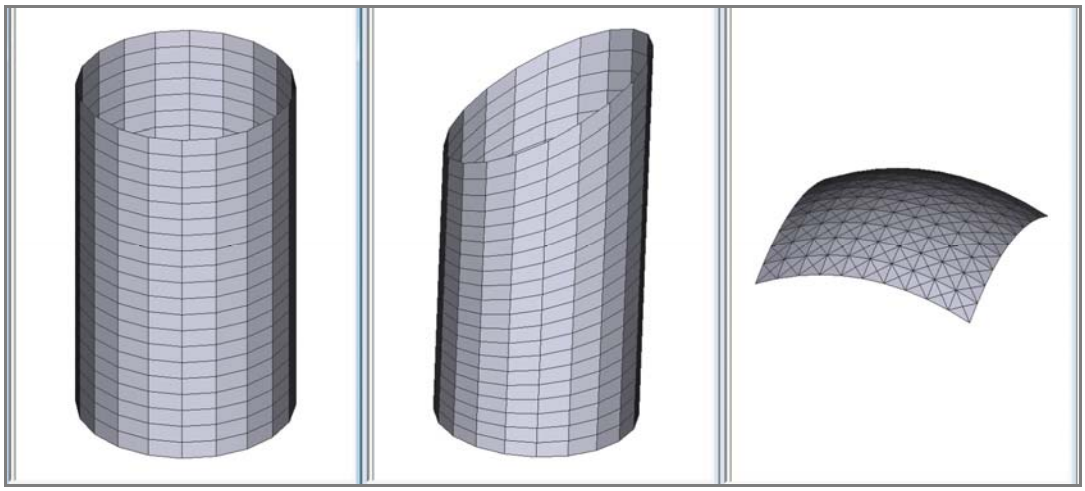
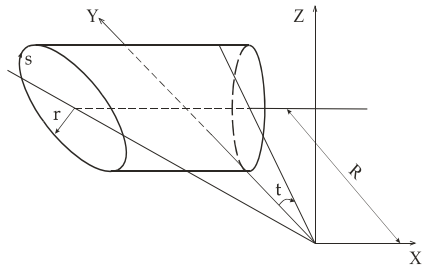


Figure 5.9-2. *Examples of design models of parametric surfaces*

Example 2. Truncated cylinder



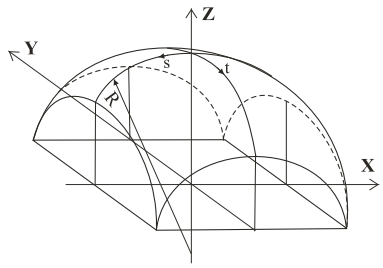
Initial data:

Radius r = 3m;
Height h = 12m ($R = h/\mathrm{tg}(30) - r$);
Angle of the cutting plane — 30^0;
Number of steps of the variable s — 20;
Number of steps of the variable t — 10.

Formulas for the generation of the model will be as follows:

Variable	Formula	Record
x	$(R + r\cos(s))\ \mathrm{tg}(t)$	(12/tan(30)–3+3*cos(360*s))*tan(30*t)
y	$R + r\cos(s)$	12/tan(30)–3+3*cos(360*s)
z	$r\sin(s)$	3*sin(360*s)

Example 3. Spherical surface



Initial data:

Radius R = 10 m;
Number of steps of the variable s — 20;
Number of steps of the variable t — 20.

Formulas for the generation of the model will be as follows:

Variable	Formula	Record
x	$R/\sqrt{2}\ \sin(t)$	10/sqrt(2)*sin(–90+180*t)
y	$R/\sqrt{2}\ \sin(s)$	10/sqrt(2)*sin(–90+180*s)
z	$R/\sqrt{2}\sqrt{2-\sin^2 t-\sin^2 s}$	10/sqrt(2)*sqrt(2–(sin(–90 + 180*t))**2–(sin(–90 + 180*s))**2)

When creating a spherical surface from shell elements it is recommended to use the division into 3-node elements, since the nodes of a 4-node element may not always lie in one plane.

5.10. Creating a Standard Analytic Surface

The software enables to generate the design models of analytic surfaces commonly used in design practice without explicitly entering formulas of the surface. To do this, select the type of surface from the drop-down list in the dialog box (see Fig. 5.10-1), specify the geometric parameters and parameters of the generation of the finite element mesh. This dialog box also provides the formulas of the surface.

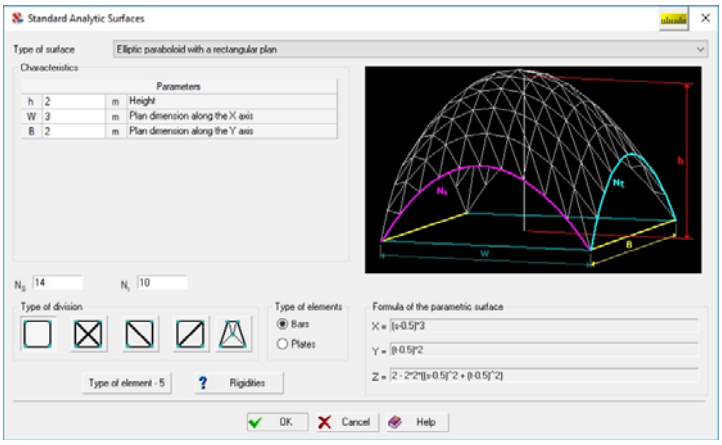
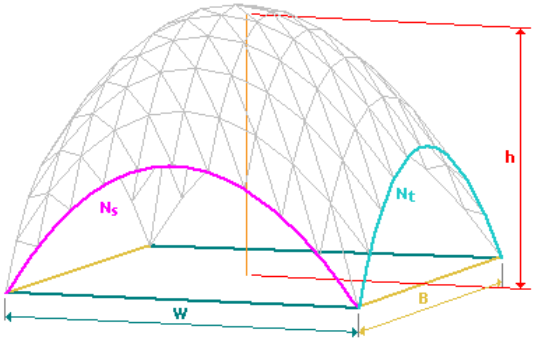
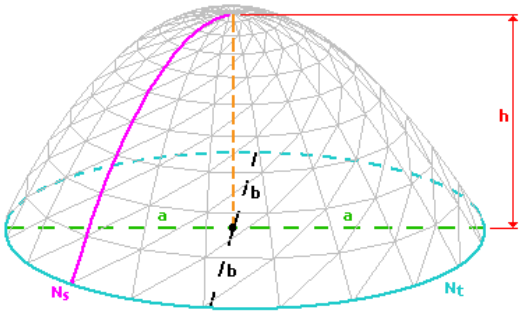


Figure 5.10-1. *The* **Standard Analytic Surfaces** *dialog box*

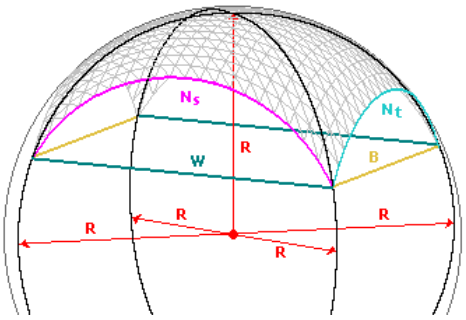
Like in the case of creating arbitrary analytic surfaces, you can also specify the types of finite elements and their stiffness properties. The following types of surfaces are implemented in the program:



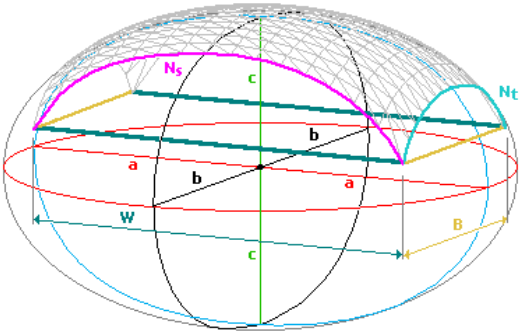
Elliptic paraboloid with a rectangular plan



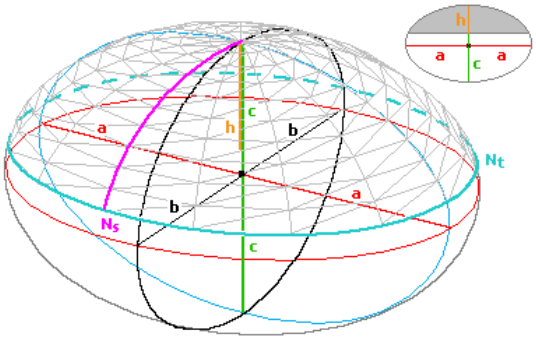
Elliptic paraboloid with an elliptical plan



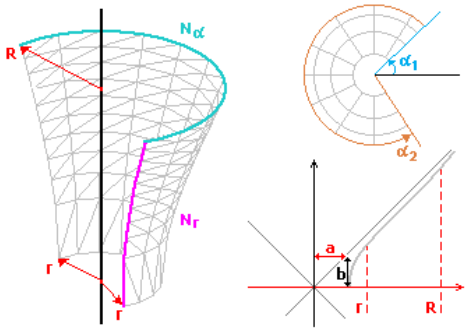
Spherical surface



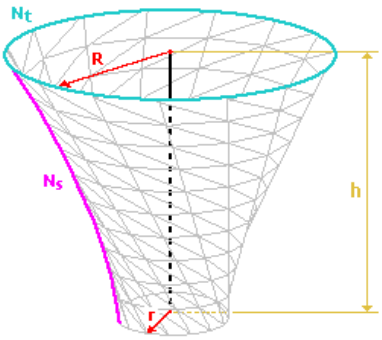
Ellipsoidal surface with a rectangular plan



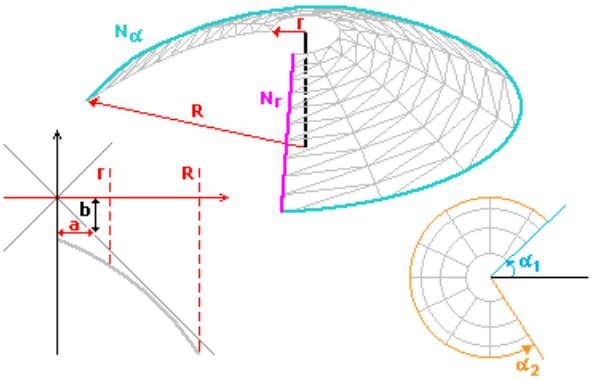
Ellipsoidal surface with an elliptical plan



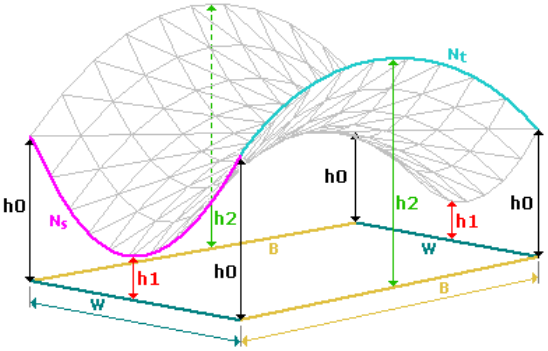
Hyperboloid of revolution, one-sheeted



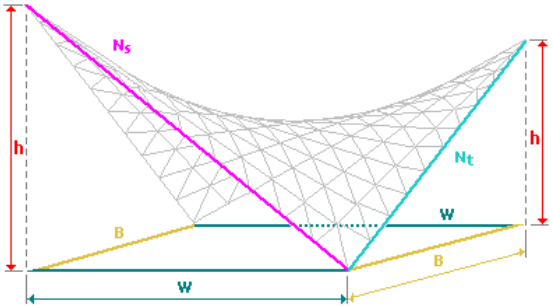
Hyperboloid of revolution, one-sheeted, parametric form



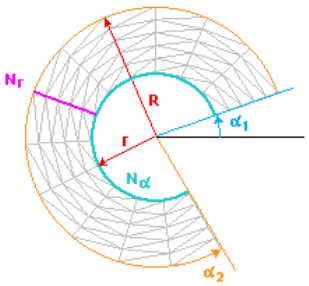
Hyperboloid of revolution, two-sheeted, lower part



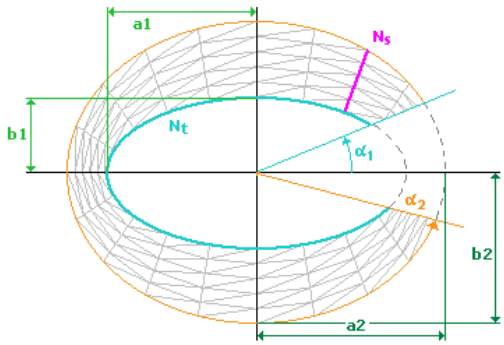
Hyperbolic paraboloid, saddle shape



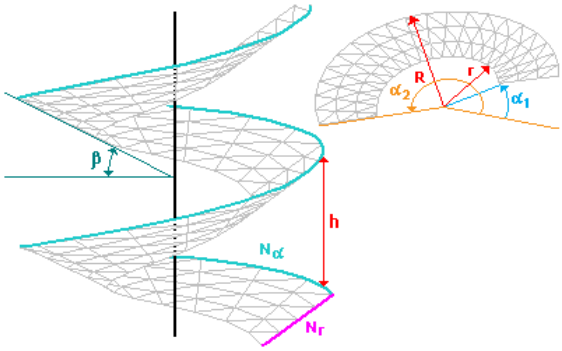
Hyperbolic paraboloid, ruled shape



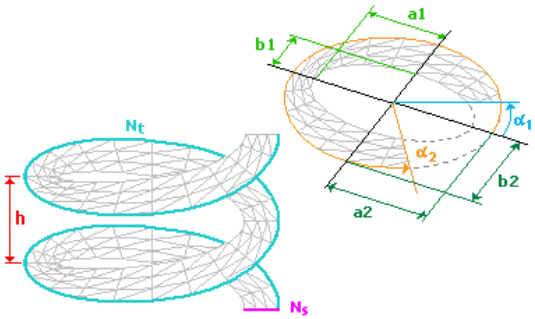
Ring



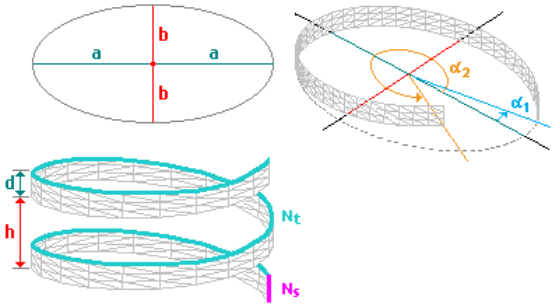
Elliptical ring



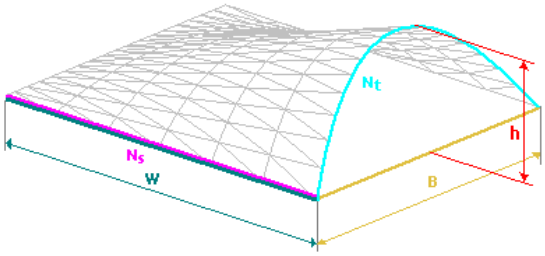
Helicoid



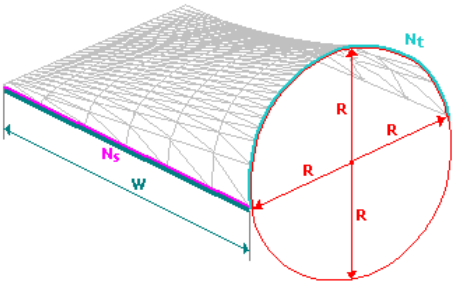
Right helicoid, elliptical



Spiral strip, elliptical



Parabolic conoid



Conoid with a director circle

5.11. Creating Standard Structures

The software enables to generate the design models of standard structures (substructures) commonly used in design practice. To do this, select the type of structure from the drop-down list in the **Standard Structures** dialog box (Fig. 5.11-1), specify the geometric parameters and parameters of the generation of the finite element mesh.

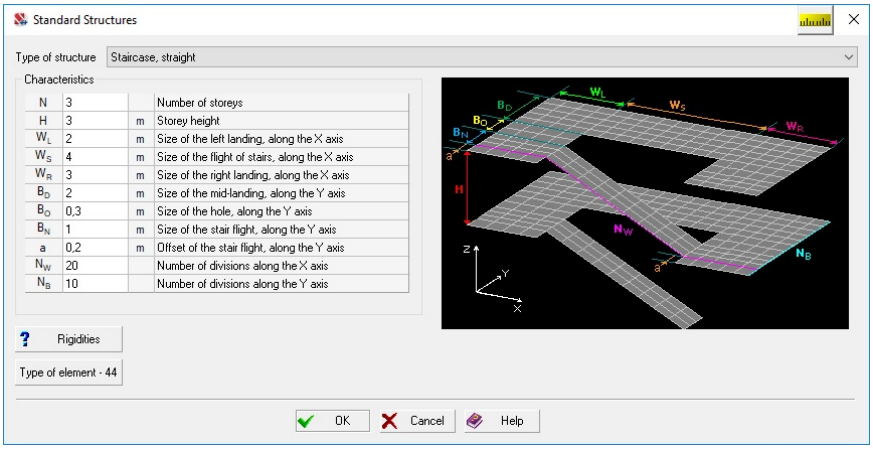
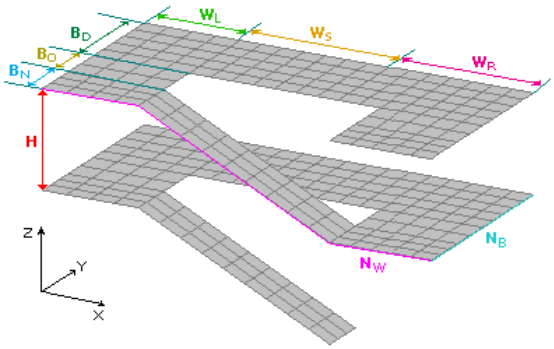
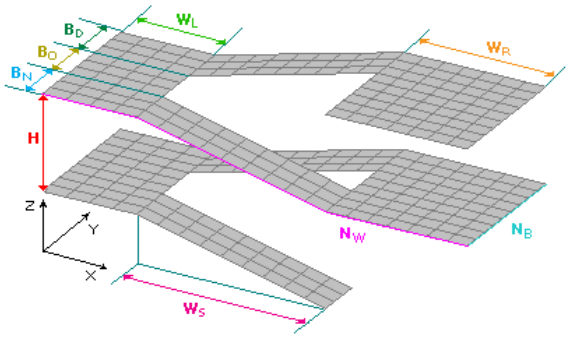


Figure 5.11-1. *The* **Standard Structures** *dialog box*

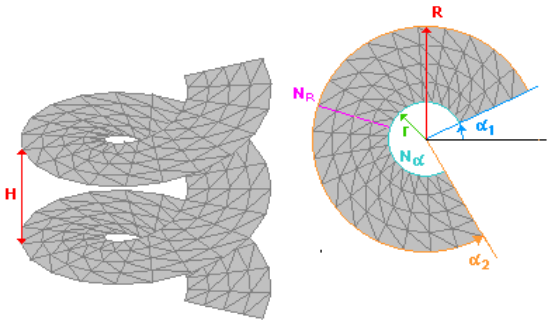
The operation enables to select the types of finite elements and their stiffness properties.
The following types of structures are implemented in the program:



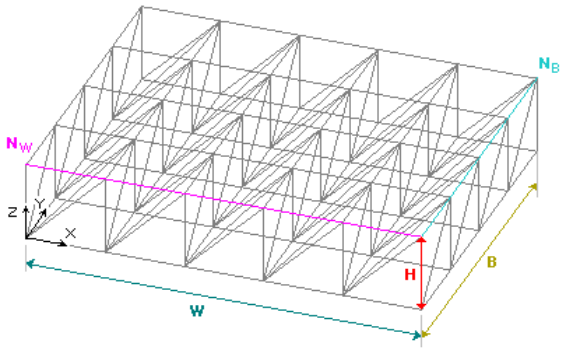
Staircase, straight



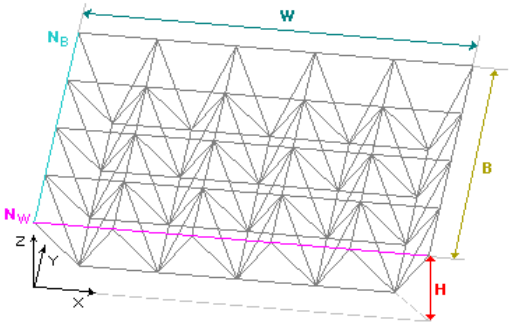
Staircase, dogleg



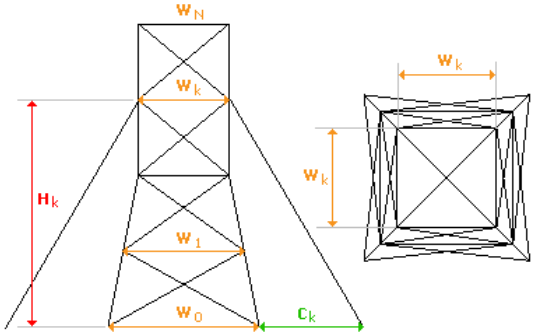
Spiral staircase



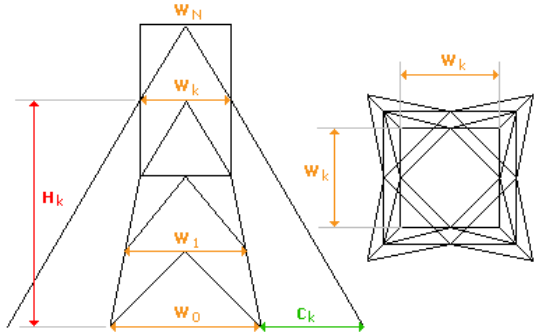
Truss-roof



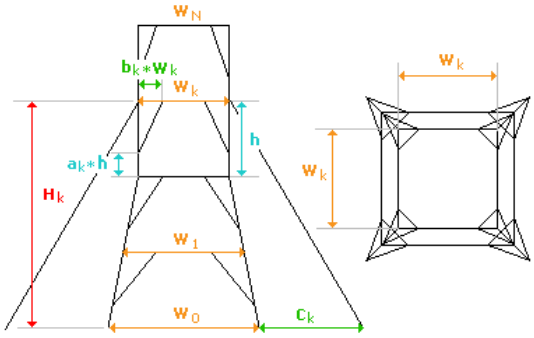
Truss-roof shifted by half step



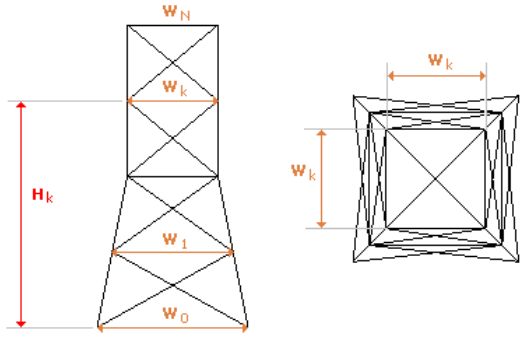
Mast with diagonals



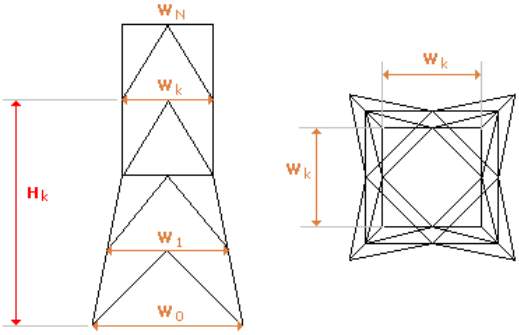
Mast with diagonals from the middle of the horizontal



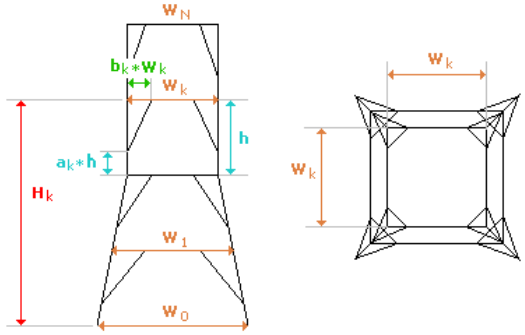
Mast with arbitrary diagonals



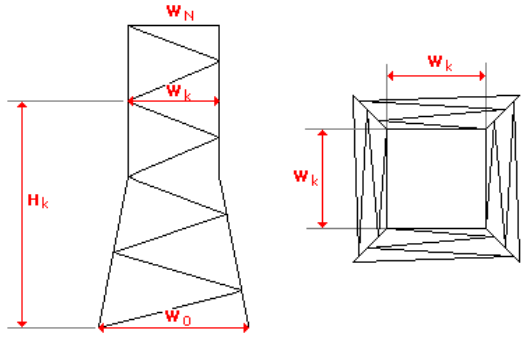
Tower with diagonals



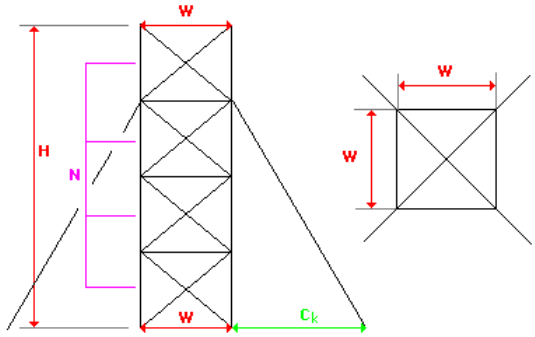
Tower with diagonals from the middle of the horizontal



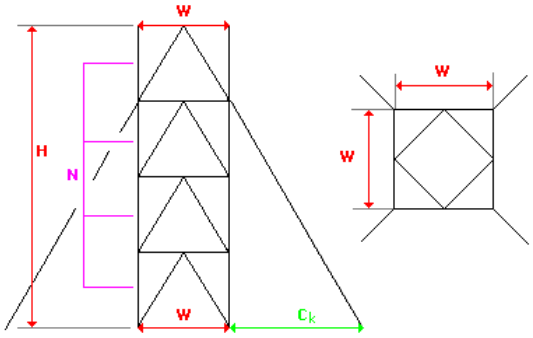
Tower with arbitrary diagonals



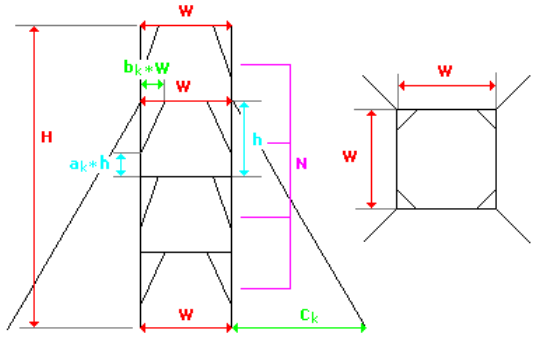
Mast with diagonals from the middle of the vertical



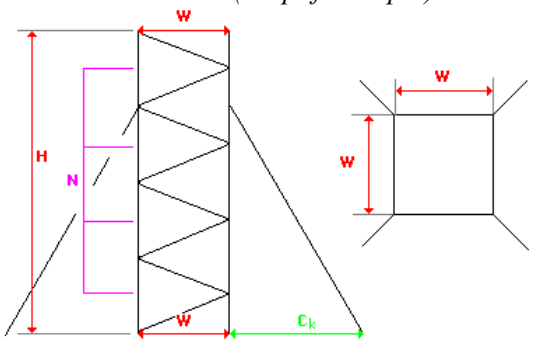
Mast with diagonals (simplified input)



Mast with diagonals from the middle of the horizontal (simplified input)



Mast with diagonals from the middle of the vertical (simplified input)



Mast with arbitrary diagonals (simplified input)

In addition to the general geometric parameters, you can also add/delete levels of the structure; specify different rigidities for elements located at different levels and specify constraints in support nodes for the above list of structures (see Fig. 5.11-2).

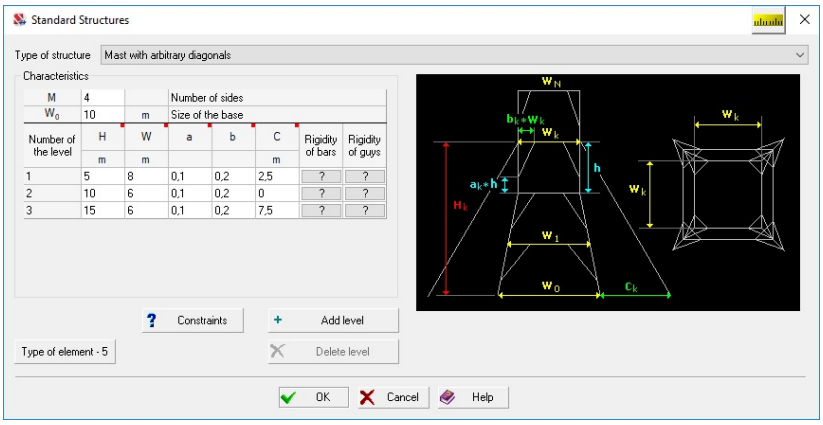


Figure 5.11-2. *The* **Standard Structures** *dialog box (masts and towers)*

5.12. Generating an Arbitrary Mesh of Finite Elements on the Plane

Automatic meshing of the closed area of an arbitrary shape on the plane is one of the most versatile methods of creating meshes of finite elements. The software enables both to perform the meshing when creating a new model, and to perform the meshing of a finished model.

Fundamental Principles

The **Generate a mesh of an arbitrary shape on the plane** operation is invoked from the **Model** section of the toolbar. Control of the meshing is performed with the buttons of the toolbar which appears after the initialization of meshing (Fig. 5.12-1).

Figure 5.12-1. *Meshing control toolbar*

These buttons enable to perform the following operations:

— Specify contour — specify the outer and inner contours bounding the area of meshing;

— Save the results of meshing as a submodel — save the results of meshing as a separate project;

— Generate a triangular mesh of FE on the plane — specify parameters and initialize the process of meshing;

— Improve the meshing quality;

— Analyze the meshing quality;

— Apply the results of meshing to the model — attach the submodel to the model obtained in the result of meshing;

— Transform bars into a meshing contour — generate a meshing contour from the preselected bars;

— Convert the convex hull of the selected nodes into the meshing contour;

— Save the meshing contour — name and save the meshing contour in the project;

— Export the meshing contour to GMSH;

— Transform the contour elements into a meshing contour — transform elements created by the **FORUM** preprocessor into contours for subsequent meshing;

— Indicate mesh refinement nodes — specify nodes in the area of which the local refinement of mesh is performed;

— Indicate mesh refinement lines — specify lines inside the meshing contour along which the refinement of mesh is performed;

— Rejection of the results of meshing/remove the contour.

In addition to the buttons the considered toolbar also contains a list of contours specified in the current project.

In order to perform the meshing of the specified area it is necessary to perform the operations in the following sequence:

- click the **Specify contour** button and select the support nodes of the outer contour with the "rubber band" ("support" nodes are nodes lying in the corner points of the contour);
- repeat the previous step for all inner contours;
- click the **Generate a triangular mesh of FE on the plane** button and specify the parameters of meshing in the **Automatic Meshing** dialog box (Fig. 5.12-5);
- after closing the dialog box by clicking the **OK** button, the specified area is divided into 3-node or/and 4-node finite elements;
- apply the result of meshing to the model by clicking the button or/and save it as a submodel.

After dividing the meshing area the contour is not removed, and you can repeat the specification of new parameters of meshing, etc. until you obtain a satisfactory result. The contour will be removed only after applying the result of meshing to the model and/or saving it as a submodel.

Specifying a Meshing Contour

Before performing the meshing, you must specify the contour bounding the meshing area. The contour is defined by a closed broken line passing through the predefined support nodes. Moreover, bar elements lying on the boundary of the meshing area can be used as a contour. If the meshing area contains inner areas where the meshing is not performed (e.g., holes), each of these areas must be defined by its "own" contour.

The following rules must be observed when specifying a contour:

- when specifying a segment of a contour you do not have to specify nodes that lie on a straight line connecting the support nodes. They will be included in the contour automatically. The exception is the meshing "only on the contour nodes" during the performance of which only the support nodes are taken into account;
- the contour is closed by double clicking after specifying the last node or by pointing at the first node;
- there must not be any duplicate nodes neither support, nor intermediate. Hereinafter, "duplicate" nodes are nodes with the same coordinates or at the distance from each other that is less than or equal to the *accuracy of estimation of the duplicate nodes*. This value is specified in the **Environment Settings** dialog box which can be invoked from the respective section of the ***Settings*** menu;
- there must not be any duplicate nodes inside the contour;
- there must not be any nodes inside the contour (including those belonging to other contours) which lie at the distance from the contour of less than 0.15 of the specified meshing step. The presence of such nodes leads to the interruption of the meshing process due to the appearance of degenerate triangles;
- inner contours must not intersect or touch the outer contour and each other;
- there must not be any contours outside the contour bounding the meshing area.

A contour (or several successively created contours) can be saved by clicking the respective button , which invokes the **Save the Meshing Contour** dialog box (Fig. 5.12-2). The **Operations with contours** group of this dialog box contains a number of buttons which enable to perform the following operations:

Add contour — the created contour or a group of contours is saved in the project under a name specified in the **Contour name** field. If this field does not contain any information, the contour is saved under the name "**Contour No…**", where the contour number is the serial number in the list of created contours.

Replace contour — the specified contour is saved instead of one of the contours saved earlier. In order to perform this operation you have to:

- specify the contour;
- select the contour you want replace in the list (its number and name will appear in the respective fields);
- click the **Replace contour** button.

Replace the contour name — the name of the contour saved earlier is changed to a new one. In order to perform this operation you have to:

- select any contour from the list of contours;
- activate the operation of saving a contour;
- select a contour the name of which you want to change in the list (its number and name will appear in the respective fields);
- enter a new name in the **Contour name** field;
- click the **Replace the contour name** button.

Delete contour — the contour saved earlier is deleted from the list of contours. In order to perform this operation you have to:

- select any contour from the list of contours;
- activate the operation of saving a contour;
- select a contour you want to delete in the list;
- click the **Delete contour** button.

Delete all contours — all the contours saved earlier are deleted from the list. In order to perform this operation you have to:

- select any contour from the list of contours;
- activate the operation of saving a contour;
- click the **Delete all contours** button.

The above operations will be performed after closing the *Save the Meshing Contour* dialog box (by clicking the OK button).

If the meshing area includes several inner contours, it makes sense to save each of them separately. Since after saving a contour the **Remove the contour** operation is performed automatically, the next contour can be specified at once after performing this operation.

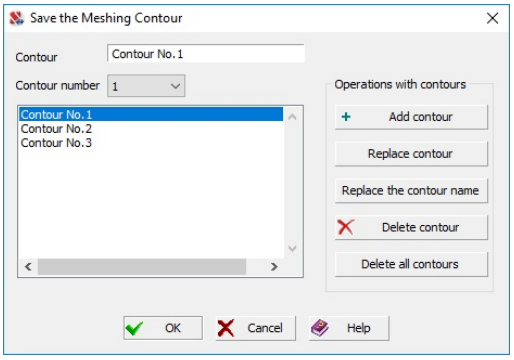


Figure 5.12-2. *The* **Save the Meshing Contour** *dialog box*

If an invalid contour has been specified, it can be removed by clicking the **Rejection of the results of meshing** button .

Meshing will not be performed if several contours belonging to different meshing areas are active. In this case before specifying a second contour you have to remove all contours from the model by clicking the **Reject …** button and repeat the input or open the respective contour from the list.

If a meshing area is specified in such a way that one or several inner contours are saved as individual contours, the outer contour must be loaded from the list the first, and then the inner ones in no particular order.

Transforming Bars into a Meshing Contour

Bar elements can be used to specify a meshing contour. In order to perform this operation select bars forming a contour and click the **Transform bars into a meshing contour** button.

The selected bar elements must meet the following requirements:

- lie in one plane;
- have non-zero length;
- do not have gaps, i.e. form a closed line without self-intersections.

Inner contours can be formed by bar elements or with the help of the **Specify contour** operation and can be saved separately from the outer contour.

If the selected bar elements do not meet the above requirements, the following message appears:

"Unambiguous determination of the contour for the selected bars is impossible"

Converting the Convex Hull of the Selected Nodes into the Meshing Contour

The meshing contour can be specified as the boundary of the convex hull of the selected nodes. In order to perform this operation you have to select nodes and click the **Convert the convex hull of the selected nodes into the meshing contour** button.

The selected nodes must meet the following requirements:

- the number of nodes must not be less than three;
- lie in one plane;
- do not lie on one line.

Converting Enlarged Elements (Objects) into a Meshing Contour

Enlarged elements and objects are created in the **FORUM** preprocessor and model such structural members as columns, beams, floors, walls and roofs. A design model of the finite element method is created by an automatic conversion of these objects into sets of finite elements. Meshing is used to convert planar objects (floors, walls and roofs).

The program also enables to perform a non-automatic meshing of enlarged elements using the considered operations. In this case each enlarged element is meshed individually and its faces are considered as a contour (openings – as inner contours).

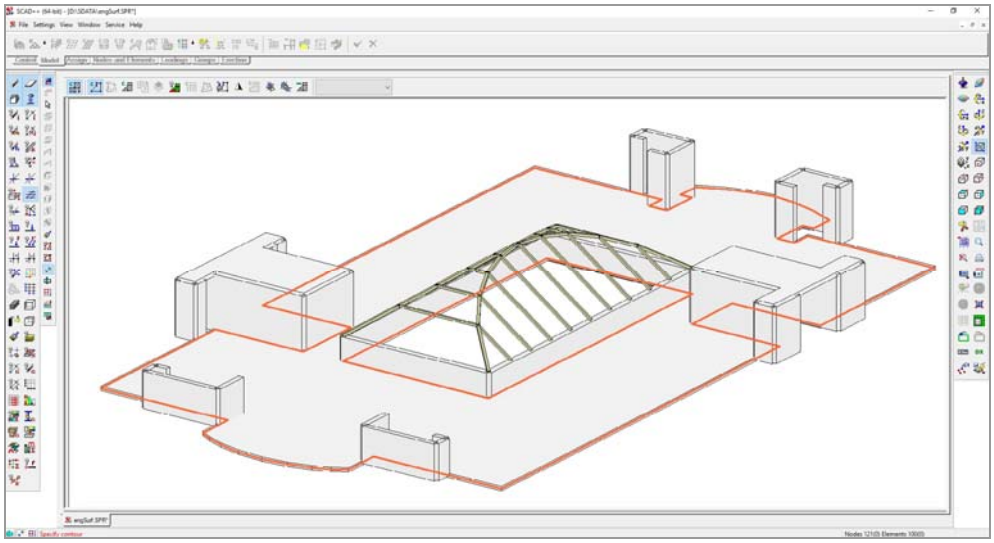


Figure 5.12-3. *Meshing contour based on the floor element*

Stiffness parameters specified for the object are inherited by all finite elements obtained in the result of meshing.

Enlarged elements can have any shape and include any number of inner contours (openings) not in contact with each other and not intersecting the outer contour.

The meshing of enlarged elements is performed in the following order:

- activate the meshing function in the **Model** section of the toolbar;
- click the **Transform the contour elements into a meshing contour** button;
- select an element by pointing at any part of the contour including the inner one. The selected element will be highlighted in color (Fig. 5.12-3);
- click the **Generate a triangular mesh of FE on the plane** button , a dialog box will appear where you have to specify the parameters of meshing. The meshing will be performed after closing the dialog box by clicking the **OK** button.

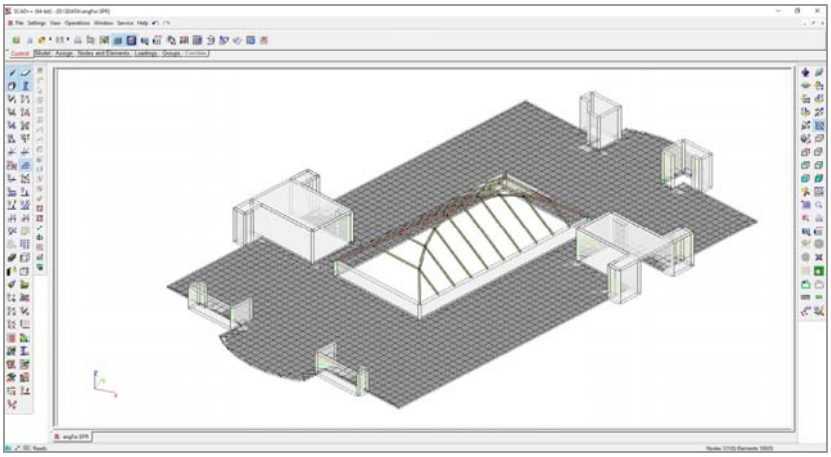


Figure 5.12-4. *Model obtained in the result of meshing of the enlarged element*

The results of meshing must be applied to the model (Fig. 5.12-4) or saved as a separate model before exiting the meshing mode.

Parameters of Meshing

Before performing the meshing you have to first select the method of meshing, specify the parameters and (if necessary) stiffness properties of finite elements (by clicking the **Rigidities** button) in the **Automatic Meshing** dialog box (see Fig. 5.12-5).

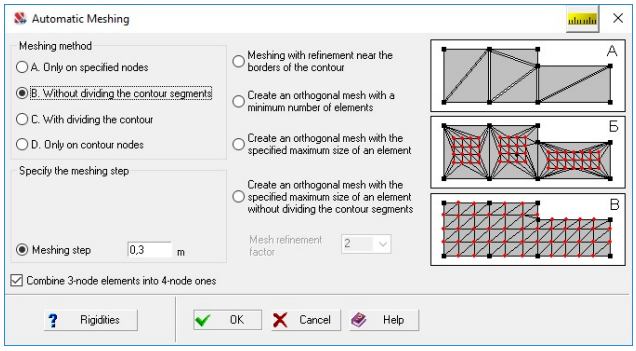


Figure 5.12-5. *The* **Automatic Meshing** *dialog box*

Depending on the selected method the meshing can be performed:

- **A — Only on specified nodes** — new nodes are not created, meshing is performed in such a way that it involves only the existing nodes on the contour and in the meshing area;

- **B** — **Without dividing the contour segments** — the meshing area is divided into finite elements with the specified maximum size of sides (meshing step) but without creating additional nodes on the segments of the contour;
- **C** — **With dividing the contour** — the meshing area and the contour are divided in accordance with the specified step;
- **D** — **Only on contour nodes** — the mesh of finite elements is generated only on the nodes of the contour, nodes in the meshing area are not involved (even if they are within the area).

If the **Combine 3-node elements into 4-node ones** checkbox is checked in the **Automatic Meshing** dialog box, then all possible pairs of triangles will be combined in the process of creating a model.

Additional features of the mesh generation can be specified, namely:

Meshing with refinement near the borders of the contour — method **B** is used, but a mesh of elements with a step number of times indicated in the **Mesh refinement factor** field (value from 2 to 4) less than the specified one is generated near the borders of the contour;

Create an orthogonal mesh with a minimum number of elements — orthogonal meshing area is divided into finite elements using mainly rectangular triangles or square and rectangular elements (if the **Combine 3-node elements into 4-node ones** checkbox is checked);

Create an orthogonal mesh with the specified maximum size of an element — a mesh with mainly rectangular triangles or square and rectangular elements (if the **Combine 3-node elements into 4-node ones** checkbox is checked), the sides of which are less than or equal to the value of the step, is generated;

Create an orthogonal mesh with the specified maximum size of an element without dividing the contour segments — a mesh with mainly rectangular triangles or square and rectangular elements (if the **Combine 3-node elements into 4-node ones** checkbox is checked), the sides of which are less than or equal to the value of the step, is generated. New nodes are not added on the contour when generating a mesh.

Too "elongated" rectangles may appear during the generation of an orthogonal mesh. In this case, the program will automatically switch to the algorithm for creating a non-orthogonal mesh to avoid the generation of elements with a bad geometric shape. The decision on whether to change the algorithm is based on the value of the *maximum aspect ratio of plates and shells* specified by the user in the calculation parameters (see Chapter 11).

Clicking the **Rigidities** button in the **Automatic Meshing** dialog box invokes the **Rigidities of Plates** dialog box, where you can specify the stiffness properties of elements.

The model created in the result of meshing is actually an independent submodel, and after completing the meshing it is necessary to attach it to the nodes of the design model which have been taken into account when performing the meshing. This can be done by clicking the button — **Apply the results of meshing to the model**.

If the submodel has to be attached to other parts of the design model as well or to other models, it has to be saved as a separate project under a unique name (different from the name of the main model) using the button — **Save the results of meshing**.

If a new model has been created in the result of meshing, it can be attached to the main model in the **Assembly** mode (see Sec. 5.13).

When performing the meshing of an area belonging to the existing design model, the contour is specified directly on the model or on its fragment.

As long as the results of meshing are not applied to the model, they can be rejected by performing the operation **Rejection of the results of meshing** — . This operation should be also used if after applying the results of meshing to the model, you have to continue the work in the meshing mode.

Some Nuances

- If bar elements including the contour ones are within the meshing area, they are divided into a number of elements along the length in accordance with the specified meshing step.
- All nodes within the meshing area which do not belong to the area of inner contours will be taken into account when generating a mesh.
- Meshing is not performed if there are duplicate nodes in the meshing area.
- If after performing the meshing it is necessary to repeat the division of the given area, for example, with a different step or method of meshing, you just have to invoke the operation **Generate a triangular mesh of FE on the plane** — again without cancelling the previous division and specify new parameters of meshing.
- If for some reason it is necessary to repeat the input of the contour created earlier, you have to first perform the operation **Rejection of the results of meshing** — , otherwise an error message will be generated.

Mesh Refinement

It is necessary to perform the following actions to refine the mesh near the existing nodes:

- specify the meshing contour or select it from the list;
- click the button — **Indicate mesh refinement nodes**;
- select nodes near which the mesh is refined (the selected nodes will be highlighted);
- click the button — **Generate a triangular mesh of FE on the plane**;
- select the meshing mode in the **Automatic Meshing** dialog box and specify the mesh refinement factor (from two to four);
- click the **OK** button in the dialog box.

Mesh refinement along the line is performed in the following order:

- specify the meshing contour or select it from the list;
- click the button — **Indicate mesh refinement lines**;
- input lines according to the rules similar to those for the input of bars (input lines will be displayed in the colour of the contour);
- click the button — **Generate a triangular mesh of FE on the plane**;
- select the meshing mode in the **Automatic Meshing** dialog box and specify the mesh refinement factor (from two to four);
- click the **OK** button in the dialog box.

Mesh refinement can be performed simultaneously near the nodes and along the lines.

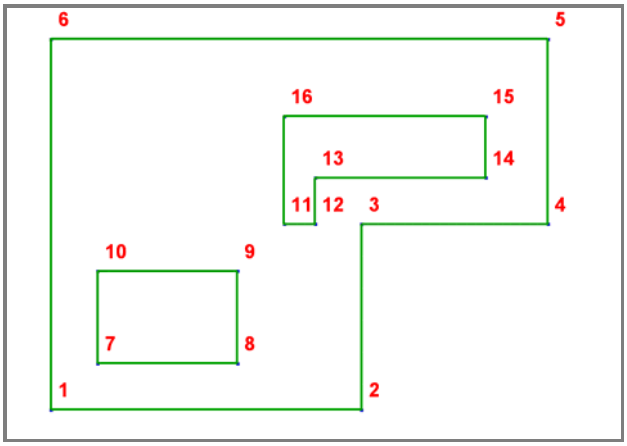


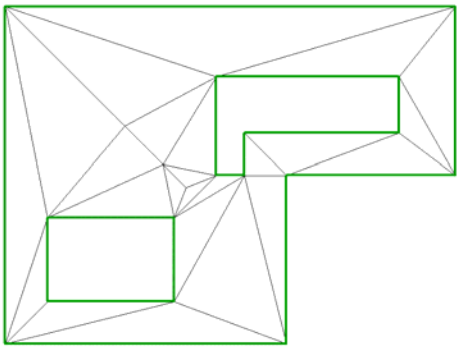
Figure 5.12-6. *Nodes in the meshing area and contours created on their basis*

As an example, let's consider the meshing of an area lying in the XOY plane with nodes, the coordinates of which are given in the following table:

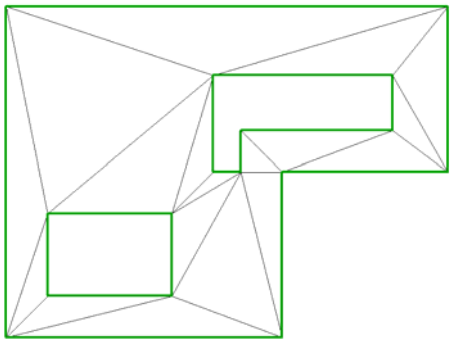
Nodes of the outer contour			Nodes of the inner contours		
Node No.	**X**	**Y**	**Node No.**	**X**	**Y**
1	0.0	0.0	7	3.0	3.0
2	20.0	0.0	8	12.0	3.0
3	20.0	12.0	9	12.0	9.0
4	32.0	12.0	10	3.0	9.0
5	32.0	24.0	11	15.0	12.0
6	0.0	24.0	12	17.0	12.0
			13	17.0	15.0
			14	28.0	15.0
			15	28.0	19.0
			16	15.0	19.0

Numbers of the input nodes and contours specified on their basis are shown in Fig. 5.12-6. Different meshes shown in the figures below can be generated in the meshing area depending on the selected method.

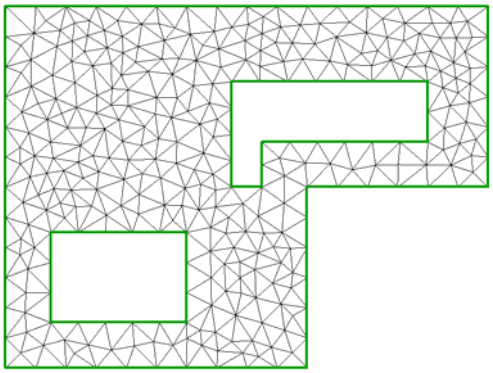
The following meshes of finite elements will be obtained in the area given in the Figure 5.12-6 depending on the selected method of meshing:



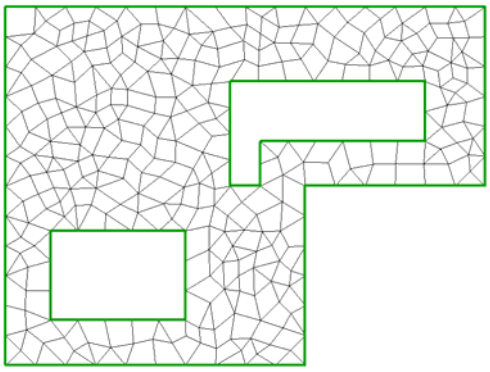
- without dividing the contour segments;



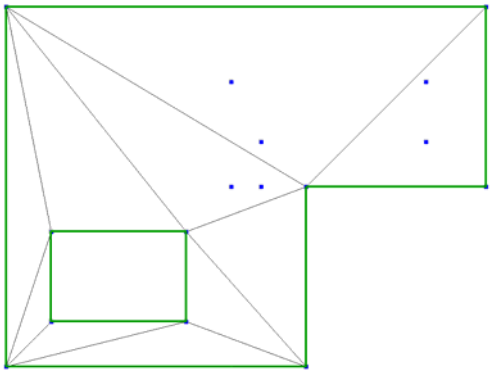
- only on specified nodes;



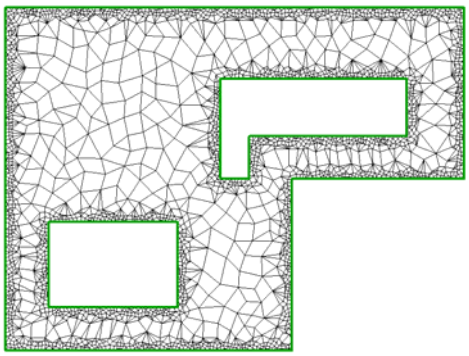
- with dividing the contour (the **Combine 3-node elements into 4-node ones** checkbox is not checked);



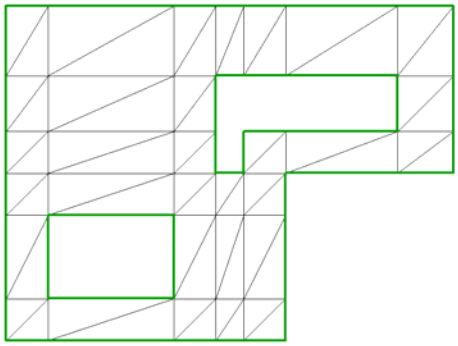
- with dividing the contour (the **Combine 3-node elements into 4-node ones** checkbox is checked);



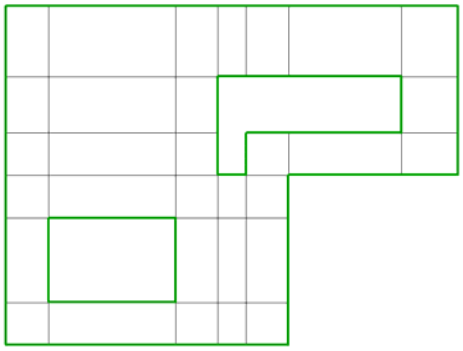
- only on contour nodes;



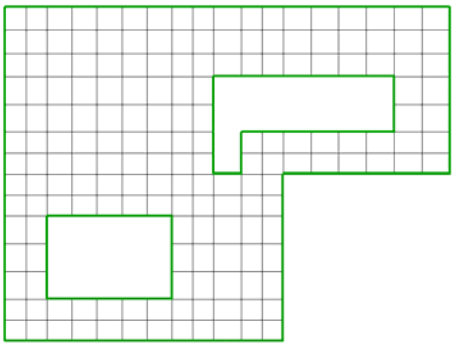
- meshing with refinement near the borders of the contour;



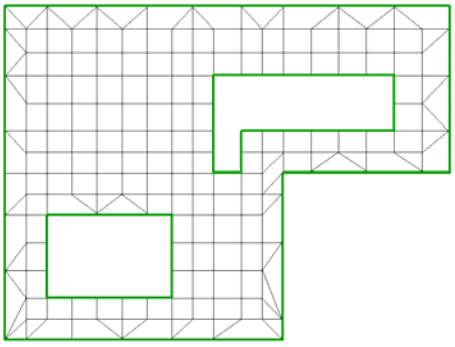
- create an orthogonal mesh with a minimum number of elements (the **Combine 3-node elements into 4-node ones** checkbox is not checked);



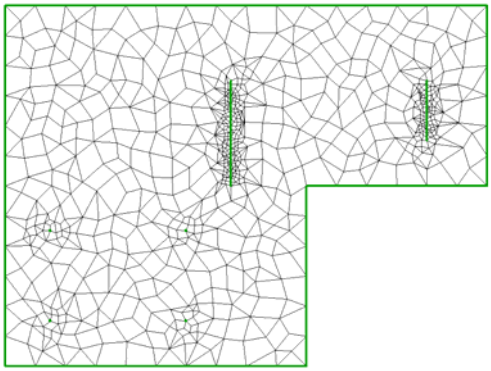
- create an orthogonal mesh with a minimum number of elements (the **Combine 3-node elements into 4-node ones** checkbox is checked);



- create an orthogonal mesh with the specified maximum size of an element;



- create an orthogonal mesh with the specified maximum size of an element without dividing the contour segments;



- mesh refinement near the selected nodes and along the specified lines.

Mesh Quality Assessment

The respective operation in the toolbar of the meshing control is used to assess the quality of the obtained mesh of finite elements. Clicking this button invokes the menu, where you can select two variants: *meshing quality* or *geometrical characteristics of elements*.

Assessment of three- and four-node elements is performed according to the following parameters:

- configuration factor — is a value calculated by the formula $\frac{\sum_{i=1,2,3,4} L_i^2}{4S}$ (for quadrangles) and $\frac{\sum_{i=1,2,3} L_i^2}{4\sqrt{3}S}$ (for triangles), where L_i — length of the *i*-th side, *S* — area of the element;
- minimum angle in the element — it is used as a quality criterion;
- maximum angle in the element — it is used as a quality criterion;
- warping (flatness deviation) for four-node elements — a plane is created based on four points, which approximates the vertices of the element in the best way. The maximum from the distances from the vertices to the plane is calculated. The result is divided by the minimum side.

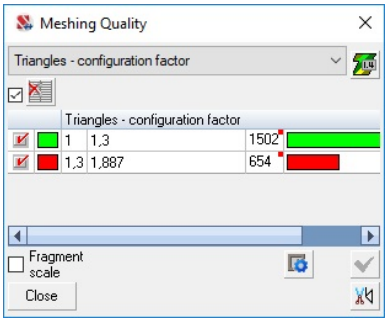


Figure 5.12-7. *The* **Meshing Quality** *dialog box*

The estimation of the solid elements is based on the following parameters:

- ratio between the lengths of the maximum and minimum sides;
- configuration factor — value equal to the maximum configuration factor for all faces.

The results of the quality assessment are displayed as a color scale (Fig. 5.12-7). Elements with the value of the considered factor within the specified range are displayed in green in the model, and those with the value outside the range are displayed in red. The range of the allowable values «**From** and **To**» is specified in the respective text fields of the **Settings** dialog box (Fig. 5.12-8), which can be invoked by clicking the **Color scale settings** button — .

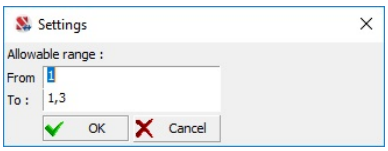


Figure 5.12-8. *The* **Settings** *dialog box*

Principles of control of the color scale do not differ from those of color scales of visualization of other parameters of the design model and are described in Sec. 4.3. Values of the factor can be output on the model by clicking the **Digitization** button — , in the color scale dialog box.

By default the results of the quality assessment are displayed as a “wire” model. You can use the **Delete lines of the invisible contour** operation in the **Display Filters** toolbar for better visualization.

The case *geometrical characteristics of elements* allows to activate color scale of lengths for two-nodes elements, area for plates, and volumes for solid elements.

Improving the Meshing Quality

If the meshing quality (it can be assessed with the help of the operation) turns out to be not satisfactory, you can try to improve it by clicking the button in the toolbar. The program will attempt to move the nodes generated in the process of creating a finite element mesh in such a way so as to improve its quality.

Export the Meshing Contour to GMSH

This operation is used to save the meshing contour in the **GMSH** format (free software for meshing). After performing the meshing by this program the obtained mesh can be imported into **SCAD** (see Sec. 35.1).

5.13. Assembling a Model from Several Models

— **Assembly mode**

This mode enables to perform the generation of a design model by assembling it from several pre-prepared models and/or groups of elements (the procedure of creating groups and work with them are described in Sec. 6.6). Models used in the assembling can include in addition to the geometry other characteristics as well, for example constraints, rigidities, loads, etc., which in the result of the assembly will be included in the resulting model. The model the other models are attached to will be called the main one, and the attached models will be called submodels. Even one node can be used as the main model, and any other model, including the main one, and groups of elements can be used as a submodel.

The toolbar in the assembly mode (Fig. 5.13-1) includes the following buttons:

— select a method of assembling;

— activate the main model;

— activate the attached model;

— assemble.

Figure 5.13-1. *Toolbar of the assembly mode*

Figure 5.13-2. *Toolbar of the window of the submodel in the assembly mode*

Submodel used in the assembly is loaded into a separate window. Toolbar of the window of the submodel (Fig. 5.13-2) includes a button for selecting a group of elements used as a submodel — , a button for loading a design model of the submodel — , and also a button for selecting one of the design models loaded earlier as a submodel — . In the latter case it can be the main model used simultaneously as a submodel.

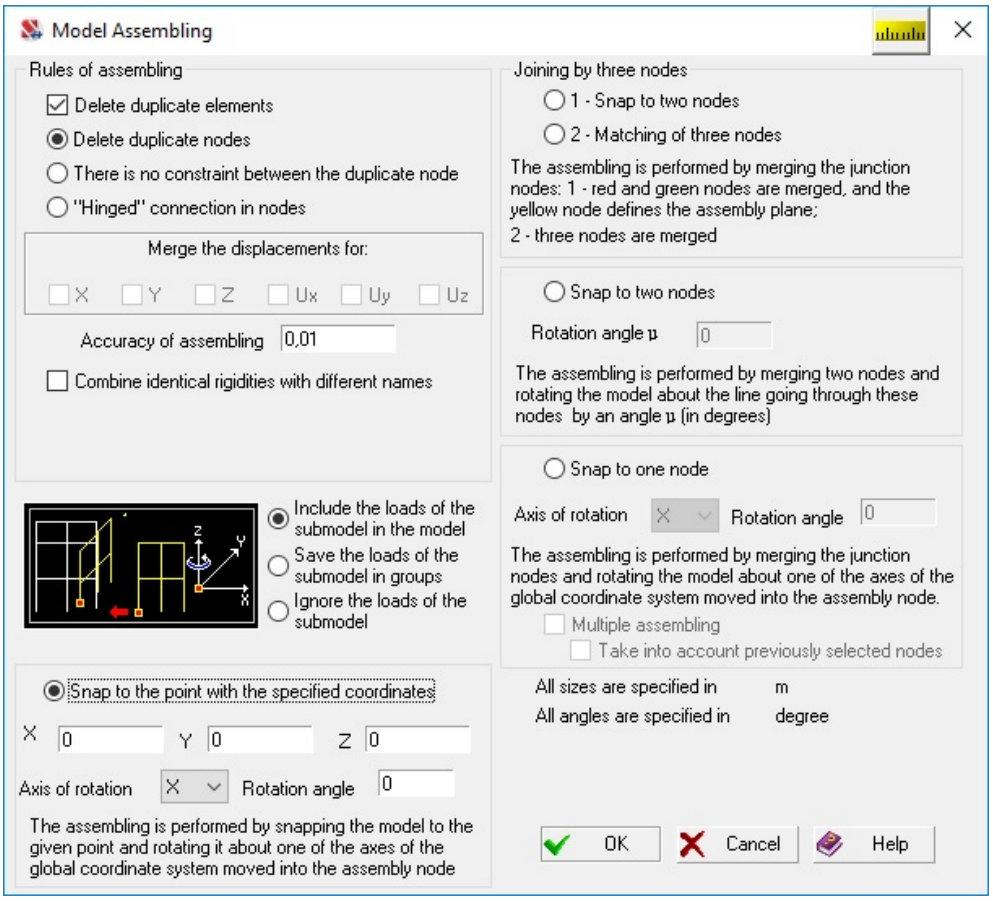


Figure 5.13-3. *The* **Model Assembling** *dialog box*

If the elements in the main model are selected before invoking the assembly mode, these elements will be automatically used as a submodel (attached model). If necessary a submodel can be replaced, or another model or group of elements can be used as a submodel.

Elements have to be selected before invoking the assembly mode.

It is recommended to follow this procedure of assembly:

- click the button — **Assembly mode** in the **Model** section, the panel of the assembly mode (Fig. 5.13-1) will appear in the toolbar of the main window, and the window for loading a submodel will appear in the work area;
- load the submodel using one of the buttons in the toolbar of the window of the submodel (the standard **Open** dialog box can be used to load another design model as a submodel);
- click the button — **Select a method of assembling**;
- select a method and rules of assembling in the **Model Assembling** dialog box (Fig. 5.13-3);
- close the dialog box by clicking the **OK** button;
- depending on the selected method of assembling indicate one, two or three junction nodes in the model and click the **OK** button in the toolbar of the main window.

Once the assembling is performed, the resulting model is output in the window of the main model and the **Result** dialog box appears where you have to select a variant for continuing the work (Fig. 5.13-4).

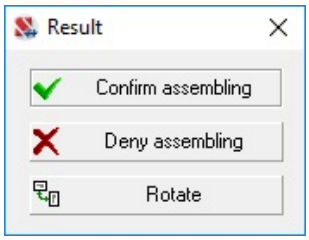


Figure 5.13-4. *The* **Result** *dialog box*

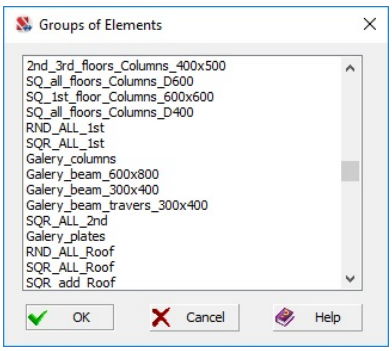


Figure 5.13-5. *The* **Groups of Elements** *dialog box*

The first option is **Confirm assembling**. A new main model will be generated in the result of combining the main model and the submodel in accordance with the selected rules of assembling.

Second option — **Deny assembling**. In the result the main model will remain the same as before the assembly.

Third option — **Rotate** (submodel). The submodel will take a new position with respect to the junction node/nodes, which is formed by rotating 180 degrees around the junction nodes or the axis of rotation specified earlier (when there is only one junction node).

If groups of elements are used as a submodel (the assembling can be performed simultaneously with several groups), they can be selected in the **Groups of Elements** dialog box (Fig. 5.13-5). Groups selected in the list are displayed in the design model in red. After closing the dialog box by clicking the **OK** button these groups will be loaded into the window of the submodel.

Methods of Assembling

Joining by three nodes implies the selection of three nodes in the model and the submodel and can be carried out in two ways — **Snap to two nodes** or **Matching of three nodes**. The method of assembling is specified by enabling the respective radio button in the **Joining by three nodes** group.

When the first method is used, the submodel is snapped to two nodes — the red one and the green one, and the yellow node defines the plane (direction) of the assembly. In this case the model can take two positions with respect to the junction nodes, therefore the **Rotate** operation described above may be useful after performing the assembly.

The second method of assembling implies matching of three nodes of the model and the submodel which defines the position of the submodel unambiguously.

Snap to two nodes is performed by merging junction nodes of the same color in the model and the submodel. Obviously, in this case the submodel may occupy any position with respect to the junction nodes, and in order to define it unambiguously it is necessary to specify the rotation angle of the submodel about the axis passing through these nodes.

If when assembling by two nodes the distances between the respective nodes of the model and the submodel are not equal, the submodel is scaled by a factor equal to the ratio of distances between the junction nodes of the submodel and the model.

Snap to one node is performed by merging the junction nodes in the model and the submodel with a simultaneous rotation of the submodel by a given angle about the specified axis passing through the junction node and parallel to the respective axis of the global coordinate system. If you check the **Multiple assembling** checkbox, then several nodes for snapping the submodel can be indicated in the main model. The submodel will be snapped to each of them by its selected node.

Snap to the point with the specified coordinates is performed similarly to the connection by one node. A point with the specified coordinates the selected node of the submodel is attached to plays the role of the node in the main model.

Rules of Assembling

The assembly can be performed according to different rules and with the specified accuracy of merging the nodes. The accuracy of merging defines which nodes should be considered as duplicate. If after performing the assembly the distance between the node of the model and the node of the submodel is less than or equal to the specified value of accuracy, such nodes are merged into one node in the mode of merging the duplicate nodes.

If the **Delete duplicate elements** checkbox is checked in the **Model Assembling** dialog box (see Fig. 5.13-3), from two matching elements of the main model and the submodel (elements are considered to be matching if all their nodes match) only the element of the main model remains in the resulting model. This element will have the same characteristics as the element of the main model. Loads of the matching element of the submodel will be added to the loads of the element of the main model. If the **Delete duplicate elements** checkbox is not checked, the resulting model will simultaneously contain all elements of the model and the submodel.

If the **Delete duplicate nodes** radio button is enabled, all nodes of the submodel matching those of the main model will be deleted. Constraints and loads applied in the nodes will correspond to those specified in the main model.

If the **There is no constraint between the duplicate nodes** radio button is enabled, the resulting model will contain all nodes of both models without any constraints between them.

If the **"Hinged" connection in nodes** radio button is enabled, hinges will be installed between the matching nodes in the directions the constraints are not imposed on. In the result merged displacements between the node of the model and the submodel are specified in the directions of the constraints. Constraints are specified by checking the respective checkboxes in the **Merge the displacements for** group.

Rigidities of the attached submodels are packed in the course of assembling. The **Combine identical rigidities with different names** checkbox enables to "affect" the packing process.

Accuracy indicator of the merging of nodes during the assembly is an independent parameter and may differ from the value defining the accuracy of matching of nodes in the model.

Working with Loads

If the models to which the loads were applied are involved in the assembly, you have to indicate how to treat the loads when specifying the rules of assembling.

There are three methods to take the loads of the submodels into account:

- including the loads applied to the submodels in the respective loadings of the main model;
- saving each loading of the submodels as a group of loads;
- ignoring the loads of the submodel.

Loads applied to the main model always remain. If one or several submodels have more loadings than the main model, then these loadings are added as groups of loads.

When transferring the loads from the submodel into the resulting model it should be taken into account that the operations of merging duplicate nodes and elements (they are performed if the respective checkboxes and radio buttons are enabled in the **Model Assembling** dialog box) have a priority and are performed before the load transfer operations. Consequently, if the loads are specified in the nodes of the submodel which coincide with the nodes of the main model, then after deleting the nodes of the submodel the loads will be deleted as well. In those cases when the loads have to be taken into account, you should disable the options of merging the duplicate objects when specifying the rules of assembling, perform the assembling, and then delete the duplicate nodes and elements using the respective modes in the **Nodes and Elements** section of the toolbar.

If the loads are not applied to the nodes or elements of the main model matching the objects of the submodel, the problem of load transfer can be solved by changing the status of the models during the assembly, i.e. take the submodel as the main model, and vice versa. These recommendations will be also useful in cases when the assembly involves models consisting only of nodes to which the loads are applied.

As a rule, they are the result of saving loads from a structural fragment as a separate model (see Sec. **Analysis of the Results of Calculation of Loads from a Structural Fragment**).

Dynamic actions of a submodel cannot be included into the static loadings of the main model.

Only masses will be added when combining dynamic loads of the submodel and the main model.

5.14. Copying a Design Model

Let's consider another effective method of creating a design model, the effect of which is based on copying a model-prototype with a given step and direction. You can copy a finished model, i.e. with the specified stiffness properties, hinges, constraints, loads, etc., or a model with only a few characteristics specified.

The copying mode is invoked by clicking the respective button in the **Model** section. The copying can be performed according to different rules.

The simplest case – a model is just repeated a given number of times. The control of duplicate nodes and elements is performed in this process, and if the **Delete duplicate elements** checkbox is checked, they are removed from the resulting model.

When copying a model with the selected nodes and elements the type of the selected objects is converted according to the following rules:

- if nodes have been selected in the model-prototype, a bar element connecting a node with its respective node in the copy is generated from each of these nodes;
- if bar elements have been selected in the model-prototype, each selected bar will generate a plate element (three-node — type 42 or four-node — type 44), the size of which in the direction of copying will be equal to the specified step of copying;
- if plate elements have been selected in the model-prototype, each selected element will generate a solid element (type 36, 33 or 34), the size of which in the direction of copying will be equal to the specified step of copying (see Sec. 5.16).

Rules for transformations of the selected objects will also hold for any combination of them.

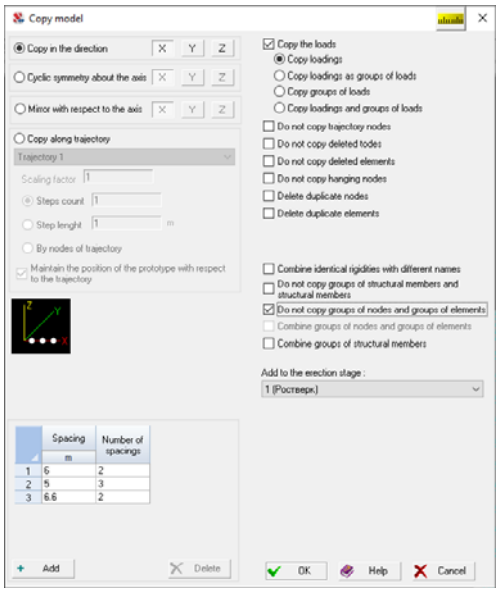


Figure 5.14-1. *The* **Copy the Model** *dialog box*

In those cases when a fragment of the design model is displayed in the window and this operation is invoked, a drop-down menu appears where you can select whether to copy the whole model or its visible fragment. In the first case, i.e. when the whole model is copied, the operation will be applied to all nodes and elements of the design model according to the above rules regardless of whether they are included in the visible fragment or not. When the visible fragment is copied, — the above actions will be performed only with the nodes and elements of the fragment.

Another type of copying is performed by mirroring the whole model or its visible part along the selected axis with respect to the origin of the global coordinate system. The operation will be performed according to the same rules as the copying in the specified direction.

The **Copy the Model** dialog box (Fig. 5.14-1) enables to specify the direction of copying:

- in a straight line along one of the axes of the global coordinate system;
- along a circular arc about the given axis (the axis of rotation passes through the origin of the global coordinate system);

- mirroring.

The steps of repetitions of prototypes and the number of repetitions for each step are specified for the first two operations as the parameters of copying (when copying along an arc a central angle is specified in degrees as a step).

The mirroring of the prototype is performed by changing the sign of the nodes' coordinates of the copy corresponding to the axis along which the mirroring is performed. Since the position of the copy in the result of mirroring depends on the position of the origin of the global coordinate system, it is necessary to move the origin into the right point before performing the operation or specify the offset of the origin in the corresponding input field.

Let's consider the rules of working with the copy. They are specified by checking the respective checkboxes and enable to perform the following operations:

- **Copy the loads**, i.e. transfer the loads from the model-prototype to the model-copy. There are following options for copying the loads:
  - **Copy loadings** — only the loadings specified in the prototype are transferred to the copies;
  - **Copy loadings as groups of loads —** only the loadings specified in the prototype are transferred and they are given in the copies as groups of loads;
  - **Copy groups of loads** — only the groups of loads specified in the prototype are transferred to the copies;
  - **Copy loadings and groups of loads** — both loadings and groups of loads are transferred to the copies.
- **Delete duplicate nodes** — if in the result of copying there are nodes with the same coordinates, only one of these nodes will be left in the resulting model. Two nodes are considered to be duplicate if the distance between them is less than the value specified by the user in the **Environment Settings** dialog box (see the ***Settings*** menu);
- **Delete duplicate elements —** if in the result of copying there are elements all nodes of which coincide, only one of these elements will be left in the resulting model;
- **Do not delete bars that produced plates and plates that produced solid elements** — this checkbox will be available if the model has selected bars or plates. If this checkbox is checked, the selected bars and plates are not deleted from the design model;
- **Do not change the original prototype** — this item enables to save the model-prototype without changes in those cases when bars producing plates are deleted;
- **Combine identical rigidities with different names** — rigidities are packed when copying the model; this checkbox enables to "affect" the packing process;
- **Do not copy groups of structural members and structural members** — if this checkbox is checked, the resulting model will contain only the structural groups of the original model;
- **Do not copy groups of nodes and elements** — if this checkbox is checked, the resulting model will contain only the original groups of nodes and groups of elements;
- **Combine groups of nodes and groups of elements** — the lists of groups of nodes/elements will be supplemented by the numbers of the corresponding nodes/elements from the created copies;
- **Combine groups of structural members** — this checkbox enables to add elements from the copies of the respective groups of steel and reinforced concrete structural members to prototype groups; this option does not apply to structural members.

Models can be copied with the generation of new elements and without the generation of new elements, the procedure in these cases is somewhat different.

When a model is copied *without the generation of new elements*:

- activate the copying mode by clicking the button in the **Model** section;
- specify the parameters of copying, rules for working with the copies, and click the **OK** button in the **Copy Model** dialog box (Fig. 5.14-1);
- after performing the copying confirm the results or reject them.

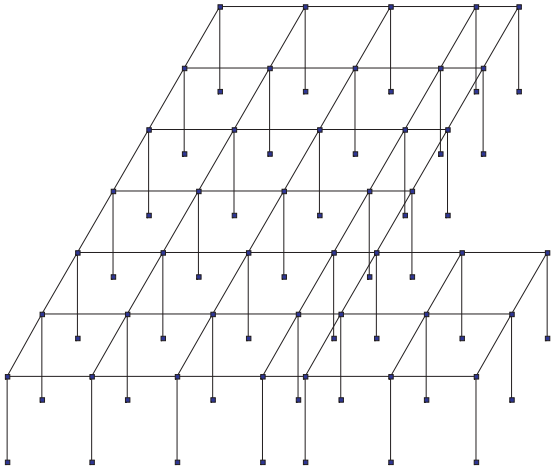


Figure 5.14-2. *Model-prototype*

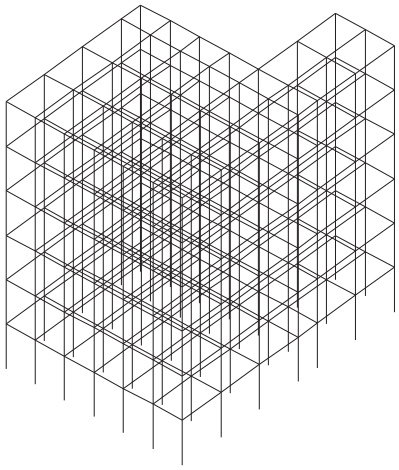
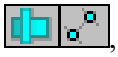


Figure 5.14-3. *Model after the copying operation*

When a model is copied *with the generation of new elements*:

- use a group of control buttons of selection of elements and nodes  , located in the left corner of the status bar (or in the **Control the Selection** panel), to select nodes bars should be generated from, bar elements which should produce plates, and plates solid elements should be generated from in the prototype;
- activate the copying mode by clicking the button in the **Model** section;
- specify the parameters of copying, rules for working with the copies, and click the **OK** button in the **Copy Model** dialog box;
- after performing the copying confirm the results or reject them.

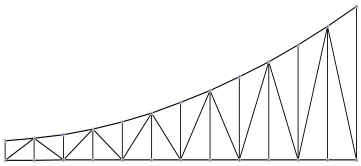


Figure 5.14-4. *Model-prototype*

Let's consider several examples of performing the copy operation. In the first case copy the model-prototype (Fig. 5.14-2) 5 times in the direction of the Z axis with a step of 3 m. Resulting model (rotated for better visualization by 60°) is shown in Fig. 5.14-3.

In the next example a bar structure with a parabolic upper chord is used as a model-prototype (Fig. 5.14-4).

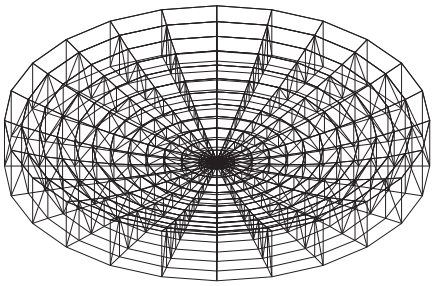


Figure 5.14-5. *Model-result of copying a prototype with the generation of bars*

Copying was performed 24 times by rotating about the Z axis by 15°. A bar was generated from each selected node. The result of the function is shown in Fig. 5.14-5.

If in addition to nodes you also select the bars of the upper chord in the model-prototype, then in the result of copying you will obtain a model shown in Fig. 5.14-6. Note, that each selected bar has produced a plate.

In the considered method of copying the operations are performed with the full design model, i.e. a fragment of the model can not be a prototype.

Questions of generation in the process of copying solid elements are considered in Sec. 5.16.

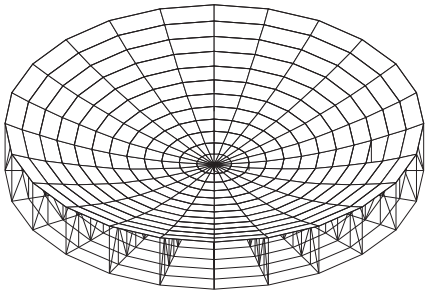
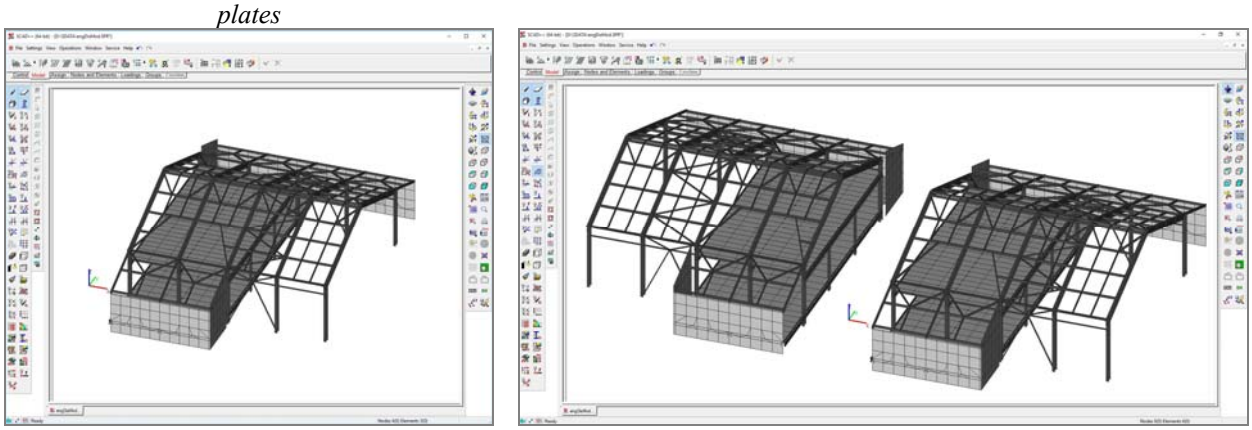


Figure 5.14-6. *Model-result of copying a prototype with the generation of bars and plates*



a) *b)*

Figure 5.14-7. Copying a fragment of the model (a) by mirroring with respect to the origin of the global coordinate system along the X axis (b)

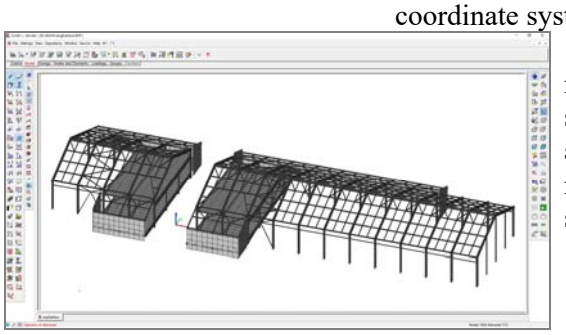


Figure 5.14-8. Result of performing copying

An example of copying the fragment of the model by mirroring is given in Figure 5.14-7. The resulting model is shown in Figure 5.14-8. It should be noted that the elements are generated from the selected nodes and elements in this mode according to the same rules as when copying in a straight line and along a circular arc.

5.15. Copying the Fragment of the Model

Besides copying a full model it is possible to copy its fragment as well. Any part of the model consisting of the selected elements can be considered as a fragment. When you copy a fragment the bars are not generated from the selected nodes, plates – from the selected bars, and solid elements – from the selected plates. The **Copy the Fragment of the Model** dialog box enables to specify the steps of repetitions of the selected fragment, the number of repetitions for each step, and the rules for working with the copy. When copying along an arc a central angle is specified as a step.

5.16. Creating Design Models from Solid Elements

The copy operation can be also used to create design models from solid elements. In this case a prototype is a model from plate elements. The following rule is used for the generation of solid elements: a plate selected on the prototype generates a solid element. If nodes, bars and plates are selected in the prototype, a complex model from bar, plate and solid elements will be created.

As an example let's consider the generation of a design model from solid elements on the basis of a prototype given in Fig. 5.16-1 (the prototype is obtained by automatic meshing of the specified area lying in the XOY plane).

Select all the elements of the prototype and activate the copy operation by clicking the button in the **Model** section. Specify the direction of copying — Z, number of repetitions of the prototype and the step in the **Copy the Model** dialog box. A design model shown in Fig. 5.16-2 will be generated after clicking the **OK** button.

You can also create a model by copying a prototype along a circular arc. The origin of the global coordinate system should be determined in such a way so that the axis of rotation does not intersect the elements of the prototype.

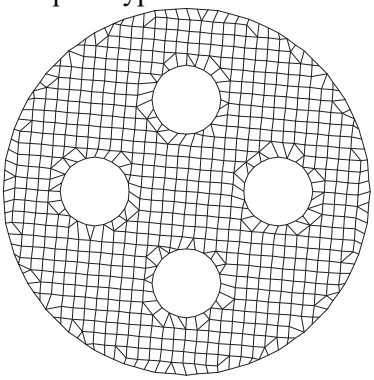


Figure 5.16-1. *Design model from plate elements used as a prototype*

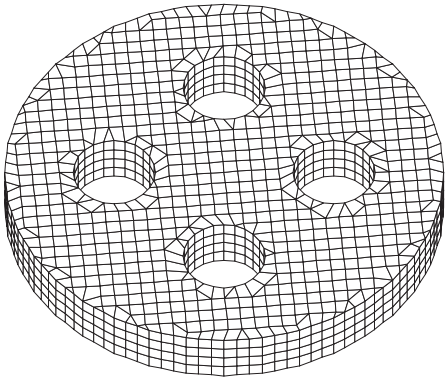


Figure 5.16-2. *Design model from solid elements*

5.17. Geometric Transformations

The set of geometric transformations enables to modify the geometry of the design model or of its fragment by changing the coordinates of nodes. The set includes: transfer of the coordinates of nodes, rotation about the given axis, scaling and mirroring.

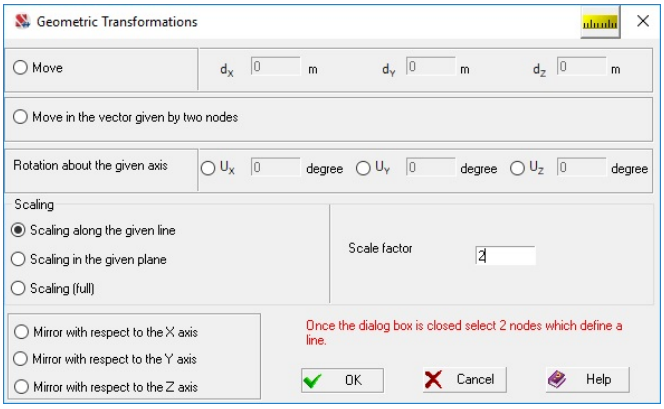


Figure 5.17-1. *The* **Geometric Transformations** *dialog box*

Clicking the respective button in the **Model** section of the toolbar invokes the **Geometric Transformations** dialog box (Fig. 5.17-1).

If the transformations refer to the whole model, they will be performed after clicking the **OK** button in the dialog box. If you want to transform a fragment (group of nodes), you have to first select nodes, and then assign and perform the necessary operation.

Moving

This operation enables to move nodes on the specified distance from the initial position. If no nodes are selected in the model, the transfer will be performed for the whole model. In those cases when a group of nodes is selected in the model, only the nodes of the group will be moved. In order to perform this function activate the **Move** option, specify the increments of coordinates and confirm them by clicking the **OK** button in the dialog box.

Moreover, you can also use the moving operation, where the distances are defined by a vector given by two nodes. In order to perform such a moving activate the respective option in the dialog box and click the **OK** button. If you click on the first node of the vector and stretch the rubber band to the second node, the moving will be performed after clicking on the second node.

Rotation about the Given Axis

This operation enables to rotate a model or a group of nodes by the specified angle about the given axis. The procedure is standard — activate an option with the name of the respective axis, specify the angle and click the **OK** button. In order to obtain a correct result you have to control the position of the origin of the global coordinate system about the axes of which the rotation is performed. The origin of the coordinate system can occupy any place in the model and even outside of it, and it can be moved into the given node by clicking the **Move the origin into the given node** button in the **Nodes and Elements** section (if there is no node, it can be created).

Despite its apparent simplicity, this operation has a lot of "pitfalls". When rotating a group you have to make sure that there are no intersections of the rotated elements with the stationary part of the model, plate elements may change their shape and lose their flatness, and the incorrect specification of the origin may lead to an unexpected result.

Scaling along the Given Line

This operation is used to change the sizes of the design model or its fragment specified by indicating the nodes along the given direction. The direction of scaling is defined by a line passing through two nodes. In order to perform the scaling you have to enable the respective radio button in the dialog box, specify the scaling factor (positive number) and click the **OK** button. If you click on the first node of the line which defines the direction of scaling and stretch the rubber band to the second node, the scaling will be performed after clicking on the second node. When specifying a line the nodes defining it do not necessarily have to belong to the set of nodes the scaling is performed for. This line defines only the direction.

Do not forget to check the results of the operation to make sure that there are no intersections of elements and they have correct shapes.

Scaling in the Given Plane

The work with this operation is similar to the previous one. Scaling is performed for the whole model or the selected group of nodes in the planes parallel to the given plane. The following procedure is recommended.

For the whole model:

- Enable the **Scaling in the given plane** radio button in the dialog box;
- specify the scaling factor;
- click the **OK** button;
- select three nodes in the model which lie in the scaling plane.

After selecting the third node the scaling is performed automatically.

For a group of nodes:

- select nodes in the model the scaling is performed for;
- invoke the mode of geometric transformations;
- enable the **Scaling in the given plane** radio button in the dialog box;
- specify the scaling factor;
- click the **OK** button;
- select three nodes in the model which lie in the scaling plane or in a plane parallel to it.

After selecting the third node scaling is performed automatically.

Mirroring with Respect to the Given Axis

These operations enable to mirror the whole model or its part selected as a group of nodes with respect to the selected axis. As in the previous cases you should not forget about the dangers due to the possible change of direction of the local axes of the elements.

An express-analysis of the design model should be performed after the geometric transformations (button in the **Control** section of the toolbar). It detects bar elements of zero length and plate elements of zero area.

Scaling (Full)

It performs the scaling of the model or of its part selected as a group of nodes in all directions along the axes of the global coordinate system in accordance with the specified scaling factor.

Scaling will be performed automatically after clicking the **OK** button in the dialog box.

Examples of Geometric Transformations

Let's consider a few examples of using geometric transformations related to the operations of rotation, scaling and mirroring.

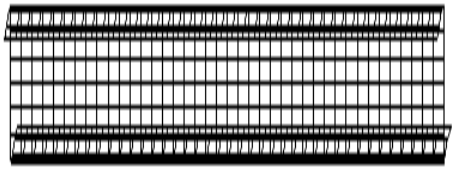


Figure 5.17-2. *Initial model of the beam*

Example 1

Let's consider the rotation operation on an example of a Z-shaped beam which will be obtained from the initial model in Fig. 5.17-2.

"Bend" the beam by 12º about the Z axis at the distance of 10 m from the left edge of the beam. To do this, perform the following actions:

- move the origin of the global coordinate system into one of the nodes lying on the bend line;
- select nodes from the bend line to the right edge of the beam;
- invoke the **Geometric transformations**;
- invoke the operation of rotation about the Z axis by 12º;
- click the **OK** button.

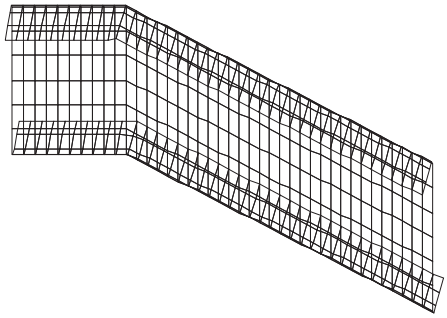


Figure 5.17-3. *Model of the beam after rotating the group of nodes by* 12º

The model obtained in the result of the above actions is shown in Fig. 5.17-3.

Repeat the performed operations for the last ten meters of the beam rotating them by 12º. The result is given in Fig. 5.17-4.

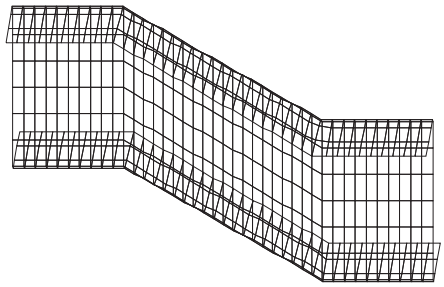


Figure 5.17-4. *Model of the beam after rotating another group of nodes by* 12º

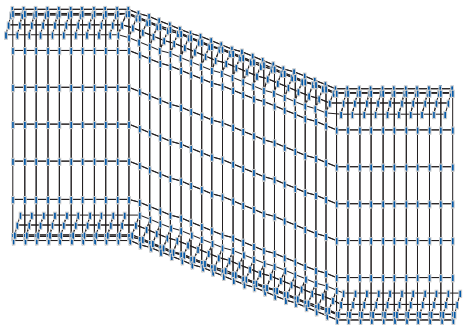


Figure 5.17-5. *Model of the beam in the result of scaling along the given line*

Example 2

Let's consider the operation of scaling along the given line on the example of the same beam. Double the height of the beam. To do this, perform the following actions:

- invoke the **Geometric transformations**;
- invoke the operation of scaling along the given line;
- specify the scaling factor equal to 2, and click the **OK** button;
- connect two nodes of the web lying on the same vertical with the rubber band.

The model will be automatically transformed after clicking on the second node (Fig. 5.17-5).

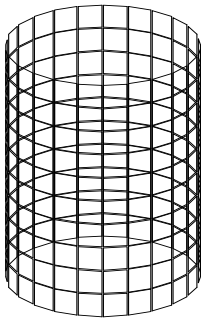


Figure 5.17-6. *Initial model of the cylinder*

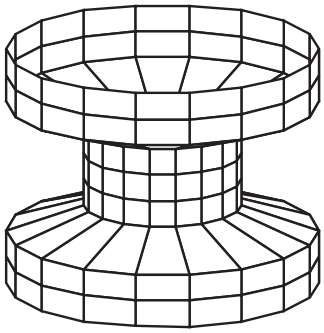


Figure 5.17-7. *Model of the cylinder in the result of scaling*

Example 3

Use the design model of a cylinder (Fig. 5.17-6) to illustrate the possibilities of scaling in the given plane. Double the diameter of two upper and two lower chords of the cylinder.

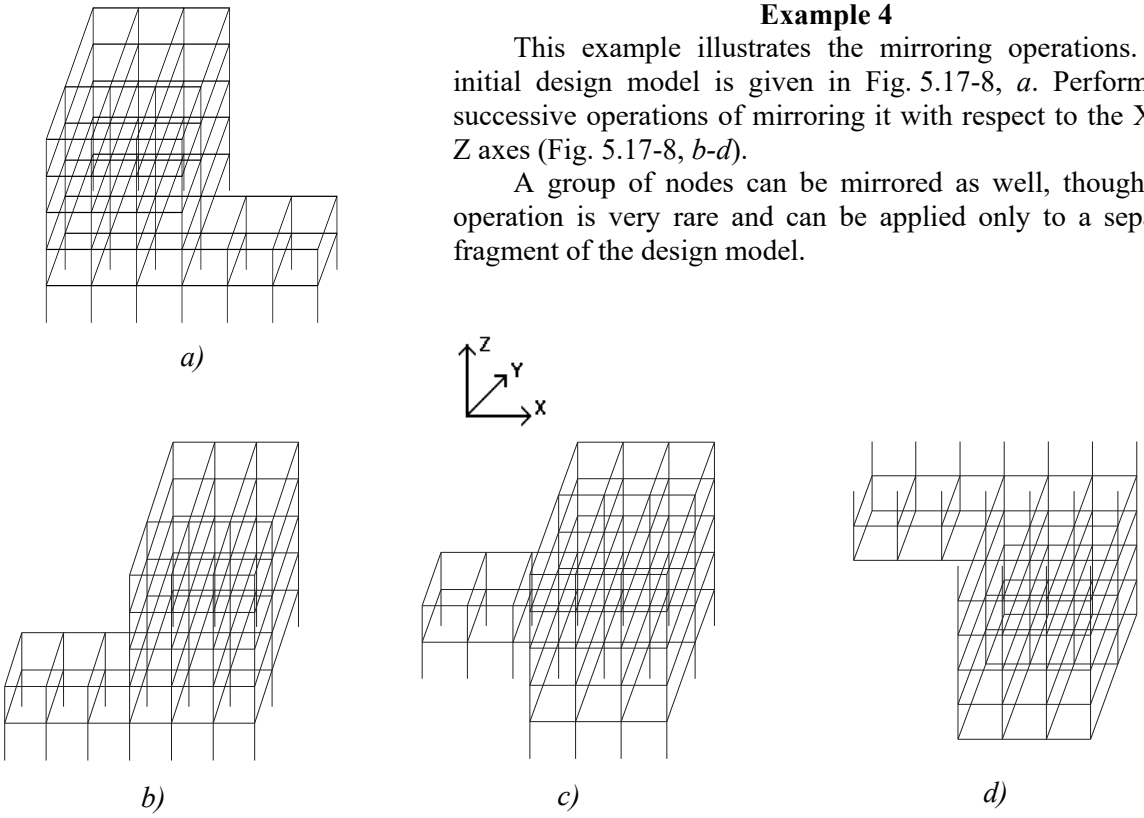
To do this, perform the following actions:

- select nodes of these chords;
- invoke the **Geometric transformations**;
- invoke the operation of scaling in the given plane;
- specify the scaling factor equal to 2;
- click the **OK** button;
- select three nodes in the model which lie in the scaling plane (in this case in the horizontal plane).

Operation will be performed after selecting the third node.

Model-result of geometric transformations is shown with removal of lines of the invisible contour in Fig. 5.17-7.

Please note that trapezoidal elements have appeared after performing the scaling operations. If these elements were specified as rectangular (type 41) in the model-prototype, they have to be changed to type 44.



Example 4

This example illustrates the mirroring operations. The initial design model is given in Fig. 5.17-8, *a*. Perform the successive operations of mirroring it with respect to the X, Y, Z axes (Fig. 5.17-8, *b-d*).

A group of nodes can be mirrored as well, though this operation is very rare and can be applied only to a separate fragment of the design model.

Figure 5.17-8. *Initial design model* (*a*) *and the results of successive mirroring with respect to the* X, Y, Z axes (*b–d*).

5.18. Specification of Grid Lines

A design model or its part can be created using grid lines. The program enables to specify grid lines consisting of several sections, each of which may have one of the following four types: rectangular, rectangular rotated by a given angle, oblique or polar. Elevations are common to all sections and are specified once when describing the first section.

Parameters of the axes are specified in the multi-tab **Specify Grid Lines** dialog box (Fig. 5.18-1).

Checkboxes for specifying the rules for display of the marks of axes

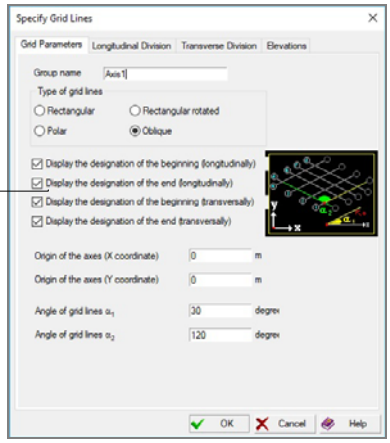


Figure 5.18-1. *The* **Specify Grid Lines** *dialog box* (*the* **Grid Parameters** *tab*)

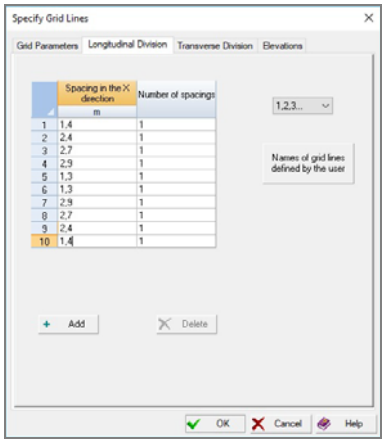


Figure 5.18-2. *The* **Specify Grid Lines** *dialog box* (*the* **Longitudinal Division** *tab*)

The **Grid Parameters** tab enables to specify the type of grid lines and the name of the section, as well as the coordinates of the snap point of the section of grid lines in the global coordinate system of the model. The rules for displaying the marks of grid lines are defined by the checked checkboxes. Moreover, depending on the type of grid lines the following data can be specified in this tab:

- rectangular grid rotated by a given angle — the angle of rotation of the section with respect to its snap point (hereinafter positive direction – counterclockwise);
- oblique grid — the angle of rotation of the grid section with respect to its snap point and the inner angle;
- polar grid — the angle of rotation of the X axis with respect to the snap point of the section.

The **Longitudinal Division** (Fig. 5.18-2) and **Transverse Division** tabs are used to specify data necessary to generate grid lines in the Y-direction and in the X-direction respectively. Division along the Z axis and the snap of the elevation of the lower level (reference mark) are specified in the **Elevations** tab.

Names of grid lines can be generated automatically or specified by the user. It is assumed by default that longitudinal grid lines are labeled with numbers in the ascending order from 1, and transverse ones – with letters in alphabetical order. The drop-down list in the tabs of longitudinal and transverse division can be used to change the order assumed by default. The list includes three items “1, 2, 3…”, “A, B, C…” and “Specified”. For example, if you select the "A, B, C ..." item in the list for the longitudinal grid lines without changing it for the transverse ones, then both grid lines will be labeled in the same way – with letters in alphabetical order.

In order to specify the labeling order different from the default one you have to specify new names, which may include up to three characters, in the **Grid Line Names** dialog box, which can be invoked by clicking the **Names of grid lines defined by the user** button (Fig. 5.18-5). The “Specified” item will be selected automatically if any changes have been made to the “standard” labeling of grid lines.

The filter — **Display grid lines** is used to control the display of grid lines. The operations of control of display are described in Sec. 10.2. It should be noted here that the software provides different variants of the output of grid lines: all sections simultaneously or in any combination of them, entirely or selecting a fragment of any section, etc.

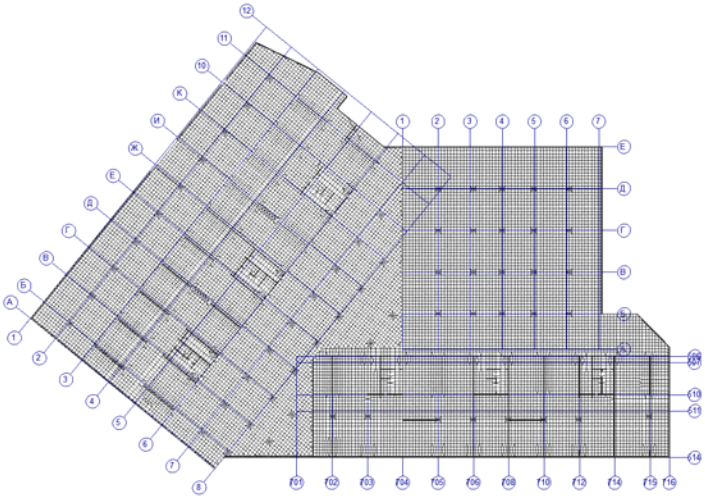


Figure 5.18-3. *Fragment of the design model with grid lines*

In order to specify a new section of grid lines you have to invoke the **Specify grid lines** operation and repeat the above actions in the respective dialog box. The opened dialog box will display data on the grid section specified last. It should be replaced with new information.

In order to adjust the parameters of the grid line section specified earlier activate the operation of specification of grid lines using the arrow on the right of the button to drop down the menu and select the **Grid line control** section in it. The **Grid Line Control** dialog box (Fig. 5.18-4) includes a table with a list of specified sections of grid lines. Perform the following actions to adjust the parameters of the grid:

- select a line with the modified grid in the table;
- click the **Modify** button;
- make all the necessary changes in the multi-tab **Specify Grid Lines** dialog box (Fig. 5.18-1);
- close the dialog box by clicking the **OK** button;
- once all the adjustments are performed close the **Grid Line Control** dialog box by clicking the **OK** button.

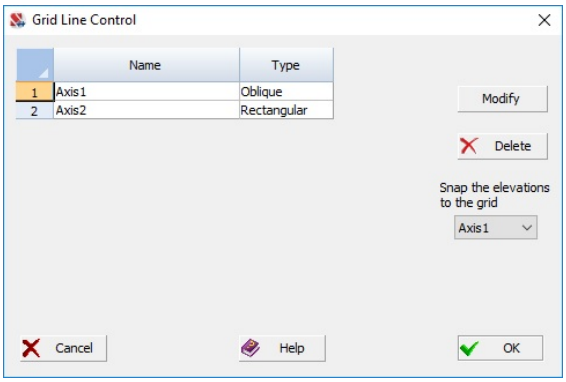


Figure 5.18-4. *The* **Grid Line Control** *dialog box*

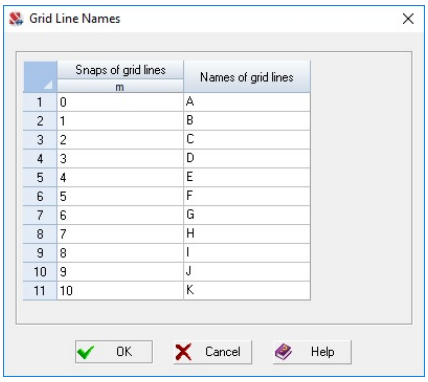


Figure 5.18-5. *The* **Grid Line Names** *dialog box*

5.19. Creating a Model on Grid Lines

Grid lines and elevations can be used when specifying the geometry of the design model. To do this, you have to snap the nodes to the grid. Button invoking the operation of generation of nodes is located in the **Nodes** section of the **Nodes and Elements** toolbar. This operation enables to create nodes at the intersection of grid lines in the specified area of the grid. Elevations are considered as Z coordinates. The added nodes can be used for all input operations provided in the software. They are the basis for creating

the area of meshing, input of bar, plate and solid elements, and are involved in the operations of geometric transformations and assembly (Fig. 5.19-1).

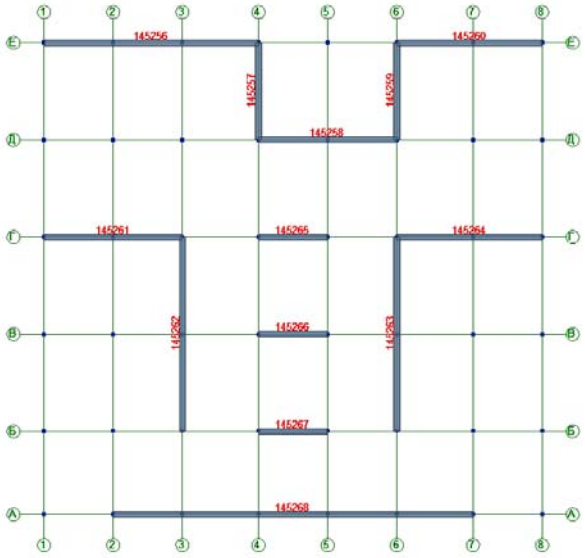


Figure 5.19-1. *Bar elements created on grid lines*

5.20. Copying the Model along Trajectories

Trajectories

This section considers the operations of creating, adjusting and using the trajectories during the generation of the design model.

The trajectory is represented by a set of nodes arranged in a certain order. In order to make it convenient to represent the trajectory in the graphical editor, the nodes are connected by segments, which are not considered as independent objects, and do not affect the characteristics of the design model. The trajectory is an auxiliary element (type 1101) which can be generated both when creating a design model, and attached to the model using the assembly operation.

The main purpose of the trajectory in the process of the generation of the geometry of the design model is to specify the direction in which the copied prototype has to be moved. Most of the other parameters are identical to those in the operations of copying in a straight line.

Creating a Trajectory and Its Application to the Existing Model.

It is recommended to perform the following operations in order to generate a trajectory by snapping its nodes to the existing nodes of the design model:

- invoke the operation by clicking the **Trajectory** button in the **Model** toolbar and select the method of specifying the trajectory in the drop-down menu, – by selecting the nodes in the order of their arrangement (**Create a new trajectory**), or selecting the bar elements of the model coinciding with the segments of the trajectory (**Create a new trajectory from the selected bars**);
- if you select the first operation, the dialog box (Fig. 5.20-1) will appear, where you have to specify the name of the trajectory (the **Name** field) and close the dialog box by clicking the **OK** button;
- select the nodes in the model in their order using the “rubber band” and click the **OK** button in the **Model** toolbar (the “marquee” and “polygon” cursors are not used for the simultaneous selection of several nodes, because they can not clearly define their order);
- a similar procedure is followed to create a trajectory using the bar elements; the name of the trajectory is specified in the **Create a New Trajectory** dialog box, bar elements are selected after closing the dialog box and the **OK** button is clicked in the **Model** toolbar (it does not matter is

which order the bars are selected, because it is assumed that the segments connecting the neighboring nodes of the trajectory form a continuous broken line).

Peculiarities of the Implementation

The finite elements and the trajectory should not use common nodes. In this case moving the trajectory can change the design model. If the peculiarities of the design model do not restrict the use of the common nodes of the model and the trajectory, the trajectory can be specified taking into account the existing nodes. Otherwise, you have to check the **Separate the trajectory nodes from the nodes of the model** checkbox in the **Create a New Trajectory** dialog box, and in the result in the process of creating a trajectory a new node belonging only to the trajectory will correspond to each node of the design model used when creating the trajectory.

In the case when the bar elements which do not form a continuous broken line are selected, the software generates a message. The user can make a decision to create several trajectories on the basis of one set of the selected bars. In the case of a positive answer to the inquiry about the possibility of creating multiple trajectories, they are generated automatically. Each trajectory is given a name specified during the activation of the operation, supplemented by a number in the order of creating the element. The first created trajectory gets the initial name.

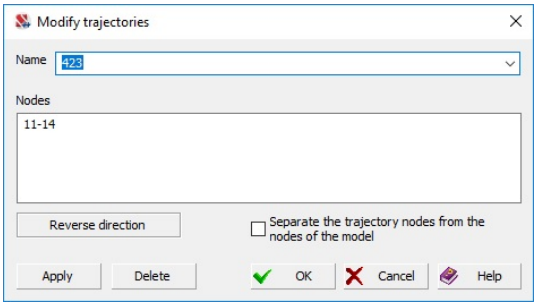


Figure 5.20-1. *The* **Modify trajectory** *dialog box*

Copying the Model Using the Trajectory as a "Director"

An important property of the trajectory is the possibility to use it as a "director" which indicates the direction of copying. Similarly to finite elements, the copying along a trajectory enables to generate one-dimensional elements from the selected nodes, two-dimensional elements from bars and three-dimensional elements from the two-dimensional (shell) elements. Moreover, when copying along the trajectory, there is a possibility of changing the size of the copy upward or downward depending on the ratio between the scaling factors at the beginning and at the end of the trajectory.

When invoking the operation of copying along the trajectory, you should follow the same rules as when copying in the direction of the axes of the main coordinate system:

- the operation of copying the model can be invoked by clicking the respective button in the **Model** toolbar;
- select the type of copying in the drop-down list – either the whole model or only its visible part;
- check the **Copying along the trajectory** checkbox in the **Copy the Model** dialog box (Fig. 5.20-2), specify the name of the trajectory by selecting it from the list of trajectories pre-attached to the project;
- if during the copying process it is necessary to change the sizes of each subsequent copy, you should specify the scaling factor (Ks) other than one. The sizes of the copy will change upward (Ks> 1) or downward (Ks <1) proportionally to the covered path if the **Step size** checkbox is checked, or to the number of performed steps along the trajectory if the **Number of steps** checkbox is checked.

When a model with the selected nodes and elements is copied, the type of the selected objects is transformed according to the following rules:

- if the nodes were selected in the model-prototype, a bar element is generated from each such node connecting it to the node-analogue in the copy;
- if bar elements were selected in the model-prototype, each selected bar will generate a plate element (3-node – type 42 or 4-node – type 44), the size of which in the direction of copying will be equal to the specified step of copying;
- if plate elements were selected in the model-prototype, each selected element will generate a solid element (type 36, 33 or 34), the size of which in the direction of copying will be equal to the specified step of copying (see Sec. 5.16).

The rules of transformation of the selected objects will be executed for any of their combination as well.

Regardless of the variant of copying along the trajectory the copies can be arranged preserving their initial position with respect to the trajectory. It can be done by checking the respective checkbox.

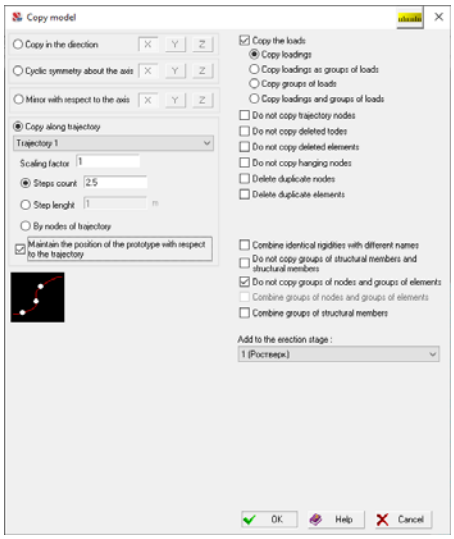


Figure 5.20-2. **Copy Model** *dialog box*.

5.21. Graphics Editor

Graphics editor, which is invoked by clicking the button , enables to specify a design model of a bar structure or of its fragment using a set of graphics operations similar to those in the 2D AutoCAD (AutoCAD Light) environment. The authors of the system were not trying to replace such recognized graphical tools as AutoCAD, Micro Station, and the like. The goal was more modest – to enable the experts and amateurs of these graphic systems to work in a familiar environment. The graphics editor is described in detail in Sec. 33.8.

6. Operations with Nodes and Elements

When creating a design model it is often necessary to modify it by adding new or deleting the existing nodes and elements, changing their position in the model, dividing elements, etc. The **Nodes and Elements** section of the toolbar contains operations which enable to perform the above actions.

Any activated operation with nodes and elements will be active until we cancel it by clicking the button in the **Nodes and Elements** toolbar or select another operation.

6.1. Selection of Nodes and Elements

Operations are performed with the selected nodes and elements in the graphical environment of **SCAD**. Several methods of selecting are implemented, i.e.: with the help of different types of cursors, from the color scales, from the tables and from the list of messages.

Objects are usually selected with cursors. A cursor with a target is used to select one object. Several objects can be selected at the same time with a marquee or a polygon. A cursor can be selected in the **Control the Selection** toolbar (Fig. 6.1-1) or from the menu (Fig. 6.1-2,a), which appears after right-clicking in the work area of the window.

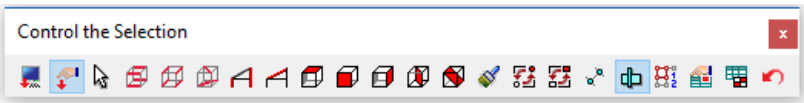


Figure 6.1-1. *The* **Control the Selection** *toolbar*

Buttons of the toolbar have the following purpose:

— **immediate execution** — this option is provided for experienced users and invokes the immediate execution of operations after selecting the objects. It applies to all types of operations, including visualization, and enables to perform them without confirming with the **OK** button in the toolbar or the **Enter** key. When working in this mode, it is recommended to use the display filters that correspond to the performed operations which will enable to control the correctness of their performance;

— **execution with confirmation** — this option is active by default and invokes the execution of most operations only after confirming them with the **OK** button in the toolbar or the **Enter** key;

— **select cursor** — clicking this button drops down a menu with a list of available cursors you can select from;

, , , , — **filters of the selection of horizontal (parallel to the axis X, parallel to the axis Y, nonparallel to the axes X and Y), vertical and inclined bars** respectively — operations of selection will apply only to the bar elements, the position of which in the model corresponds to the activated options (filter buttons may be activated in any combination, if they are all "depressed", then all objects are selected);

, , , , — **filters of the selection of plates lying in the XOY, XOZ, YOZ plane, and of vertical and inclined plates** respectively — operations of selection will apply only to the plate elements, the position of which in the model corresponds to the activated options (filter buttons may be activated in any combination, if they are all "depressed", then all objects are selected);

— **reset data input filters** — all filters are reset to the default initial state;

— **invert the selection of nodes** — is used to simultaneously change the state of selection of all the nodes visible on the screen to the opposite one (nodes selected earlier will be deselected, and all the other ones will be selected);

— **invert the selection of elements** — is used to simultaneously change the state of selection of all the elements visible on the screen to the opposite one (elements selected earlier will be deselected, and all the other ones will be selected);

— **select a group of elements** — activates the selection of elements;

— **select a group of nodes** — activates the selection of nodes;

— **select objects in the model from the tables** — this option enables to select the objects in the design model by indicating them in the interactive tables (with the initial data and the results);

— **display only the selected objects in the tables** — the interactive tables will display only the information on the objects selected in the design model;

— **select objects for displaying information** — this option is used in those cases when the **Display information on the model only for the selected elements and nodes** checkbox is checked in the settings. In this case, a special selection operation is used which is not related to the execution of any actions with the selected objects, except for the output of the related information on the model;

— **previous selection** — select previously selected objects.

Selection of the cursor and the settings of the selection control options can be also performed with the help of the menu. It is convenient to use a menu in cases when only one action is performed, such as the selection of the type of cursor or activation of one option. This is due to the fact that after selecting the necessary item the menu is closed.

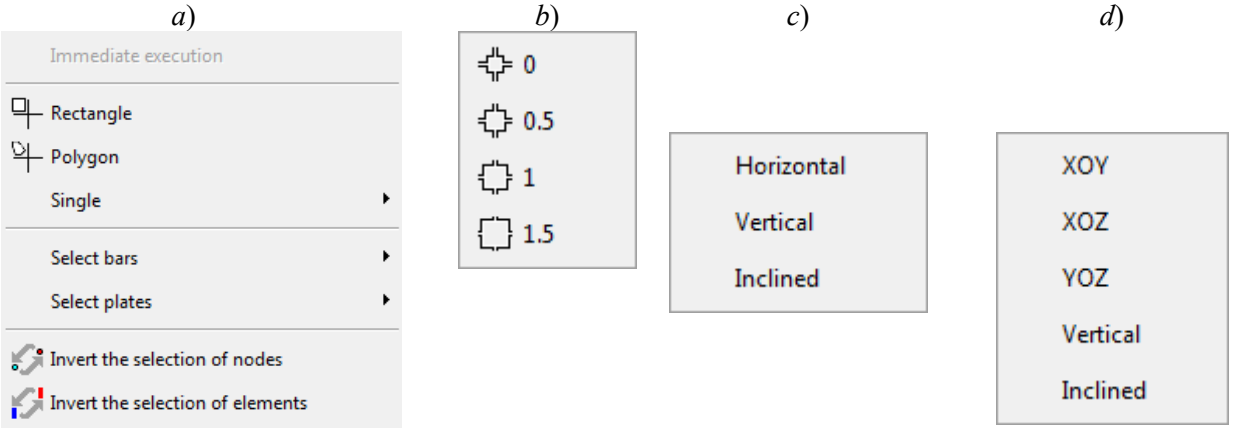


Figure 6.1-2. *a) Menu of the control of cursors;*
b) selection of the accuracy of positioning the cursor with a target; c) filters of the selection of bars;
d) filters of the selection of plate elements

The accuracy of positioning the cursor for a single selection is defined by the size of the screen area covered by the cursor target. The minimum size of the target is approximately equal to the default size of the image of the node in the design model. Such a "precise" indication is usually used in dense design models, where the distance between the nodes or the sizes of elements are commensurate with the size of the target. Values 0.5, 1 and 1.5 (size of the target is increased by 50, 100 and 150% respectively) enable to increase the area of positioning of the cursor when selecting the object.

When selecting nodes or elements with the mouse, you can use the **Shift** and **Ctrl** keys, which change the rules for selecting nodes/elements. When the **Shift** key is pressed, the previously selected nodes/elements are not deselected, and the nodes/elements which were not previously selected become selected. When the **Ctrl** key is pressed, the previously selected nodes/elements are deselected, and the nodes/elements which were not previously selected are not selected.

Right-clicking on the Select nodes/elements buttons (,) invokes a menu which enables to select nodes belonging to the selected elements or select elements adjacent to the selected nodes.

Objects can be also selected by rectangular and polygonal marquees. Nodes will be selected if they are within the marquee area. In order to select elements, all their nodes have to be within the marquee area. Rigid bodies are an exception. In order to select it, you just have to indicate one of its nodes or lines

connecting the master node and one of the slave nodes. The selected nodes and elements are highlighted in red.

The following rules for working with cursors have to be taken into account when selecting nodes and elements:

- the assigned combination of filters of cursors is not reset automatically, and the filters will be active until you change or cancel the specification. The current state of filters is displayed by icons in the **Control the Selection** toolbar;
- the selected type of cursor will be active until a new operation is selected;
- when selecting objects you can change the type of cursor many times;
- clicking on the selected object for the second time deselects it.

The **Invert the selection of nodes** and **Invert the selection of elements** operations are used to select all the nodes or elements visible on the screen respectively.

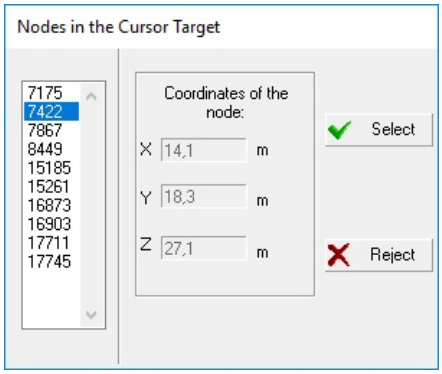


Figure 6.1-3. *The* **Nodes in the Cursor Target** *dialog box*

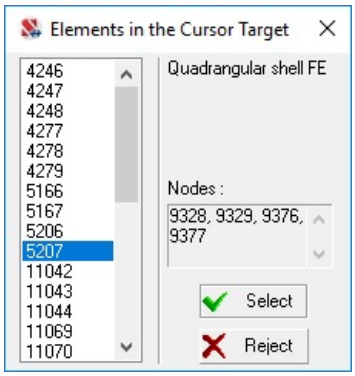


Figure 6.1-4. *The* **Elements in the Cursor Target** *dialog box*

If when selecting one node or element, there are several objects in the cursor target at once, all of them will be given in the lists in the **Nodes in the Cursor Target** or **Elements in the Cursor Target** dialog box (Fig. 6.1-3 and 6.1-4). In order to select a particular node or element you have to indicate it in the respective list and click the **Select** button. If several nodes have close but not equal coordinates, a node should be selected according to the values of its coordinates which are displayed in the respective fields of the **Coordinates of the node** group. Coordinates of nodes are displayed in the output fields with the accuracy and in the units of measurement specified in the **Units of Measurement** tab of the **Parameters** dialog box (see Sec. 34.1), and can be changed.

6.2. Operations with Nodes

Operations with nodes are invoked from the **Nodes** section (Fig. 6.2-1) of the **Nodes and Elements** toolbar.

Figure 6.2-1. *The* **Nodes** *section of the* **Nodes and Elements** *toolbar*

The following operations with nodes are implemented in the software:

— delete nodes;

— restore deleted nodes;

— create nodes;

— create additional nodes between nodes;

— move nodes;

— merge duplicate nodes;

— generate nodes along an arc;

— move the origin;

— create nodes at the specified distance from the selected ones;

— replace a node in elements;

— select nodes;

— generate nodes at the intersection of grid lines;

— pack data;

— generate generators by the formula;

— move nodes to the given plane;

— round the node coordinates;

— renumbering nodes;

— copy node properties.

The **Nodes** filter button is used to display nodes on the design model.

Create Nodes

This operation enables to create one or more nodes located at the same distance from each other by specifying their coordinates. Initial data are specified in the **Create Nodes** dialog box, which appears on the screen after invoking the operation (Fig. 6.2-2). A special feature of this window is that it does not close and remains available all the time when the operation of input is used.

When the **Create nodes** operation is invoked the **Nodes** visualization filter for displaying nodes on the screen is enabled automatically.

In order to create one node you just have to specify its coordinates in the fields of the left part of the window and click the **Add** button.

The presence of the repeater enables to create a group of nodes at once. In order to do it, perform the following actions:

- specify the coordinates of the first node of the group (X, Y, Z) in the text fields of the coordinates;
- check the **Repeat** checkbox;
- specify the number of repetitions in the **N** field;
- specify the values of the step of repetition in one or more directions (ΔX, ΔY, ΔZ) in the text fields for the increments of coordinates;
- click the **Add** button.

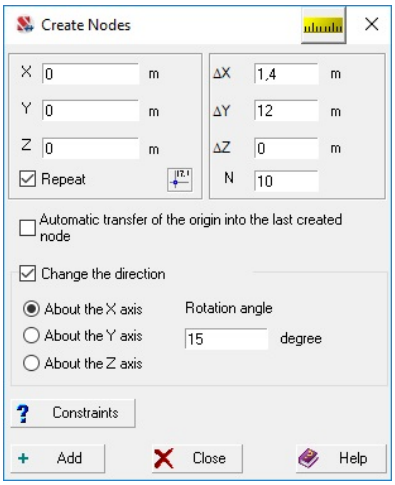


Figure 6.2-2. *The* **Create Nodes** *dialog box*

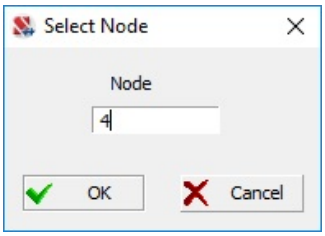


Figure 6.2-3. *The* **Select Node** *dialog box*

Nodes are created in the axes of the global coordinate system of the design model. The **Automatic transfer of the origin into the last created node** option moves the origin and the coordinates are now measured from this node.

You can specify the nodes that lie on a straight line, the position of which is defined by the coordinates of the first node and the increment of these coordinates. This line can be also rotated about one of the axes of the global coordinate system. The center of rotation is at the origin of the global coordinate system. Changing the direction of input of nodes is invoked by the respective option and enables to rotate the nodes by the given angle about the specified axis of the global coordinate system. If a group of nodes is input using a repeater, all the nodes of this group will lie on a straight line passing through the origin and rotated by a given angle.

Constraints can be specified in the nodes simultaneously with the input. The respective button is used for this purpose, clicking which invokes the **Constraints** dialog box (see Sec. 7.4).

The operation of selecting coordinates of a node enables to obtain the coordinates of the node by specifying its number in the **Node number** field of the **Select Node** dialog box (Fig. 6.2-3), and is used in cases when the increments are measured from the node created earlier. Once this dialog box is closed the coordinates of the node will appear in the **X**, **Y**, **Z** text fields.

Delete Nodes

It is necessary to perform the following actions to delete the nodes:

- invoke the **Delete nodes** operation;
- select the nodes you want to delete in the design model (the selected node is displayed in red);
- click the **OK** button in the **Nodes** section of the toolbar or press the **Enter** key.

Nodes will be deleted together with the elements adjacent to them. If you have mistakenly deleted the “wrong” node, it can be restored using the **Cancel** operation.

Despite the fact that the OK button has been already used a number of times, we will remind you that only your "own" button can be used to perform this operation, i.e. set in the respective section of the toolbar.

Restore Deleted Nodes

It is necessary to perform the following actions to restore the deleted nodes:

- invoke the operation, the filter displaying the deleted nodes on the model will be enabled automatically;
- select the nodes you want to restore (they will be displayed in red);
- click the **OK** button in the **Nodes** section of the toolbar or press the **Enter** key.

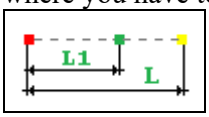
Note that this operation restores only the nodes. In those cases when the nodes have been deleted together with the elements connected with them, they can be restored with the help of the similar operation in the **Elements** group of buttons.

Deleted nodes can be made visible on the model without invoking the operation of restoring. Click the filters button , and the deleted nodes will be highlighted in yellow (or other color specified in the settings).

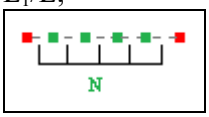
Deleted nodes can be restored only if the packing of data was not performed after they had been deleted.

Create Additional Nodes between Nodes

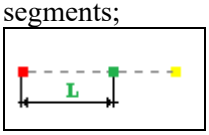
This operation enables to add new nodes on the line passing through two selected nodes. After invoking the operation the **Create Additional Nodes between Nodes** dialog box appears (Fig. 6.2-4), where you have to select one of the following five operations:



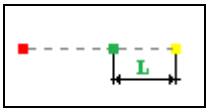
— create one node dividing the interval between two selected nodes in the given ratio L_1/L;



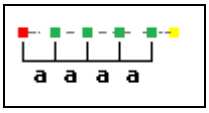
— create several nodes dividing the interval between two selected nodes into N equal segments;



— create one node at the given distance from the first selected node;



— create one node at the given distance from the second selected node;



— dividing the interval between two selected nodes into equal segments of the given length (the remainder is attached to the second selected node).

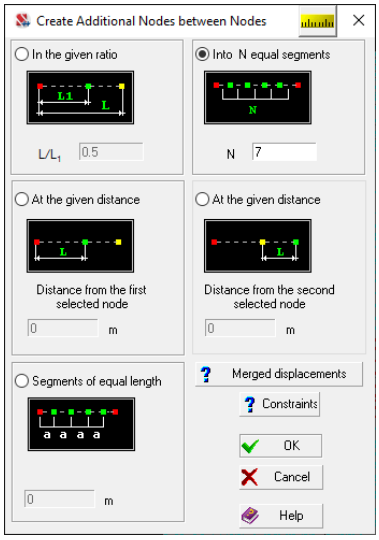


Figure 6.2-4. *The* **Create Additional Nodes between Nodes** *dialog box*

In order to select the operation enable the respective radio button in the dialog box and specify the numerical characteristics. It is necessary to perform the following operations after you close the dialog box by clicking the **OK** button:

- select the first node (it is significant which node is selected first for the first, third, fourth and fifth variants);
- select the second node;
- click the **OK** button in the **Nodes** section of the toolbar or press the **Enter** key.

In the icons of the operations where the order of the selection of nodes is important the first selected node is shown in green, and the second one – in yellow. Constraints can be specified in the nodes simultaneously with the input. The respective button is used for this purpose, clicking which invokes the **Constraints** dialog box (see Sec. 7.4). Similarly, the **Merged displacements** button enables to merge all newly created nodes and selected nodes into a group of merged displacements (the required directions are selected in the respective dialog box).

This operation enables to create nodes. When the new nodes are created on bar elements, the latter are not divided. Special operations in the Elements toolbar are used to divide bars.

Move Nodes

This operation is used to move a node into the point with the specified coordinates (to actually change the coordinates of the node) or to move several selected nodes in the given vector.

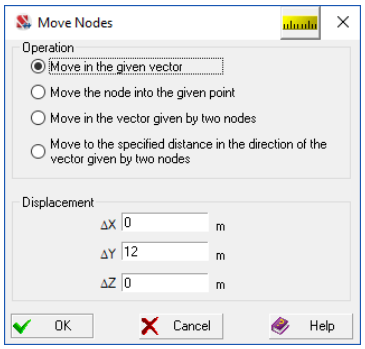


Figure 6.2-5. *The* **Move Nodes** *dialog box*

When moving several nodes each i-th node is moved in space into the point determined by adding the current coordinates of this node (X_i, Y_i, Z_i) with increments specified for each direction (ΔX, ΔY, ΔZ). In the result all the selected nodes are moved over the given distance. The parameters of transfer are specified in the **Move Nodes** dialog box (Fig. 6.2-5).

It is necessary to perform the following actions to move the nodes:

- click the **Move nodes** button in the **Nodes** section of the toolbar;
- the **Move Nodes** dialog box will appear where you have select the type of operation using the respective radio buttons: **Move in the given vector** or **Move the node into the given point** (when the second type of operation is performed, one node is moved);
- depending on the selected operation specify the increments to the coordinates or new coordinates of the node;
- close the dialog box by clicking the **OK** button;
- select nodes or a node in the model;
- click the **OK** button in the **Nodes** section of the toolbar or press the **Enter** key.

In the example given in Fig. 6.2-6 the selected nodes are moved in the positive direction (same with the direction of the axis) along the X axis.

Moreover, you can also use the move operation, where the increments of coordinates are defined by a vector given by two nodes. In order to perform such a movement activate the respective option in the dialog box and click the **OK** button. Select the nodes you want to move and click the **OK** button in the **Nodes** section of the **Nodes and Elements** toolbar (or press the **Enter** key). Then click on the first node of the vector and stretch the rubber band to the second node; the movement will be performed after clicking the **OK** button in the toolbar.

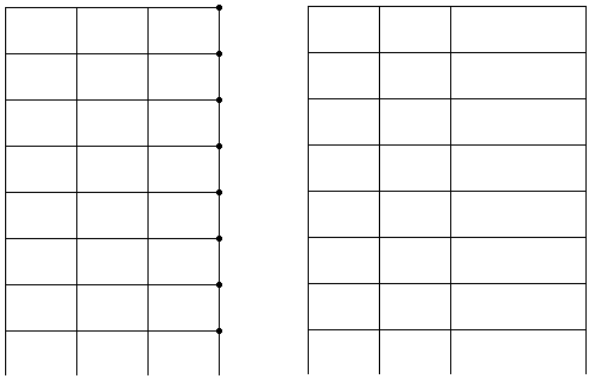


Figure 6.2-6. *Example of using the operation of moving the nodes in the given vector*

Another type of transfer operation is similar to the previous one, but the vector defines only the direction, and the distance to which the nodes are moved is specified by the user.

If you use the operation of moving nodes, you have to make sure that there are no unacceptable changes of the shape of finite elements in the result.

Merge Nodes with the Same Coordinates

In the result of this operation only one node from each group of nodes with the same coordinates will be left in the model, and the others will be deleted. In the elements adjacent to the deleted nodes the numbers of the deleted nodes will be replaced with the numbers of the nodes remaining in the model.

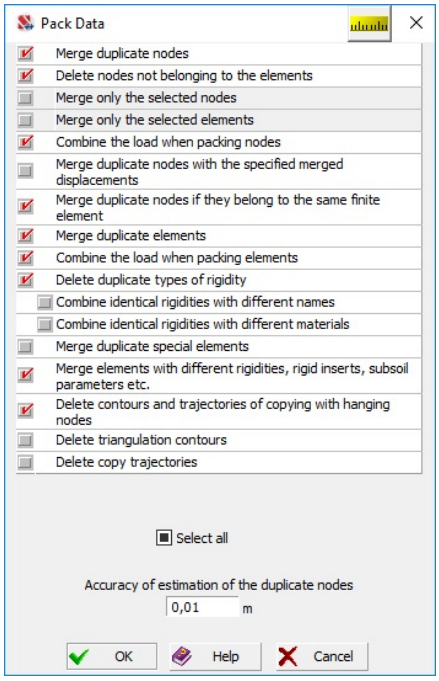


Figure 6.2-7. *The* **Pack Data** *dialog box*

The rules for merging are specified in the **Pack Data** dialog box, which appears after invoking the operation and includes the following options:

- *Merge only the selected nodes* — only the nodes selected before invoking the operation are merged;
- *Combine the load when packing nodes* — the loads from all the duplicate nodes are added and applied to the node which remains in the model;
- *Merge duplicate nodes with the specified merged displacements* — nodes are merged and the index of merged displacements is deleted if the model does not have any other nodes included into the same group of merged displacements;
- *Merge duplicate nodes if they belong to the same finite element* — after the node is deleted the element does not change its type automatically.

The operation is performed after you close the dialog box by clicking the **OK** button and does not require the confirmation by clicking the similar button in the toolbar.

Nodes are considered as duplicate if the distance between them is less than the value of the accuracy of estimation of the duplicate nodes specified in the **Pack Data** dialog box.

Generate Nodes along an Arc

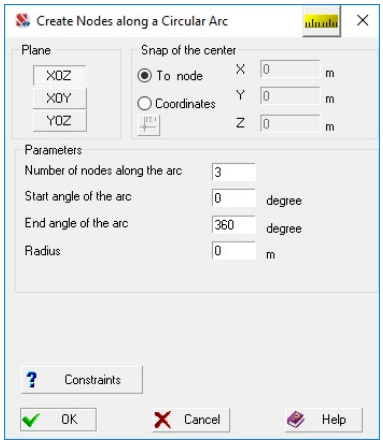


Figure 6.2-8. *The* **Create Nodes along a Circular Arc** *dialog box*

This operation enables to create nodes along a circular arc lying in the given plane. Control of the operation and specification of the parameters of the arc is performed in the **Create Nodes along a Circular Arc** dialog box (Fig. 6.2-8), which is invoked by clicking the respective button.

This dialog box is used to specify the plane the new nodes lie in, the number of nodes along the arc, start and end angles of the arc, radius of the circular arc and its center. The center can be snapped either to a node or to a point with the specified coordinates.

The button enables to invoke the dialog box (see Fig. 6.2-3), where you can specify the number of the node. After clicking the **OK** button the coordinates of this node will appear in the **X**, **Y**, **Z** text fields (they can be changed if necessary) and will be used as the coordinates of the center.

In the case when the center is specified by the coordinates, the new nodes will be added to the model immediately after clicking the **OK** button in the dialog box. If the center is specified as lying in the node, you have to select this node, click the **OK** button in the toolbar, and the new nodes will appear in the model.

Constraints can be specified in the nodes simultaneously with the input. The respective button is used for this purpose, clicking which invokes the **Constraints** dialog box (see Sec. 7.4).

Move the Origin

Once this operation is invoked, the dialog box (Fig. 6.2-9) appears on the screen, where you have to use the respective radio buttons to select the rule for specifying the point the origin is moved into. If you have selected the transfer into the given node, then after closing the dialog box you have to select this node in the model and click the **OK** button in the toolbar.

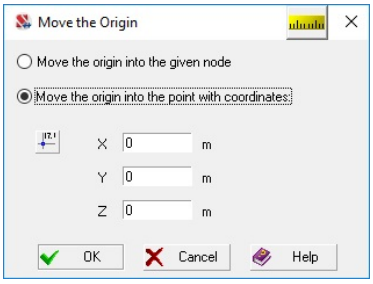


Figure 6.2-9. *The* **Move the Origin** *dialog box*

If the origin is moved into the point with the given coordinates, then these coordinates are specified in the respective text fields of the dialog box, and the origin will be moved after closing the dialog box by clicking the **OK** button. The button enables to invoke the dialog box (see Fig. 6.2-3), where you can specify the number of the node. After closing this dialog box the coordinates of this node will appear in the **X**, **Y**, **Z** text fields, and will be used as the new origin.

This operation is often used when performing geometric transformations, for example when rotating a part of the model by the given angle. Since it is performed by rotating a group of nodes (and therefore – the elements attached to them as well) about the *origin of the global coordinate system*, the result will depend on the position of the center of rotation.

Create Nodes at the Specified Distance from the Selected Ones

The operation enables to create new nodes, each of which will be located at the given distance from the nodes-prototypes selected in the model. Once the operation is invoked, the **Create Nodes at the Specified Distance from the Selected Ones** dialog box appears. It contains the table where you have to

specify the distances in the form of increments to the coordinates of the selected nodes and the number of repetitions (Fig. 6.2-10). If a few lines are filled in the table, the values specified in them are treated as increments to the coordinates of the nodes input according to the values of the previous line.

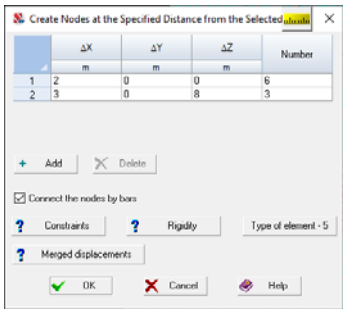


Figure 6.2-10. *The* **Create Nodes at the Specified Distance from the Selected Ones** *dialog box*

In order to perform the operation you have to specify all the necessary information in the dialog box, close the dialog box, select the nodes-prototypes in the model and click the **OK** button in the toolbar.

If nodes have to be connected by bars, you should check the respective checkbox. This will make the **Rigidity** and **Type of element** buttons accessible. Clicking them invokes the respective dialog boxes where you can specify the stiffness properties and the type of element (see Chapter 7).

Constraints can be specified in the nodes simultaneously with the input. The respective button is used for this purpose, clicking which invokes the **Constraints** dialog box (see Sec. 7.4). Similarly, the **Merged displacements** button enables to create groups of merged displacements (the required directions are selected in the respective dialog box). In this case, every selected node will generate "its own" group of merged displacements.

New lines can be added to the table by clicking the **Add** button. The selected lines are deleted by clicking the **Delete** button.

A completely zero line is considered as the input of increments equal to zero which will lead to duplicate nodes.

Replace a Node in Elements

This operation enables to replace the node all the elements are adjacent to by another one.

The node you want to replace should be selected first (it is highlighted in red), and then the node it will be replaced by (it is highlighted in green), and click the **OK** button in the toolbar. Once the operation is performed, the node highlighted in red is not deleted.

If both selected nodes belong to the same finite element, a replacement will be executed only after the user's confirmation, since it will lead to the appearance of a degenerate element. You have to make sure that there are no unacceptable changes of the shape of finite elements in the result.

Select Nodes

You can use this option in cases when it is necessary (or convenient) to select nodes before invoking the operation you want to perform with these nodes, for example, when copying or performing the geometric transformations. After invoking this option you can use any cursor to select nodes. The selection can be canceled by clicking the **Rejection** button in the **Nodes** section of the toolbar or the **Reset all marks** button in the **Display Filters** toolbar.

The **Control the Selection** toolbar also contains this button (see Sec. 6.1).

Create Nodes at the Intersection of Grid Lines

This operation enables to create nodes at the intersection of grid lines in the given area of the grid. The elevations are considered as the coordinates of nodes along the Z axis. The created nodes can be used for all operations of input available in the software.

The control of the creation of nodes is performed in the **Create Nodes at the Intersection of Grid Lines** dialog box (Fig. 6.2-11). You can create nodes in the area bounded by the specified marks of the grid lines of the beginning – left list, and the end – the right list for each grid and each direction of grid lines of a particular grid. Before specifying the area of the grid where the nodes are created, it is necessary to uncheck **The whole interval** checkbox in the line corresponding to this grid (Fig. 6.2-11). Uncheck the **Involved in the intersection** checkbox if you want to completely eliminate the creation of nodes at the intersection of grid lines.

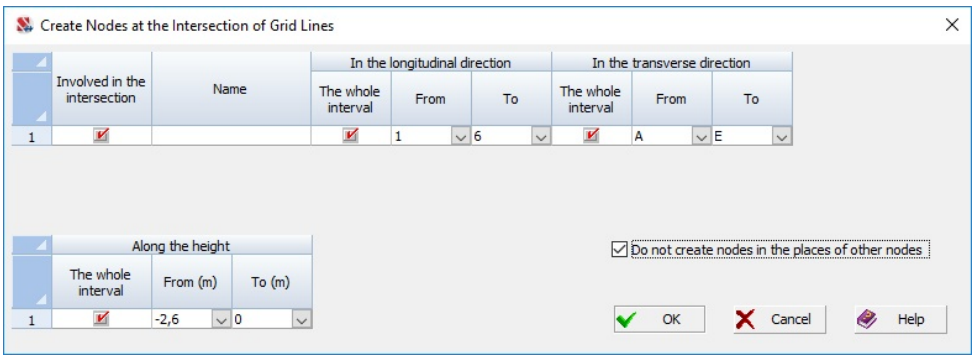


Figure 6.2-11. *The* **Create Nodes at the Intersection of Grid Lines** *dialog box*

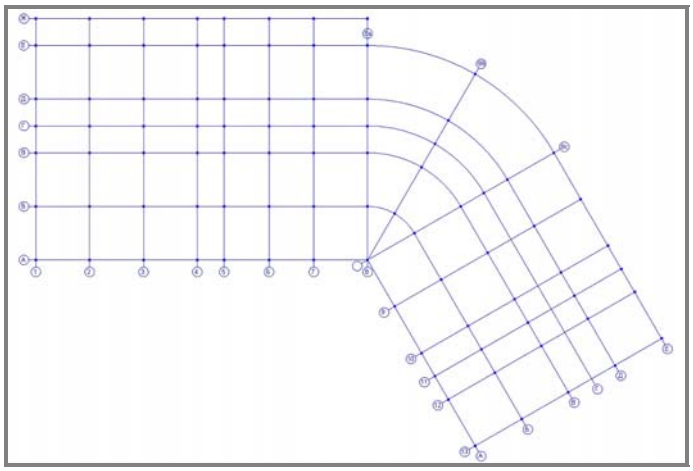


Figure 6.2-12. *Grid lines with nodes at the intersections of grid lines*

In order to prevent the appearance of duplicate nodes at the intersection of grid lines check the **Do not create nodes in the places of other nodes** checkbox.

Generate a Generator by the Formula

This operation enables to generate nodes along a generator defined by the function *a = f(b)* and lying in one of the planes of the global coordinate system. Once the operation is invoked the **Generate a Generator by the Formula** dialog box (Fig. 6.2-13) appears, where you have to enter the formula of the

generator, select the plane the nodes will lie in, specify the initial (**Beg.**) and final (**End**) values of the argument of the function, and also the number of nodes (**Numb.**) on the generator.

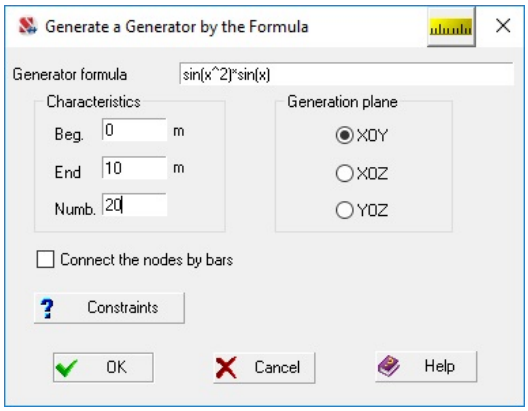


Figure 6.2-13. *The* **Generate a Generator by the Formula** *dialog box*

The following functions will be performed depending on the selected plane of generation:

- XOY plane — $y = f(x)$;
- XOZ plane — $z = f(x)$;
- YOZ plane — $z = f(y)$.

If nodes have to be connected by bars, you should check the respective checkbox. This will make the **Rigidity** and **Type of element** buttons accessible. Clicking them invokes the respective dialog boxes where you can specify the stiffness properties and the type of element (see Chapter 7).

Constraints can be specified in the nodes simultaneously with the input. The respective button is used for this purpose, clicking which invokes the **Constraints** dialog box (see Sec. 7.4).

Move Nodes to the Given Plane

Clicking the **Move nodes to the given plane** button drops down the menu (Fig. 6.2-14), where you have to select the necessary operation.

If there are no selected nodes in the model, only one menu item is available. It enables to move the selected nodes to the plane parallel to one of the planes of the global coordinate system. The setting of the operation is performed in the respective dialog box (Fig. 6.2-15).

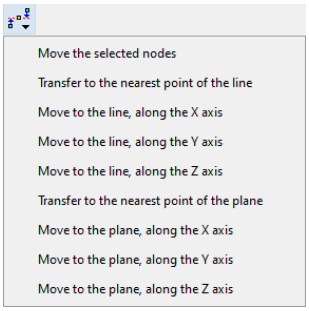


Figure 6.2-14. *The* ***Move nodes to the given plane*** *menu*

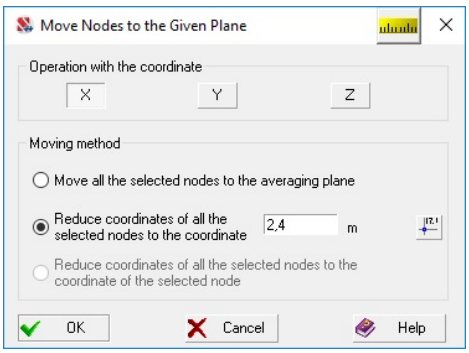


Figure 6.2-15. *The* **Move Nodes to the Given Plane** *dialog box*

If the **X** coordinate is selected in the **Operation with the coordinate** group, the nodes are moved to a plane parallel to **YOZ**, if **Y** is selected, they are moved to a plane parallel to **XOZ**, and if **Z** is selected — to a plane parallel to **XOY**.

Depending on the option selected in the **Moving method** group the transfer can be performed in three ways. In the first case (the **Move all the selected nodes to the averaging plane** radio button is selected) the selected nodes are moved in the given direction to the plane the point with the average coordinates belongs to. In the second case (the **Reduce coordinates of all the selected nodes to the coordinate** radio button is selected) the transfer is performed by reducing the given coordinate to the specified value in all

the selected nodes. The third method of moving (the **Reduce coordinates of all the selected nodes to the coordinate of the selected node** radio button is selected) enables to transform the coordinates of the selected nodes in such a way that all of them will have the value of the coordinate specified in the **Operation with the coordinate** group equal to the coordinate of the selected node. In order to perform the operation you have to:

- select the nodes in the model the coordinates of which will change;
- invoke the operation by selecting the **Move the selected nodes** option in the drop-down menu (Fig. 6.2-14);
- specify the coordinate in the **Operation with the coordinate** group of the **Move Nodes to the Given Plane** dialog box (Fig. 6.2-15);
- select the **Reduce coordinates of all the selected nodes to the coordinate of the selected node** radio button;
- close the dialog box by clicking the **OK** button, and select the node with the necessary value of the coordinate in the model;
- click the **OK** button in the toolbar.

Clicking the button invokes the **Select Node** dialog box (see Fig. 6.2-3), which enables to specify the node one of the coordinates of which is used as a coordinate for snapping the selected nodes. This operation is usually necessary in those cases when the model is obtained in the result of importing the data from the graphic systems. It often happens so that the nodes which have to lie in one plane are in fact in its vicinity. Such an "inaccuracy" may result for example in non-planar four-node elements.

If you first select nodes in the model, transformations — **Move to the line** and **Move to the plane**, will be available in the drop-down menu which appears after invoking the **Move nodes to the given plane** operation. In both cases it is possible to move the nodes to the nearest point (along the normal) or to move them along the selected direction of the global coordinate system. If you select **Move to the line**, you have to select two nodes in the model which define a line in space and click the **OK** button in the toolbar. In the result all the selected nodes will be moved to this line. When the **Move to the plane** operation is selected, you have to select three nodes in the model which do not lie on the same line and define a plane in space. After clicking the **OK** button all the selected nodes will be moved to this plane.

Round the Node Coordinates

This operation enables to round the coordinates of nodes of the design model with the specified accuracy. When the operation is invoked, the **Round the Node Coordinates** dialog box (Fig. 6.2-16) appears, where you have to specify the accuracy of rounding. Once the dialog box is closed by clicking the **OK** button, the coordinates of nodes will be rounded with the specified accuracy. If you want to round the coordinates only of a certain set of nodes, these nodes should be selected and the respective checkbox should be checked in the dialog box.

You might need to round the coordinates of nodes after loading a model from graphics editors or design systems.

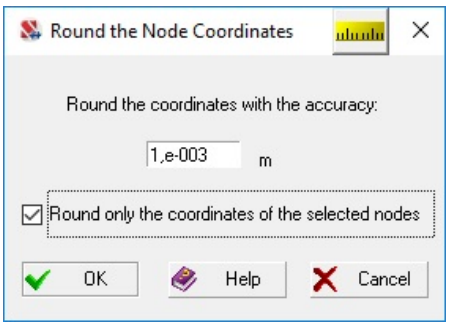


Figure 6.2-16. *The* **Round the Node Coordinates** *dialog box*

If the model includes planar elements which do not lie in the planes of the principal coordinate system, using this operation may distort their shape (for example, some nodes may be moved from the original plane).

Renumbering Nodes

The operation enables to renumber the nodes of the design model in accordance with the rules selected by the user. The numbering can be performed in such a way so that the preselected nodes get the minimum numbers, and the deleted nodes are placed in the end of the list. For all other nodes or only the selected ones (if the **Renumber only the selected nodes** checkbox is checked) the rules for renumbering are set in the table (see figure 6.2-17).

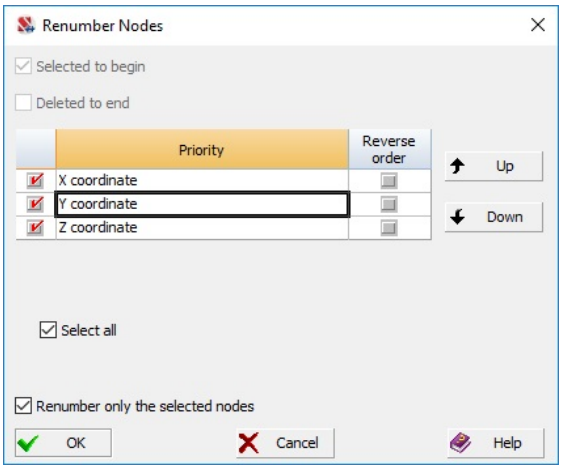


Figure 6.2-17. *The* **Renumber Nodes** *dialog box*

Select the components of the coordinates which will be taken into account when sorting by checking the respective checkboxes, and set the priorities with the help of the **Up** and **Down** buttons. For example, if the table has the form shown in the figure, all the selected nodes will have the first numbers, the deleted nodes will have the last numbers, for all other nodes the minimum numbers will be assigned to the nodes in the ascending order of the Z coordinate; if the nodes have the same Z coordinates, the renumbering will be performed in the ascending order of the X coordinate and so on. The **Reverse order** checkbox can be used for the coordinates, which enables to perform the renumbering in the descending order of values instead of the ascending one.

Copy Node Properties

The operation is used in those cases when properties similar to those of the sample node have to be assigned to several nodes of the design model. The set of copied properties includes:

- constraints;
- merged displacements;
- belonging to the groups;
- loads.

The **Copy Node Properties** dialog box (Fig. 6.2-18) appears after invoking this operation. It contains two groups of controls. The **Sample node** group provides the selection and visualization of a node the properties of which are copied, and the **Inheriting nodes** group provides the selection and visualization of nodes which obtain the properties of the sample node.

The copying enables to assign certain properties of the sample to the “inheriting nodes”, which can be selected in the **Parameters for Copying** dialog box (Fig. 6.2-19). The dialog box can be invoked by clicking the respective button after selecting the “sample” and “inheriting nodes”.

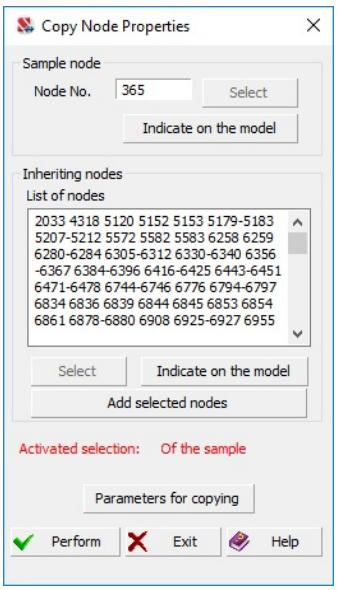


Figure 6.2-18. *The* **Copy Node Properties** *dialog box*

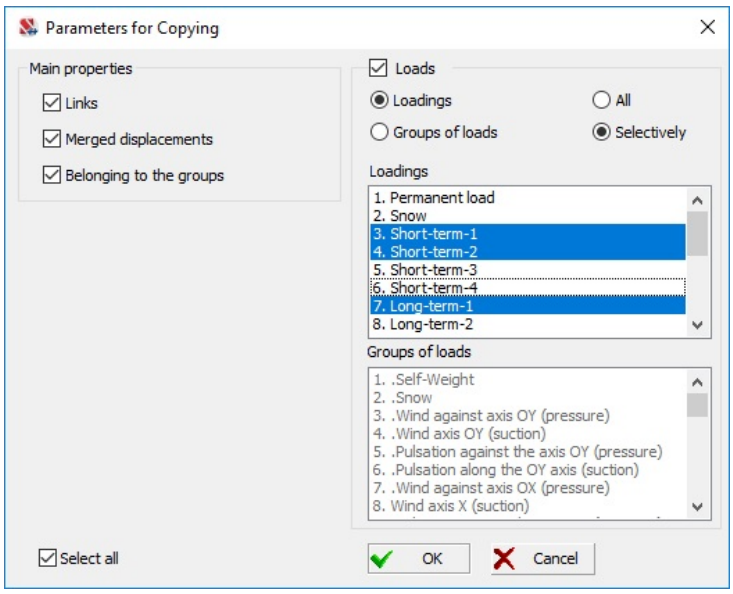


Figure 6.2-19. *The* **Parameters for Copying** *dialog box*

A set of properties is assigned with the help of checkboxes in the **Main properties** group in this dialog box. Moreover, the rules for copying loads (loadings or groups, all or selectively) can be specified in the **Loads** group. When the loads are copied selectively, the names of the copied loadings or groups are selected from the respective lists. The following procedure is recommended for copying:

- specify the number of the sample node in the *Node No.* field in the **Copy Node Properties** dialog box and click the **Select** button. The selected node will be highlighted in green in the model. If it is more convenient to perform this action with the help of the cursor, click the **Indication on the model** button and select the necessary node with the cursor;
- specify the numbers of the inheriting nodes in the *List of nodes* field and click the **Select** button to select these nodes in the model. If it is more convenient to perform the selection with the help of the cursor, click the **Indication on the model** button before performing this action;
- invoke the dialog box for assigning the properties to the inheriting nodes by clicking the **Parameters for copying** button and specify the set of copied properties in it;
- click the **Perform** button in the **Copy Node Properties** dialog box after selecting all inheriting nodes.

6.3. Operations with Elements

Operations with elements are invoked from the **Elements** section (Fig. 6.3-1) of the **Nodes and Elements** toolbar.

Figure 6.3-1. *The* **Elements** *section of the* **Nodes and Elements** *toolbar*

The **Elements** section of the **Nodes and Elements** toolbar includes a *group of buttons* which enable to perform the following operations:

— add plate;

— add bars;

— add solid elements;

— add bars taking into account the intermediate nodes;

— create elements along an arc;

— delete elements;

— restore deleted elements;

— bar division;

— divide bars at the intersection point;

— division of 4-node plates;

— division of 4-node plates into 3-node ones;

— division of 3-node plates;

— division of solid elements;

— division of plates taking into account the intermediate nodes;

— division of bars taking into account the intermediate nodes;

— combine bar elements;

— separate elements;

— move the element;

— attach additional nodes to the elements;

— detach all additional nodes from the faces of the elements;

— merge duplicate elements;

— pack data;

— select elements;

— divide plates along the intersection line;

— combine 3-node elements into 4-node ones;

— extend the bar to the plate;

— chamfer bars;

— create a bar perpendicular to the selected one;

— divide bars by a polygon;

— renumbering elements;

— copy element properties;

— improve the meshing quality;

— meshing quality;

— export polygon from the selected plates.

Create Plates

This operation is used to add three- and four-node plate elements to the model (snap them to the already created nodes). The software enables to create elements with pre-specified characteristics and with "default" characteristics.

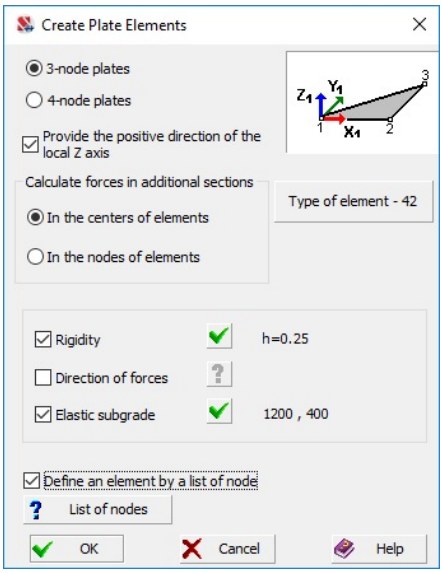


Figure 6.3-2. *The* **Create Plate Elements** *dialog box*

The characteristics can be specified in the **Create Plate Elements** dialog box (Fig. 6.3-2), which is invoked by clicking the **Add plate** button. You can specify the following characteristics in this dialog box:

- number of nodes of the created elements (three or four);
- rule for calculating the forces (in the centers of elements or in the centers and nodes of elements);
- stiffness properties (the **Rigidity** checkbox and the respective button);
- direction of the force output (the **Direction of forces** checkbox and the respective button);
- type of element (the **Type of element** button);
- characteristics of the elastic subgrade (the **Elastic subgrade** checkbox and the respective button).

The nodes are selected and the elements are created after closing the dialog box by clicking the **OK** button.

The rules for specifying the above characteristics are given in the respective sections (see Chapter 7). The elements will have these characteristics until you invoke the **Add plate** operation again. Some characteristics may be not specified when creating the elements, in this case they will be taken “by default”.

If the **Provide the positive direction of the local Z axis** checkbox is checked, regardless of the sequence of selection of nodes the new element will be added in such a way that the local Z_1 axis will have a positive direction (if the nodes were selected in a sequence which ensures the fulfillment of the given condition, it will be taken into account when creating the element).

If you want to create elements with the “default” characteristics, start the work by clicking on the arrow on the right from the **Add plate** button. You can select one of the following variants in the drop-down menu that appears after clicking the arrow:

- Create 3-node plates with the positive direction of the local Z_1 axis;
- Create 4-node plates with the positive direction of the local Z_1 axis;
- Create plates with the positive direction of the local Z_1 axis;
- Create 3-node plates taking into account the sequence of the selection of nodes;
- Create 4-node plates taking into account the sequence of the selection of nodes;

- Create plates taking into account the sequence of the selection of nodes.

The first three variants provide the specified orientation of the Z_1 axis when creating the elements and do not guarantee allowing for the sequence of the selection of nodes by the user (a rectangular marquee or a polygonal marquee can be used to select a group of nodes). The multiple selection makes no sense for the last three variants, and only a pointer can be used; if the nodes have been already selected, their selection will be cancelled.

After selecting the nodes the element will be created if you double left click or click the **OK** button in the toolbar, or press the **Enter** key in the keyboard. The elements have the following characteristics by default:

- type of element — is defined by the type of the design model and the number of nodes;
- stiffness properties — not specified;
- direction of the force output — coincides with the direction of the local X_1 axis;
- characteristics of the elastic subgrade — not specified;
- calculation of forces — only in the center of the element.

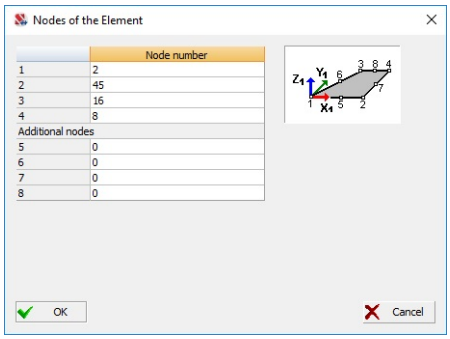


Figure 6.3-3. *The* **Nodes of the Element** *dialog box*

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2). If an element with additional nodes on the sides is created, then you have to enter 0 in the part of the table which lists the additional nodes to indicate that there is no additional node.

Create Bars

This operation is performed according to the same rules as the previous one and enables to add bar elements to the model (snap them to the already created nodes). The software enables to create elements with pre-specified characteristics and with "default" characteristics.

In order to specify the characteristics before creating the elements you have to click the **Add bars** button by pointing with the cursor at the icon with the image of the bar, the **Create Bar Elements** dialog box (Fig. 6.3-4) will appear where you have to specify:

- the principle of creating bars (one by one or in a series);
- number of sections the forces are calculated in;
- stiffness properties (the **Rigidity** checkbox and the respective button);
- hinges (the **Hinges** checkbox and the respective button);
- type of element (the **Type of element** button);
- rigid inserts (the **Rigid inserts** checkbox and the respective button).

The elements are created after closing the dialog box by clicking the **OK** button.

The rules for specifying the above characteristics are given in the respective sections of this book (see Chapter 7). The elements will have these characteristics until you invoke the **Add bars** operation again. Some characteristics may be not specified when creating the elements, in this case they will be taken “by default”.

If you want to create elements with the “default” characteristics, start the work by clicking on the arrow on the right from the **Add bars** button. You have to select the principle of creating bars — one by one or in a series, in the drop-down menu and proceed to creating elements. The elements have the following characteristics by default:

- type of element — is defined by the type of the design model;

- number of sections the forces are calculated in — not specified (a value defined in the calculation parameters will be used in the analysis);
- hinges — no;
- stiffness properties — not specified;
- characteristics of the elastic subgrade — not specified;
- rigid inserts — not specified.

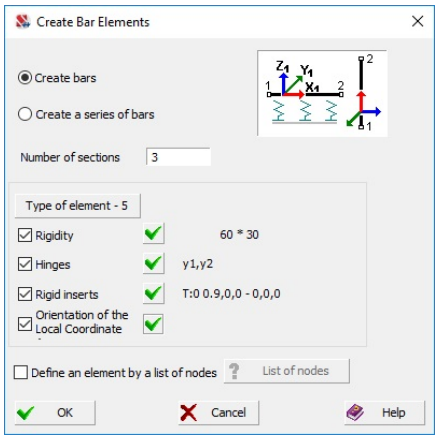


Figure 6.3-4. *The* **Create Bar Elements** *dialog box*

In order to create one bar place the cursor with a target over the node and left-click, stretch the rubber band to the second node and click the mouse button again. The node selected first will be specified as the first node of the bar, i.e. the origin of the local coordinate system of the element.

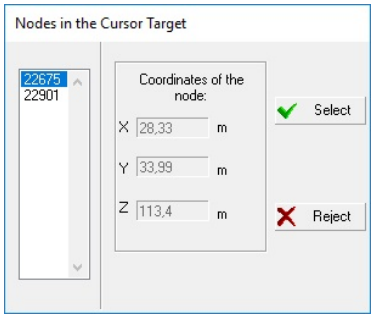


Figure 6.3-5. *The* **Nodes in the Cursor Target** *dialog box*

The process of creating a series of bars will continue until you click the button in the toolbar (or press the **Enter** key in the keyboard). After this the operation of creating bars is not cancelled and can be continued starting with any other node.

The operation of creating a series of bars can be interrupted by clicking the **Rejection** button— in the toolbar or pressing the **Esc** key in the keyboard.

If there is more than one node in the cursor target, the **Nodes in the Cursor Target** dialog box (Fig. 6.3-5) appears. You have to select the necessary node in the list in this dialog box and click the **Select** button.

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

Create Solid Elements

Similarly to plates and bars you can create solid elements with pre-specified or default parameters. In any case, first you have to specify the type of the element that will be created.

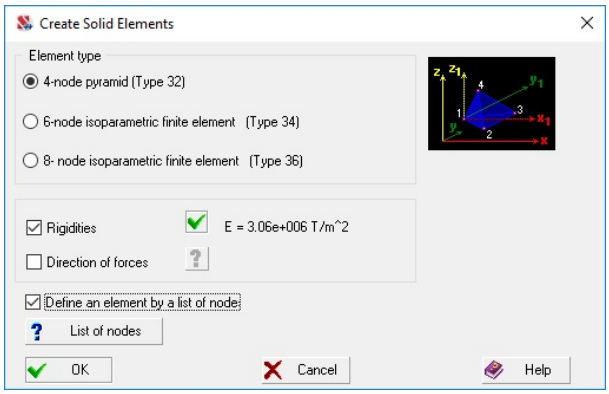


Figure 6.3-6. *The* **Create Solid Elements** *dialog box*

In order to specify the characteristics before creating the elements you have to click the **Add solid elements** button by pointing with the cursor at the icon with the image of the elements, the **Create Solid Elements** dialog box (Fig. 6.3-6) will appear where you have to specify:

- type of element (you can create elements of three types — four-node pyramids (type 32), and six- (type 34) and eight-node (type 36) elements);
- stiffness properties (the **Rigidity** checkbox and the respective button);
- direction of the force output (the **Direction of forces** checkbox and the respective button).

The elements are created after closing the dialog box by clicking the **OK** button.

The rules for specifying the above characteristics are given in the respective sections of this book (see Chapter 7). The elements will have these characteristics until you invoke the **Add solid elements** operation again. Some characteristics may be not specified when creating the elements, in this case they will be taken "by default".

If you want to create elements with the "default" characteristics, start the work by clicking on the arrow on the right from the **Add solid elements** button. Select the type of element in the drop-down menu and proceed to creating elements. The elements have the following characteristics by default:

- stiffness properties — not specified;
- direction of the force output — along the axes of the global coordinate system.

In order to create an element, you just have to select the nodes in the model the number of which must correspond to the specified type of the element, and the element will be created automatically. The nodes can be selected only by the cursor with a target.

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

Delete Elements

In order to delete elements you have to select them in the model (they will be highlighted in red) and click the **OK** button in the toolbar or press the **Enter** key in the keyboard.

If the **Do not delete "hanging" nodes after deleting the element** checkbox is not checked in the **Environment Settings** dialog box (hanging nodes are the nodes not adjacent to other elements of the model except for the deleted ones), then such nodes are deleted together with the elements and can be restored with the help of the **Restore deleted nodes** operation.

Elements can be selected by any type of cursor. When the cursor with a target is used — you have to place the target in the area of the element and left-click. If there are several elements in the cursor target,

their list will be given in a special dialog box **Elements** in the cursor target. Once the necessary element is selected in the list, click the **Select** button. When a marquee is used only those elements are selected all nodes of which are within the rectangle or polygon. Clicking on the selected element for the second time deselects it.

Restore Deleted Elements

This operation enables to restore the deleted elements if the packing of data was not performed after they had been deleted. It is necessary to perform the following actions to restore the deleted elements:

- select the elements you want to restore (they will be displayed in red);
- click the **OK** button in the toolbar.

When restoring the elements, deleted nodes these elements are adjacent to will be restored as well.

The deleted elements can be highlighted in the model with the help of the **Deleted elements** filter .

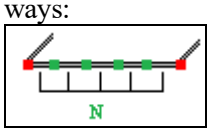
Create Bars Taking into Account the Intermediate Nodes

This operation enables not only to create a new bar, but also to automatically divide it into several elements taking into account all the intermediate nodes lying along the line of the bar. The operation can be performed both for 2D and 3D design models according to the same rules as in the case of creating one bar element or a series of bars.

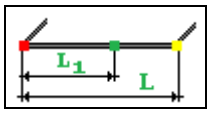
The accuracy of determining whether a bar passes through a node is defined in the environment settings by a variable which determines the accuracy of estimation of the duplicate nodes.

Divide the Bar

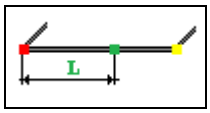
This operation is used to divide a bar into several bars and enables to perform the division in three different ways:



— N equal segments (bars);



— into two bars in the given ratio;



— into two bars of the given size (in this case you have to specify the length of the new bar adjacent to the first node of the original one).

The reference node of the bar (the first or the second one) can be specified for the last two methods of division.

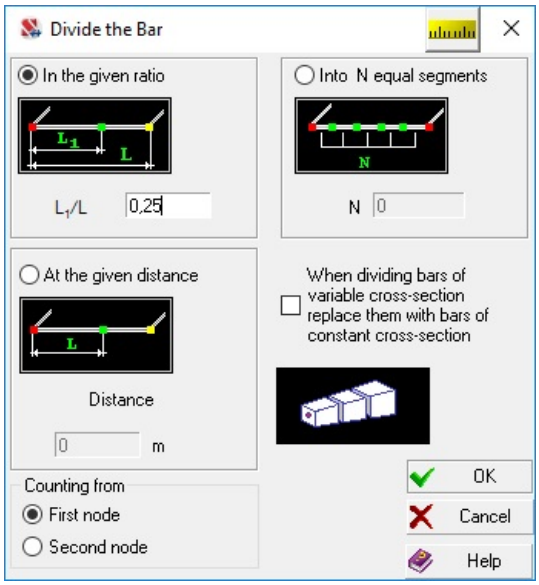


Figure 6.3-7. *The* **Divide the Bar** *dialog box*

In order to select the method of division enable the respective option in the dialog box (Fig. 6.3-6) and specify the numerical characteristics. After closing the dialog box select the elements in the model and click the **OK** button in the toolbar.

When specifying the data you should remember that when dividing a bar in the given ratio the value L_1/L must be positive and less than one, and when dividing into two bars of the given size — the length of the new bar must not be greater than the length of the original bar.

When performing the operation all the loads specified for the original bar will be transferred to the new elements.

The boundary conditions of the original bar (hinges and sliders), and rigid inserts lying along the bar will be transferred to the ends of the new series of bars. Rigid inserts out of the plane of the bar will be moved correctly to the new elements only in the case if they have the same length and lie in one plane.

When dividing bars of variable cross-section you can either create new bars of variable cross-section or obtain some new bars with a constant cross-section.

Divide Bars at the Intersection Point

This operation enables to divide two intersecting bars in such a way that the intersection point becomes a node the new bars obtained in the result of the intersection are adjacent to. In order to perform it you just have to select the necessary bars and click the **OK** button in the toolbar.

The boundary conditions of the original bar (hinges and sliders), and rigid inserts lying along the bar will be transferred to the ends of the new bars. Rigid inserts out of the plane of the bar will be moved correctly to the new elements only in the case if they have the same length and lie in one plane.

Combine Bar Elements

This operation enables to combine bar elements that have a common node, or a series of bar elements that lie on one straight line into one element. Clicking the arrow drops down the menu where you can select one of the following actions:

- Combine two bar elements;
- Combine bars that lie on one line;
- Replace bars that lie on one line with a variable cross-section bar.

The operation of combining two bars is selected by default.

After invoking the operation, select the bars you want to combine and click the **OK** button in the toolbar.

Regardless of the selected action rigid inserts and hinges (if any) remain in the end nodes of the chain of combined bar elements. Rigid inserts and hinges in other nodes are ignored.

When combining bars that lie on one straight line, rigid inserts are taken into account and the continuity of the flexible part is checked when the elements are joined. When two bars are combined, their flexible parts may not lie on one straight line.

The element type, orientation of the local axes, rigidity (except for the case of creating a variable cross-section bar), elastic subgrade and the value of prestressing remain the same as specified for the element adjacent to the first node of the combined element.

When creating a variable cross-section bar, the bars at the beginning and end of the chain have the same type of parametric description. If this is not the case, it is assigned the type of rigidity of the element adjacent to the first node. The rigidities of intermediate bars are ignored.

Uniformly distributed, concentrated and trapezoidal loads remain without changes. If the flexible parts of the elements do not lie on one straight line, then their full compliance is not guaranteed.

The self-weight specified for some elements in the chain is applied to the combined element.

Once the bars are combined you have to check the type of the obtained element and the orientation of the local axes. The thermal load specified for individual elements in the chain is transferred from one of the elements and is checked once the operation is performed.

Create Bars along an Arc

This operation is used to create bar elements adjacent to the nodes lying along a circular arc. The control of the input is performed in the **Create Elements along a Circular Arc** dialog box (Fig. 6.3-8). Initial data necessary to perform the operation include:

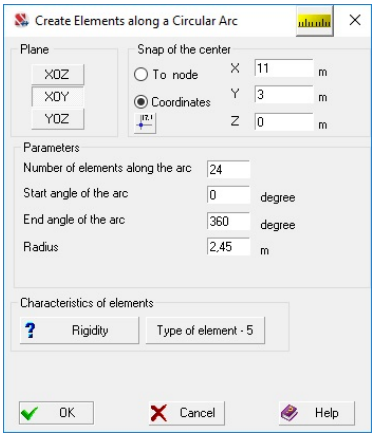


Figure 6.3-8. *The* **Create Elements along a Circular Arc** *dialog box*

- selection of a plane the new elements lie in (only the planes parallel to the planes of the principal coordinate system are allowed);
- number of elements along the circular arc;
- values of the start and end angles of the arc (for a full circle, for example, they are 0^0 and 360^0 or 0 and 6.28 radian respectively);
- radius of the arc;
- snap of the center of the circle (to the node or the values of the coordinates).

If the center is specified by the coordinates, the new elements will be added to the model automatically after closing the dialog box by clicking the **OK** button. If the center lies in the node, then after closing the dialog box you have to select this node and click the **OK** button in the toolbar or press the **Enter** key in the keyboard.

When specifying the parameters of the arc you can also specify the rigidities and the type of elements (the **Rigidity** and **Type of element** buttons respectively).

In order to specify a node with a certain number as the center of the circle use the **Obtain the coordinates of the node with the specified number** button , the **Select Node** dialog box will appear (see Fig. 6.3-3), where you can specify its number. The coordinates of this node will appear in the **X**, **Y**, **Z** text fields and can be changed.

Select Elements

This operation is similar to the respective operation in the **Control the Selection** toolbar. The selection of elements will be performed taking into account the states of filters of this toolbar. The operation and the selection of all elements can be cancelled by clicking the **Rejection** button in the toolbar or the **Cancel the selection** button in the **Display Filters** toolbar.

Merge Duplicate Elements

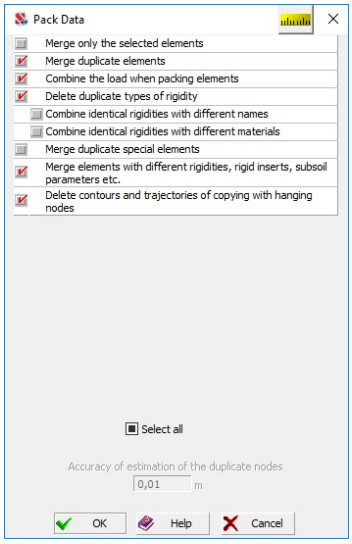


Figure 6.3-9. *The* **Pack Date Merge Duplicate Elements** *dialog box*

The operation enables to exclude the duplicate elements from the model, i.e. such elements of the same type, all nodes of which have the same numbers. The rules for combining are specified in the dialog box (see Fig. 6.3-9).

This dialog box can be used to specify or reject combining special finite elements, deleting duplicate types of rigidity, ...

If the model has no selected duplicate elements, after invoking the operation the combination will be performed for the entire model, if there are selected duplicate elements — the operation will be performed only with the selected elements, provided that the user confirms this by checking the respective checkbox.

To indicate the duplicate elements in the model, use the filters button .

Divide 4-node Plates

This operation enables to divide the selected four-node elements into several elements.

The initial data necessary to perform the operation are specified in the **Divide Plates** dialog box (Fig. 6.3-9) and include the number of divisions of the faces of the element lying along the directions of the local X_1 and Y_1 axes. Duplicate nodes may appear in the model after performing the operation.

If the **Pack duplicate nodes for the whole model** checkbox is checked in the dialog box, then after performing the division this operation will be invoked automatically. Otherwise, the user has to detect the duplicate nodes and combine them (see Sec. 6.2).

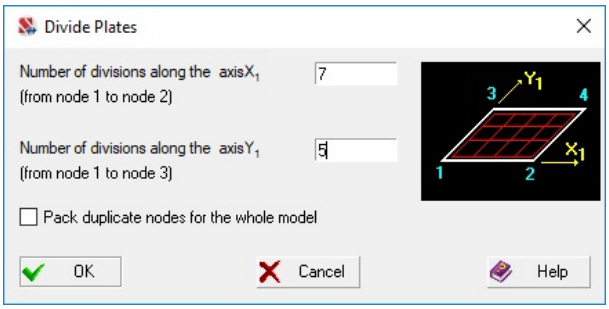


Figure 6.3-10. *The* **Divide Plates** *dialog box*

When this operation is performed, elements with additional nodes on the sides are ignored.

Divide 4-node Plates into 3-node Ones

The operation is used to divide four-node elements into three-node ones. The type of division is specified in the dialog box (Fig. 6.3-11) which appears after invoking the operation. The procedure is standard:

- invoke the operation;
- the dialog box will appear where you have to specify the type of division;
- select the necessary elements in the model;
- click the **OK** button in the toolbar.

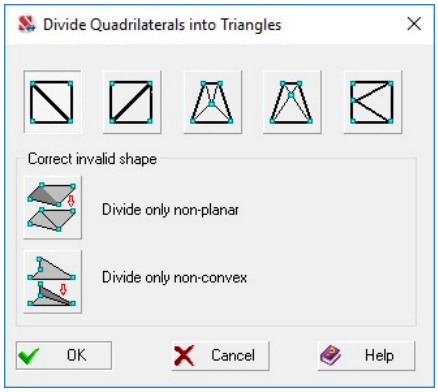


Figure 6.3-11. *The* **Divide Quadrilaterals into Triangles** *dialog box*

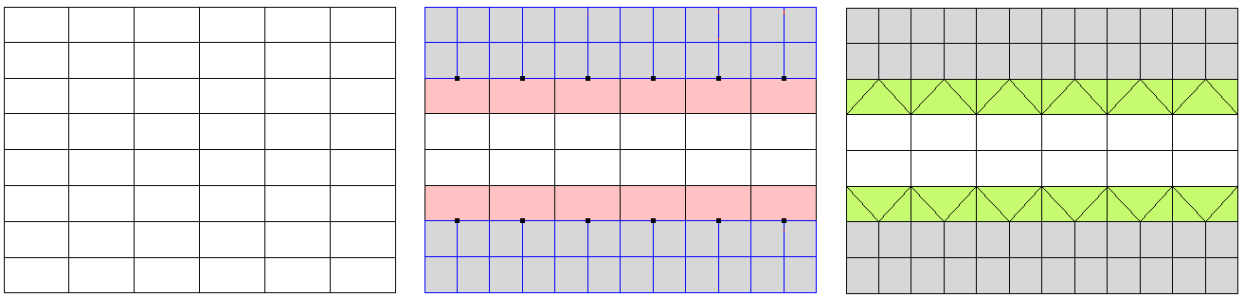


Figure 6.3-12. *Example of connecting finite element meshes with different division steps*

This operation enables to connect the original mesh of 4-node elements with a mesh obtained by dividing some elements with a factor of 2. In this case a “free” node appears on the edge of the original elements lying on the border of the mesh refinement area. If you select such elements and apply the following type of division to them, each element will be divided into three triangles and meshes will be connected, as shown in Fig. 6.3-12.

Another example, when you have to divide four-node elements into three-node ones is the necessity to "correct bad" quadrangular elements. In this case, you can use the following options:

- divide only non-planar;
- divide only non-convex.

If one of the above markers is selected, the operation will be performed only for the selected elements with the respective properties.

When this operation is performed, elements with additional nodes on the sides are ignored.

Divide 3-node Plates

The operation is used to divide three-node elements. When it is performed the sides of the selected triangles are divided into equal segments the number of which is specified in the **Divide 3-node Plates** dialog box (Fig. 6.3-13).

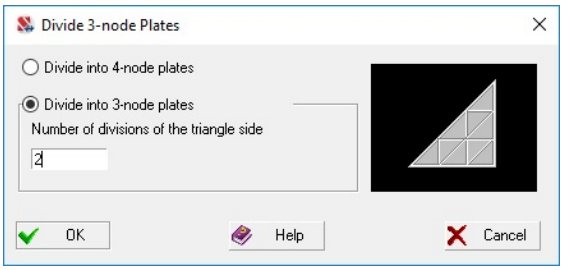


Figure 6.3-13. *The* **Divide 3-node Plates** *dialog box*

Moreover, after selecting the respective option in this dialog box a three-node element can be divided into four-node ones (perpendiculars are dropped from the center of mass of the triangle to its sides).

When this operation is performed, elements with additional nodes on the sides are ignored.

Divide Solid Elements

The operation enables to divide solid (3-D) finite elements and provides mesh refinement with different spacing both in the plane of the base of elements and along the height. Two types of division depending on the type of elements are provided – by additional nodes and with a specified number of steps. The first type of division is performed for elements of type 35 (15-node prism), 37 (20-node isoparametric finite element) and 38 (10-node pyramid), which have additional nodes, and provides the division only if there are such nodes. The division with a specified number of steps can be performed for these elements and all other types of solid elements.

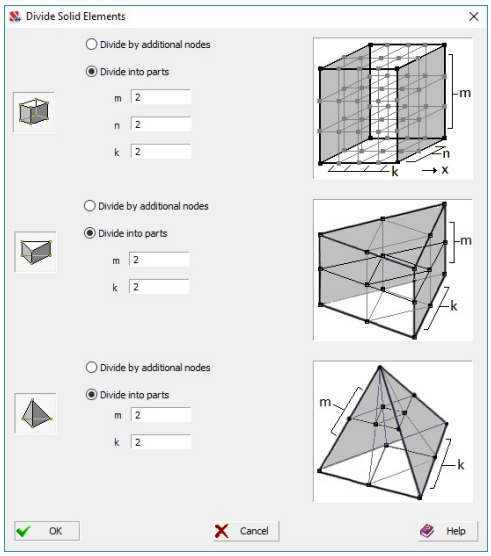


Figure 6.3-14. *The* **Divide Solid Elements** *dialog box*

The **Divide Solid Elements** dialog box (Fig. 6.3-14) appears after invoking this operation, where you have to perform the following actions:

- enable the marker buttons with an image of the types of elements that have to be divided;
- select the operation of division for each type (**Divide by additional nodes** or **Divide into parts**);
- close the dialog box by clicking the **OK** button and select the elements in the model;
- click the **OK** button in the toolbar.

Divide Plates Taking into Account the Intermediate Nodes

This operation enables to divide plate elements taking into account the nodes lying in their area (including the nodes on the edges). A node is considered to be in the area of an element if the distance from it to the nearest face does not exceed the value of accuracy of estimation of the duplicate nodes.

When the operation is performed, the quality of the obtained finite elements is not controlled. It is recommended to perform the control using the **Meshing quality** operation, the description of which is given below in this section.

Elements obtained in the result of division inherit the stiffness properties, subsoil parameters, orientation of the axes of the force output and loads specified for the original element.

Separate Elements

This operation is used in cases when it is necessary to separate adjacent elements which have common nodes from each other (e.g., to model an expansion joint or introduce elastic constraints between the adjacent nodes of these elements). In order to perform the operation it is necessary to select nodes the "cut" passes through, and elements which are separated from the selected nodes.

The essence of the performed action is that two nodes with the same coordinates will appear in the place of each selected node – the original (selected node) and a new node. Only the selected elements will be attached to the new node.

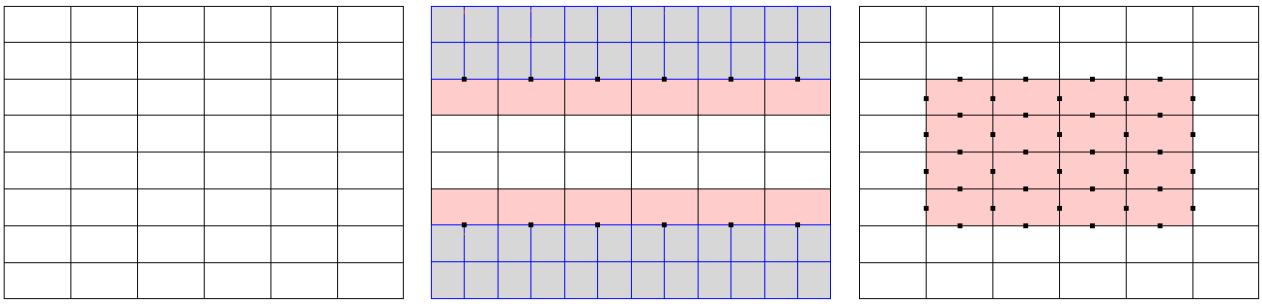
Simultaneously with the separation of elements you can also create a linear hinge along the separation line, which is modeled by an automatic specification of the merged displacements in linear directions between the pairs of duplicate nodes. The type of operation (separation or separation with the creation of a hinge) can be selected in the menu which is dropped down by clicking on the arrow on the right from the button invoking the operation.

The following procedure is recommended to perform the operation:

- invoke the operation of separation and select elements which have to be separated from the selected nodes;
- click the **Select nodes** button in the **Control the Selection** toolbar and select nodes the “cut” will pass through;
- click the **OK** button in the **Elements** section of the **Nodes and Elements** toolbar.

Attach Additional Nodes to the Elements

The operation enables to attach one additional (intermediate) node to one or more faces of the plate elements. The introduction of additional nodes enables to improve the quality of the results without refining the mesh of elements, and to connect finite elements of the design model correctly (Fig. 6.3-15).



a) *b)* *c)*

Figure 6.3-15. *Results of the application of the* **Attach additional nodes to the elements** *operation*
a) initial model; *b*) connecting meshes of elements with different division steps; *c*) adding nodes on the faces of the selected elements

In order to attach additional nodes the type of finite element must be changed beforehand to the type corresponding to the type of high-precision finite elements given in the description of the library of finite elements (see Chapter 3). Such elements include 50 and 45 shell elements, 27, 30 and 25 — deep beams, 20 and 15 — plates, and 35, 37, 38 — solid elements.

After invoking the operation the menu appears, the first item of which — **Attach the existing nodes to the faces of the elements** attaches the existing nodes on the faces of the selected elements to these elements. If the second item of the menu is selected — **Add nodes on the faces of the elements**, a new node will be added to the center of each face of each selected element (Fig. 6.3-15, c).

A standard procedure is used to perform the operation:

- invoke the operation and select its type in the drop-down menu;
- select elements the nodes are attached to in the model, and click the **OK** button in the **Elements** section of the **Nodes and Elements** toolbar.

Divide Bars Taking into Account the Intermediate Nodes

This operation enables to divide a bar into several elements taking into account nodes lying along the original bar. The accuracy of determining whether a node belongs to the original bar is defined by the **Accuracy of estimation of the duplicate nodes** parameter (the ***Environment Settings*** menu section).

The rules of inheriting the parameters of the original bar by the elements obtained in the result of division are as follows:

- stiffness properties (if they have been specified) are inherited by new elements;
- hinges will be adjacent to the same nodes as in the original bar;
- rigid inserts specified along the directions of the local X_1 axis, i.e. along the bar, will be adjacent to the same nodes as in the original bar;
- rigid inserts specified in other directions will be deleted (a respective message will be generated about it);
- loads specified for the original bar will be transferred to the new elements.

In order to perform the operation you have to select necessary elements in the model and click the button in the toolbar.

Move Elements

This operation is used to move a fragment of the design model consisting of the selected elements in the given direction. The distance the fragment is moved, and the direction are specified in the **Move Elements** dialog box (Fig. 6.3-16), which appears after invoking the operation.

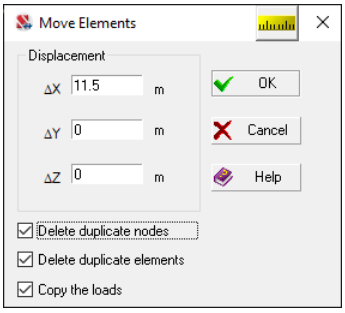


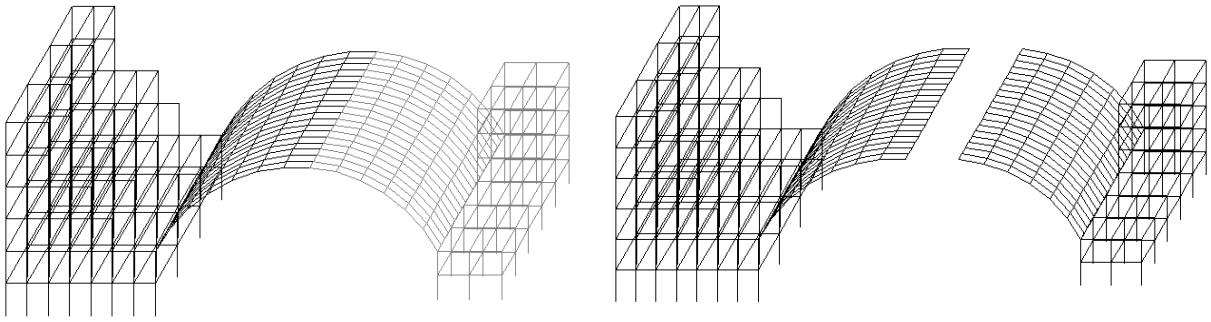
Figure 6.3-16. *The* **Move Elements** *dialog box*

In order to perform the operation you have to:

- invoke the operation, a dialog box will appear, where you have to specify the value of displacement;
- select elements of the model which are included in the fragment you want to move;
- click the **OK** button in the toolbar.

In the dialog box, you can use markers to require or refuse to delete the duplicate nodes or elements after the move is performed, and copy the loads.

If all elements adjacent to the node are moved, then this node will be moved together with the elements. Nodes belonging simultaneously to the fragment you want to move and to the stationary part are replaced by two nodes, one of which stays in place, and the other is moved together with the fragment.



a) *b*)

Figure 6.3-17. *The original model* (*a*) *and the resulting model* (*b*) *after performing the operation of moving*

If in the result of the transfer a new node will coincide with the existing one in the model, it will be deleted and only the existing node will be left in the model. An example of performing the operation is given in Figure 6.3-17.

Detach All Additional Nodes from the Faces of the Elements

This operation enables to “detach” all the additional nodes from the selected high-precision elements. In order to perform it you have to select elements with additional nodes on their edges and click the **OK** button in the toolbar.

Detached nodes are not deleted automatically after performing this operation.

Combine Elements

This operation is used to combine pairs of three-node elements that have a common face into four-node ones, and also to combine several plate elements into one. The second variant of performing this operation can be selected after clicking on the “right arrow”. The combination is performed according to the following rules:

- if the combination of three-node elements into four-node ones is selected, the possibility of combining all pairs of selected triangular FE is analyzed, the operation is performed for the allowable pairs;
- if plate elements are combined, only thee- or four-node elements can be obtained in the result without taking into account the nodes adjacent to the faces;
- if the elements involved in the operation have the same rigidities, the rigidity of the resulting element will correspond to the rigidity of the combined elements, otherwise the resulting element will not have any stiffness properties;
- loads specified for the original elements are lost;
- the control of the geometric shape of the resulting elements is performed during the process of combining.

The procedure of performing the operation is standard — invoke the operation (select a variant of performance in the menu), select elements and click the **OK** button. If the operation is not performed for the selected group of elements, you should try to perform it for a part of the group.

Considering that most parameters assigned earlier to three-node elements will be lost in the result of the combination, it is recommended to use this operation on the early stages of the model generation.

Divide Plates along the Intersection Line

The operation enables to divide plate elements along the lines of their intersection and generate consistent meshes of FE. Elements can intersect at any angle, but must not lie in one plane.

When the operation is performed, the quality of the obtained finite elements is not controlled. It is recommended to perform the control using the **Meshing quality** operation, the description of which is given below in this section.

Elements obtained in the result of division inherit the stiffness properties, subsoil parameters, orientation of the axes of the force output and loads specified for the original element.

In order to perform the operation after invoking it select the elements involved in it and click the **OK** button in the **Elements** section of the toolbar.

Extend the Bar to the Plate

This operation enables to “extend” the selected bar to the selected plate. If the point of intersection of the bar with the plate does not coincide with the existing node, the plate is divided and the mesh of FE is generated.

When the operation is performed, the quality of the obtained finite elements is not controlled. It is recommended to perform the control using the **Meshing quality** operation, the description of which is given below in this section.

Elements obtained in the result of division inherit the stiffness properties, subsoil parameters, orientation of the axes of the force output and loads specified for the original element.

In order to perform the operation after invoking it select the elements involved in it and click the **OK** button in the **Elements** section of the toolbar.

Chamfer Bars

The operation is used to connect two bars lying in one plane by “extending” them to the point of their intersection. A node is created at the intersection point of bars. If a value other than zero is specified for the radius in the **Chamfer Bars** dialog box (Fig. 6.3-18), the bars are connected by the arc of the given radius which is divided into n segments.

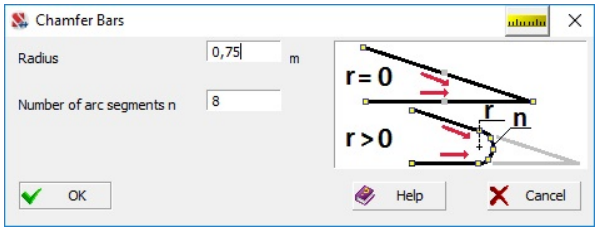


Figure 6.3-18. *The* **Chamfer Bars** *dialog box*

In order to perform the operation after invoking it select the elements involved in it and click the **OK** button in the **Elements** section of the toolbar.

Create Bars Perpendicular to the Selected One

This operation enables to create new bars coming from the selected nodes, directed perpendicular to the specified bar element and dividing it at the intersection points. An example of performing this operation is shown in Fig. 6.3-19, where

E — original bar element;

u_1, u_2, u_3 — selected nodes;

$\bar{u}_1$, $\bar{u}_2$, $\bar{u}_3$ — nodes created at the intersection points;

E_1 – E_4 — elements obtained in the result of dividing the original bar;

E_5 – E_7 — elements coming from the selected nodes.

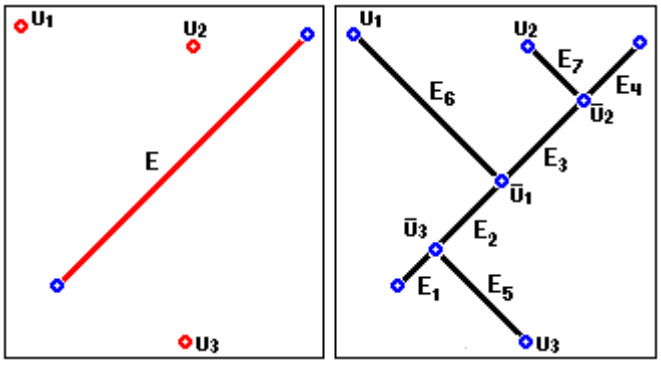


Figure 6.3-19. *Creating bars perpendicular to the selected one*

The following procedure is used to perform the operation:

- invoke the operation and select an element perpendicular to the new bars you want to create;
- select nodes in the model from which the perpendiculars will be dropped to the selected element;
- click the **OK** button in the **Elements** section of the **Nodes and Elements** toolbar.

Meshing Quality Assessment

This operation enables to assess the quality of the finite element mesh and duplicates the similar operation considered earlier when describing the generation of arbitrary meshes of finite elements on the plane (see Sec. 5.12).

Improve the Meshing Quality

If the meshing quality (it can be assessed using the operation considered above) is not satisfactory, you can use this operation to try and improve the quality criteria by an automatic transfer of nodes.

The operation is performed in two steps. On the first step after invoking the operation and before its execution a message appears about the necessity to select elements on the design model the shape of which has to be changed. If you do not select any elements and just click the **OK** button – in the toolbar, all elements of the model will be involved in the operation.

On the second step a similar message appears suggesting to select nodes the coordinates of which must not be changed (for example, if you want to improve the quality of meshing of the floor slab elements, nodes columns are adjacent to probably should be marked as fixed). If the nodes are not marked and you click the **OK** button – in the toolbar, it will be assumed that there are no restrictions, and any nodes can be moved.

Export Polygon from the Selected Plates

The operation enables to save the polygon consisting from an arbitrary set of plate finite elements lying in one plane in the format of the file with the initial data of the **Consul** section builder (extension **.con**).

Renumbering Elements

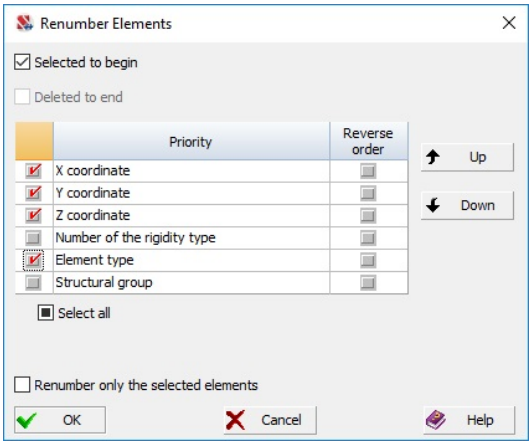


Figure 6.3-20. *The* **Renumber Elements** *dialog box*

The operation enables to renumber the elements of the design model in accordance with the rules selected by the user. The numbering can be performed in such a way so that the preselected elements get the minimum numbers, and the deleted elements are placed in the end of the list. For all other elements or only the selected ones (if the **Renumber only the selected elements** checkbox is checked) the rules for renumbering are set in the table (see figure 6.3-20).

Select the components of the coordinates of the element mass center which will be taken into account when sorting by checking the respective checkboxes, and set the priorities with the help of the **Up** and **Down** buttons. For example, if the table has the form shown in the figure, all the selected elements will have the first numbers, the deleted nodes will have the last numbers, for all other elements the minimum numbers will be assigned to the elements in the ascending order of the finite element type; if the elements have the same type, the renumbering will be performed in the ascending order of the Z coordinate and so on. The **Reverse order** checkbox can be used for the parameters, which enables to perform the renumbering in the descending order of values instead of the ascending one.

Copy Element Properties

The operation is used in cases when properties similar to the properties of the sample element have to be assigned to several elements of the design model. The set of the copied properties depends of the type of the element. The following properties are copied for bars:

- stiffness properties;
- boundary conditions (hinges and rigid inserts);
- orientation of the principal axes of inertia;
- prestressing;
- subsoil parameters;
- belonging to the groups (including groups of structural members) and the specified reinforcement;
- loads.

The following properties are copied for planar elements:

- stiffness properties;

- subsoil parameters;
- prestressing;
- orientation of force output axes;
- offset of the plate midplane;
- belonging to the groups (including groups of structural members) and the specified reinforcement;
- loads.

The **Copy Element Properties** dialog box (Fig. 6.3-21) appears after invoking the operation, which contains two groups of controls. The **Sample element** group provides the selection and visualization of the element the properties of which are copied, and the **Inheriting elements** group provides the selection and visualization of elements which receive the properties of the sample element.

The copying enables to assign certain properties of the sample to the "inheriting elements", the selection of which is performed in the **Parameters for Copying** dialog box (Fig. 6.3-22). The dialog box is invoked by clicking the respective button after selecting the "sample" and the "inheriting elements".

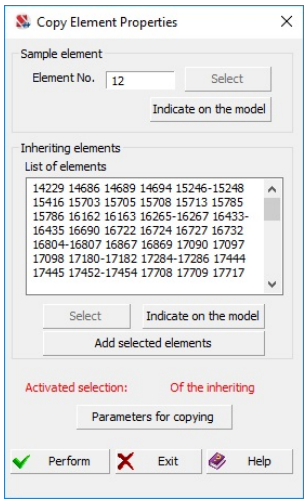


Figure 6.3-21. *The* **Copy Element Properties** *dialog box*

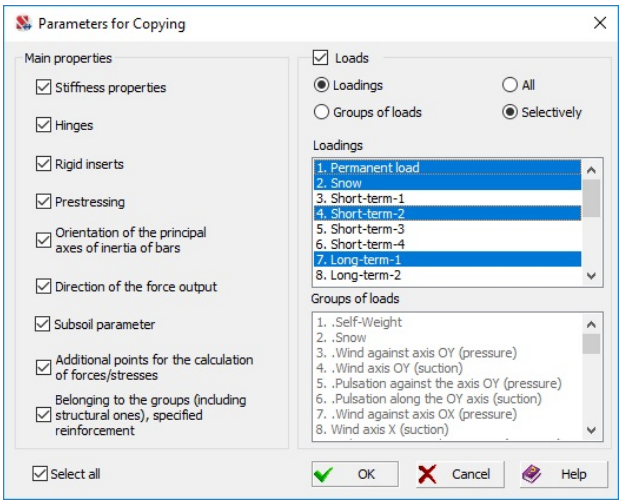


Figure 6.3-22. *The* **Parameters for Copying** *dialog box*

Clicking the **Parameters for copying** button invokes the respective dialog box (Fig. 6.3-22), where you can specify a set of properties using the checkboxes of the **Main properties** group. You can also specify the rules for copying loads in the **Loads** group (loadings or groups, all or selectively). When the loads are copied selectively, the names of the copied loadings or groups are selected from the respective lists. The following procedure is recommended for copying:

- specify the number of the sample element in the *Element No.* field in the **Copy Element Properties** dialog box and click the **Select** button. The selected element is displayed in the model in green. If it is more convenient to perform this action using the cursor, then click the **Indication on the model** button and select the necessary element with the cursor;
- click the **Parameters for copying** button and invoke the dialog box where you can assign the properties to the inheriting elements and specify the set of properties you want to copy in it;
- specify the numbers of the inheriting elements in the *List of elements* field and click the **Select** button to select these elements in the model. If it is more convenient to perform the selection using the cursor, then click the **Indication on the model** button before performing this action.
- after selecting all the inheriting elements click the **Perform** button in the **Copy Element Properties** dialog box.

Divide Bars by a Polygon

This operation enables to divide the bars in the points of their intersection by a plane defined by a polygon. The operation is used in those cases when there is no node lying on the bar in the intersection point. For example, it can be useful for dividing the bars by a meshing plane before the meshing.

The following procedure is recommended for performing this operation:

- invoke the operation, and the following message will appear: “Select nodes that are the vertices of the polygon”;
- create the polygon according to the rules similar to those for creating the polygon when specifying the meshing contour;
- the “Select elements” message will appear once you finish creating the polygon (double left click);
- select the bar elements which have to be divided in the model;
- click the **OK** button in the toolbar.

6.4. Input and Assign Parameters of Special Finite Elements

Operations of control of the input and assigning characteristics to special finite elements are concentrated in the **Nodes and Elements** toolbar of the pre-processor and can be opened by clicking the **Special elements** button .

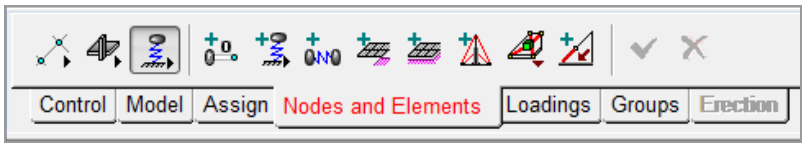


Figure 6.4-1. *The* **Special elements** *group of buttons*

The group of control buttons for creating special elements (Fig. 6.4-1) is used to perform the following operations:

— create 0-elements;

— create constraints of finite rigidity;

— create elastic constraints;

— create external node dampers;

— create single-node peripheral elements of plate on the elastic subgrade;

— create two-node peripheral elements of plate on the elastic subgrade;

— create cable-stayed elements;

— create rigid bodies;

— create and specify parameters of unilateral constraints.

Create a 0-element

The operation is used to assign characteristics and create special finite elements of type 154 (so called “0-elements”), which ensure that the linear and angular displacements are constrained in the directions of the axes of the local coordinate system of the element, and provide the analysis for the specified displacements which do not coincide with the direction of the axes of the global coordinate system.

The following procedure is recommended to create new elements:

- invoke the operation, the dialog box (Fig. 6.4-2) will appear where you have to specify the directions in the local coordinate system of the element for which the displacements are constrained or specified (the direction is specified by a number greater than zero in the respective text field);

– close the dialog box by clicking the **OK** button and create the elements according to the same rules as for creating individual bars with the help of the rubber band.

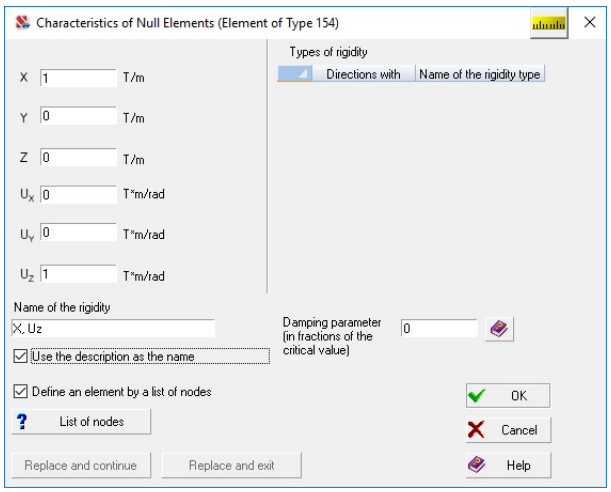


Figure 6.4-2. *The* **Characteristics of Null Elements (Element of Type 154)** *dialog box*

It should be taken into account that the first node of the elements of this type must be adjacent to the structure, and the second node – must be free (it can not belong to any element or be fixed).

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

When correcting the characteristics of the elements created earlier it is recommended to:

– invoke the operation, the dialog box will appear where you have to select a table line with the characteristics you want to change (the text fields will be filled with the respective values);
– change the values in the necessary fields or the name of the type in the respective field;
– depending on whether the characteristics of other types of rigidity will be corrected, click the **Replace and exit** button (correction of other types is not performed) or the **Replace and continue** button (involves the correction of other types).

Create Constraints of Finite Rigidity

This operation enables to create elements which model single-node constraints of finite rigidity (element of type 51), and to specify their stiffness properties. The direction and stiffness properties of constraints are specified in the **Constraints of Finite Rigidity (Element of Type 51)** dialog box (Fig. 6.4-3), which depends on the index of the model type.

The following procedure is recommended to create new constraints:

– invoke the operation, the dialog box will appear, where you have to specify the values of the stiffness of the constraint for the respective directions (a constraint is considered to be working in the selected direction if the value of stiffness for this direction is greater than zero);
– close the dialog box by clicking the **OK** button and select the nodes in the model the constraints are snapped to;
– click the **OK** button in the toolbar.

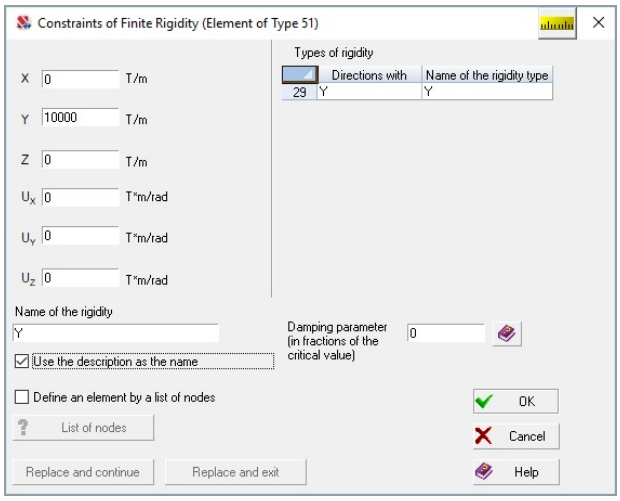


Figure 6.4-3. *The* **Constraints of Finite Rigidity (Element of Type 51)** *dialog box*

When correcting the characteristics of the already specified elastic constraints it is recommended to:

- invoke the operation, the dialog box will appear where you have to select a table line with the number of the type of rigidity you want to change;
- change the values of stiffness in the respective text fields or the name of the type of rigidity in the respective field;
- depending on whether the characteristics of other types of rigidity will be corrected, click the **Replace and exit** button (correction of other types is not performed) or the **Replace and continue** button (involves the correction of other types).

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

Create an Elastic Constraint

This operation enables to create and specify stiffness properties of elements which model an elastic constraint between two nodes (type 55).

Direction and stiffness properties of constraints are specified in the **Elastic Constraints (Element of Type 55)** dialog box (Fig. 6.4-4), which depends on the index of the model type. The following procedure is recommended to create new constraints:

- invoke the operation, the dialog box will appear, where you have to specify the values of the stiffness of the constraint for the respective directions (a constraint is considered to be working in the selected direction if the value of stiffness for this direction is greater than zero);
- close the dialog box by clicking the **OK** button and create the elements.

Elements are created according to the same rules as the individual bars – place the cursor with a target over the node and left-click, stretch the rubber band to the second node and click the mouse button again. Repeat the above actions to create another element. The node selected first will be specified as the first node of the constraint, i.e. the origin of the local coordinate system.

It should be taken into account that this type of element can have zero length, i.e. be adjacent to nodes with the same coordinates.

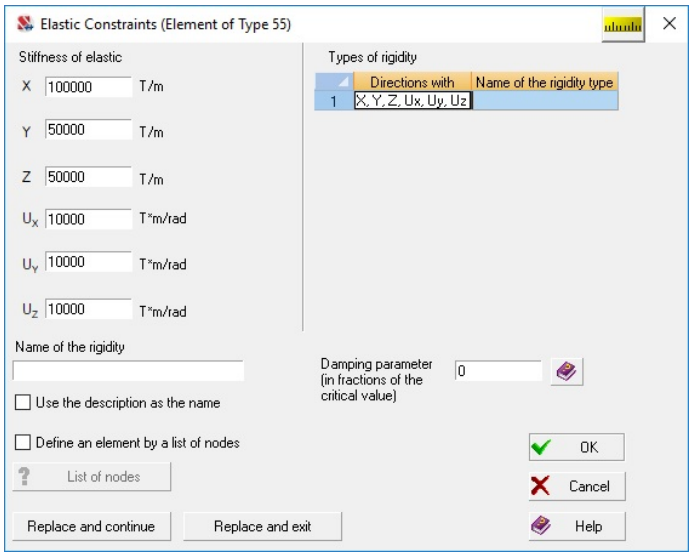


Figure 6.4-4. *The* **Elastic Constraints (Element of Type 55)** *dialog box*

Characteristics of the elastic constraint can be specified in the local or in the global coordinate system. For the constraints of zero length they will be specified only in the global coordinate system. If the nodes do not coincide, the local coordinate system of the element can be determined in the same way as for bar elements with the help of the **Orientation of the axes of inertia** operation, which is described in detail in Sec. 7.3.

When correcting the characteristics of the already specified constraints it is recommended to:

- invoke the operation, the dialog box will appear where you have to select a table line with the number of the type of rigidity you want to change;
- change the values of stiffness in the respective text fields or the name of the type of rigidity in the respective field;
- depending on whether the characteristics of other types of rigidity will be corrected, click the **Replace and exit** button (correction of other types is not performed) or the **Replace and continue** button (involves the correction of other types).

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

Create Two-node Peripheral Elements of the Plate

The operation enables to specify characteristics and create two-node finite elements which model the passive soil pressure beyond the slab. The characteristics of elements are specified in the **Two-node Peripheral Element of Plate (FE-53)** dialog box (Fig. 6.4-5). The following procedure is recommended to create new elements:

- invoke the operation, the dialog box will appear where you have to specify the characteristics of the elastic subgrade described by a pair of coefficients C_1 and C_2;
- close the dialog box by clicking the **OK** button and create the elements.

Elements are created according to the same rules as the bars taking into account the nodes they intersect.

When correcting the characteristics of the elements created earlier it is recommended to:

- invoke the operation, the dialog box will appear where you have to select a table line with the characteristics you want to change (**C_1** and **C_2** text fields will be filled with the respective values);
- change the values in the respective text fields or the name of the type of rigidity in the respective field;

– depending on whether the characteristics of other types of rigidity will be corrected, click the **Replace and exit** button (correction of other types is not performed) or the **Replace and continue** button (involves the correction of other types).

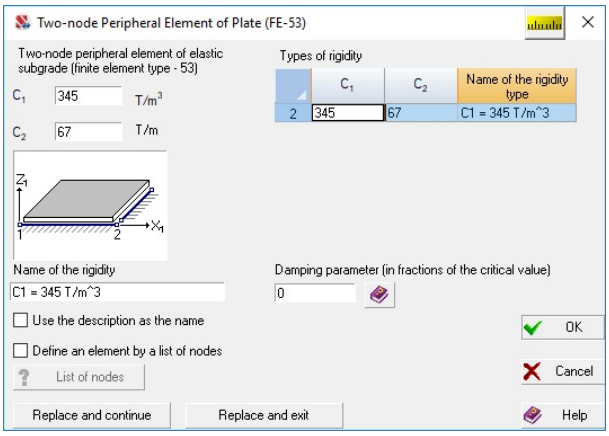


Figure 6.4-5. *The* **Two-node Peripheral Element of Plate (FE-53)** *dialog box*

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

Create Single-node Peripheral Elements of the Plate

The operation enables to specify characteristics and create single-node finite elements which model the passive soil pressure of the corner area of the foundation slab (type 54). The characteristics of elements are specified in the **Single-node Peripheral Element of Plate** dialog box (Fig. 6.4-6) and include the subsoil parameter $\mathbf{C_2}$ and the angle of the soil area **φ**.

Rules for creating and correcting the characteristics of an element are the same as the rules for a two-node element described above. Elements are created in the same way as single-node elements which model a constraint of finite rigidity (type 51).

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

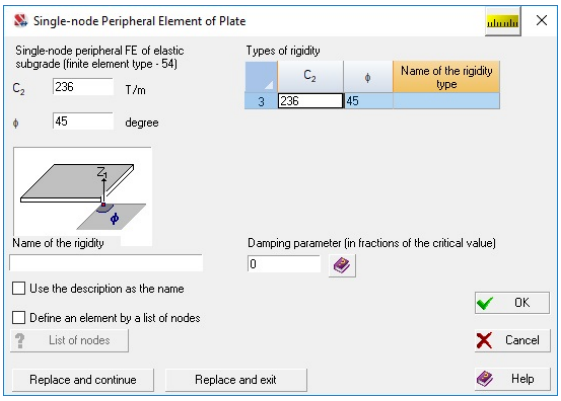


Figure 6.4-6. *The* **Single-node Peripheral Element of Plate** *dialog box*

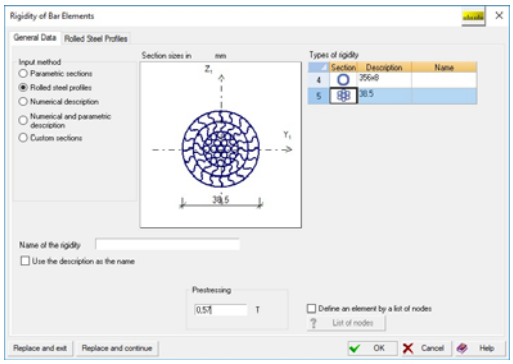
Create Cable-stayed Elements

The characteristics are specified and the cable-stayed elements (type 308) are created in the same way as two-node elements which model elastic constraints — stiffness properties of the elements are specified before creating them. The elements are created according to the same rules as bars. Information on the implementation of elements of this type and the peculiarities of the analysis of systems which include cable-stayed elements is given in Section 27.

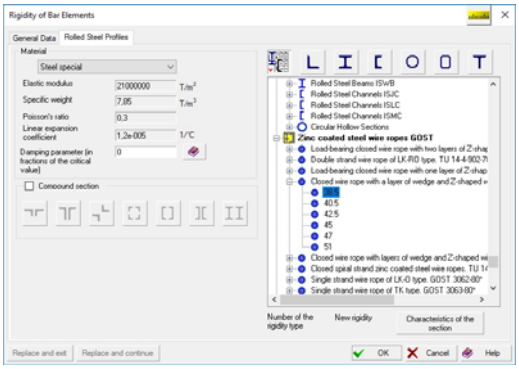
The following procedure is recommended to create new elements:

- invoke the operation, the **Rigidity of Bar Elements** dialog box will appear where you have to specify the stiffness properties of the elements of cable stays and the value of prestressing;
- close the dialog box by clicking the **OK** button and create the elements.

Stiffness properties of cable stays can be specified as parametric sections described numerically or taken from the assortment of rolled steel profiles. “Zinc coated steel wire ropes” are available in the assortment of rolled steel profiles only for cable-stayed elements. Characteristics are specified according to the same rules as for bar elements (see section *Specifying Characteristics of Nodes and Elements)*. The values of prestressing for cable stays (unlike bars) must be specified.



a)



b)

Figure 6.4-7. *The* **Rigidity of Bar Elements** *dialog box*
a) *The* **General Data** *tab* *b*) *The* **Rolled Steel Profiles** *tab*

After closing the dialog box the elements are created by stretching the rubber band between the nodes the cable stays are adjacent to.

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

Create Unilateral Constraints

This operation is used to specify the characteristics and create elements of unilateral constraints (the description of elements is given in Section 27), which enable to model the contacts between the fragments of the model, and between the model and another structure or subgrade. Elements can be single-node or two-node, and their application in the design model provides the execution of the nonlinear analysis.

The following procedure is recommended to create new elements:

- invoke the operation, the **Unilateral Constraints** dialog box (Fig. 6.4-8) will appear where you have to specify the type and characteristics of the constraint;
- close the dialog box by clicking the **OK** button and create the elements.

Single-node elements are created in the same way as single-node elements which model a constraint of finite rigidity (type 51), and two-node ones — in the same way as bars.

The characteristics of a constraint define its axial stiffness **EF**, and performance conditions (compression or tension). The direction of a constraint for single-node and two-node elements of zero length is specified in the global coordinate system. Two-node elements of non-zero length define a

constraint in the local coordinate system of the element. The restrictions of displacements in one node in two or more directions are modeled by creating several elements.

The initial state of the constraint is defined by the gap or by creating prestress specified as a prestressing force.

It is necessary to perform the following actions to correct the characteristics of unilateral constraints:

- open the **Unilateral Constraints** dialog box;
- select the line with the parameters of the element you want to correct in the **Type of rigidity** table (characteristics of the selected type of rigidity will be input into the text fields of the dialog box);
- change the parameters;
- click the **Replace and exit** button if you do not want to change the parameters of other constraints, or the **Replace and continue** button if other types of rigidity will be corrected as well.

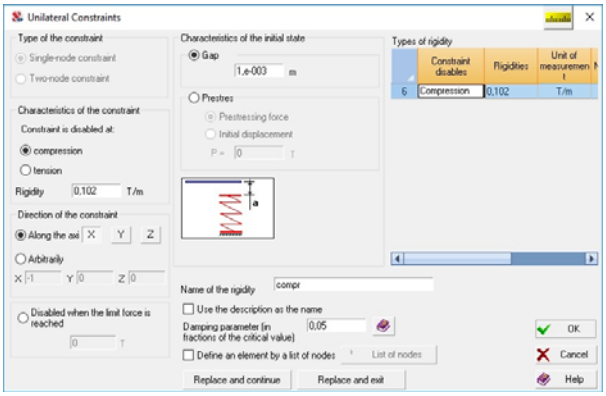


Figure 6.4-8. *The* **Unilateral Constraints** *dialog box*

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

Create External Node Dampers

This operation enables to create elements which model the external dampers responsive to the velocity of the node displacement in the directions of the degrees of freedom in the global coordinate system (element of type 56), and to specify the viscous damping ratios. The direction and the coefficients are specified in the **Node Damper** dialog box (Fig. 6.4-9.), which depends on the index of the model type.

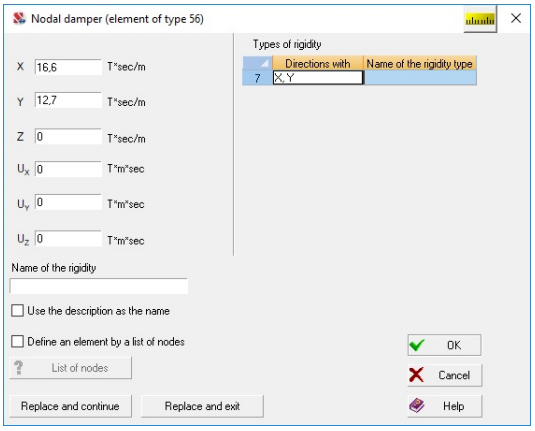


Figure 6.4-9. The **Node Damper** dialog box

The following procedure is recommended to create dampers:

− invoke the operation, the dialog box will appear, where you have to specify the values of the damping ratios for the respective directions (a damper is considered to be working in the selected direction if the value of the ratio this direction is greater than zero);

− close the dialog box by clicking the **OK** button and select the nodes in the model the dampers are snapped to;

− click the **OK** button in the toolbar.

When correcting the characteristics of the already specified node dampers it is recommended to:

− invoke the operation, the dialog box will appear where you have to select a table line with the number of the type you want to change;

− change the values of ratios in the respective text fields or the name of the type in the respective field;

− depending on whether the characteristics of other types of rigidity will be corrected, click the **Replace and exit** button (correction of other types is not performed) or the **Replace and continue** button (involves the correction of other types).

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

6.5. Input and Assign Parameters of Rigid Bodies

Operations of creating and correcting rigid bodies are invoked from the **Special elements** section of the **Nodes and Elements** toolbar. Clicking the **Rigid bodies** button drops down a menu which contains a set of operations with rigid bodies (Fig. 6.5-1). They include:

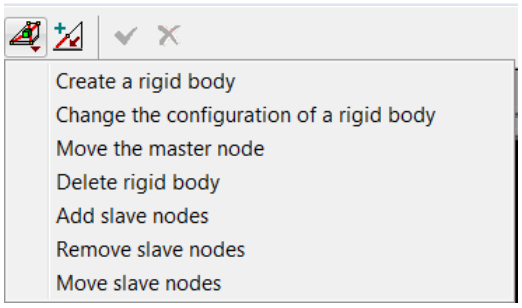


Figure 6.5-1. *Menu of control of creating and correcting rigid bodies*

- Create a rigid body;
- Change the configuration of a rigid body;
- Move the master node;
- Delete rigid body
- Add slave nodes
- Remove slave nodes
- Move slave nodes

If there are no rigid bodies in the model, the **Create a rigid body** operation will be invoked immediately.

The rigid body is a set of nodes. This set has a selected node (master node).

In terms of the results of the calculation it does not matter which node is the master node. The master node is used when drawing a rigid body (lines connecting it with all the other nodes of the finite element are drawn from this node). In order to select a rigid body you just have to indicate one of its nodes or a line connecting the master node and one of the slave nodes.

Create a Rigid Body

The **Characteristics of a Rigid Body** dialog box (Fig. 6.5-2) appears after selecting this menu item. You have to perform the following actions in this dialog box:

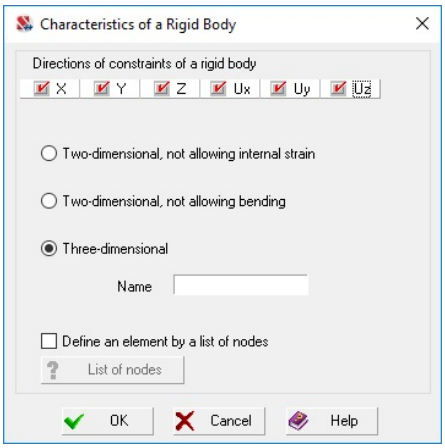


Figure 6.5-2. *The* **Characteristics of a Rigid Body** *dialog box*

- select the type of the rigid body using the radio buttons in the **Characteristics of a Rigid Body** group (two-dimensional, not allowing internal strain, two-dimensional, not allowing bending, three-dimensional);
- specify the codes of constraints (directions of the degrees of freedom) of the nodes of a rigid body (the checkboxes of directions are checked by default depending on the selected type of rigid body);
- specify the name of the rigid body if necessary.

After clicking the **OK** button in the dialog box select the nodes belonging to the rigid body with the mouse and click the button in the toolbar.

After performing the last action the rigid body will be displayed in the model as a "spider" (Fig. 6.5-3), with its master node connected with other nodes of the body by lines. The master node will be automatically assigned to the node of this new finite element located the closest to the geometric center of the set of nodes.

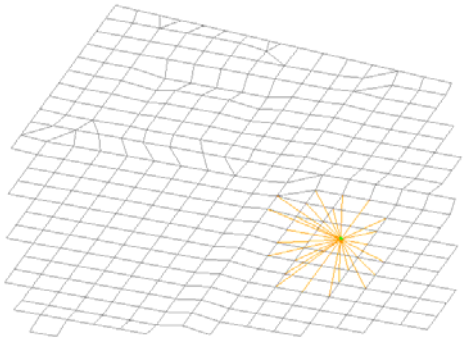


Figure 6.5-3. *Image of a rigid body.*

It is necessary to explain the difference between two types of the two-dimensional rigid body:

- a body that does not allow internal strain (disc), can be deformed as the bending of a plane of the original location of its nodes as it happens in a flat diaphragm installed to prevent distortion of the cross-section of a thin-walled bar;
- a body that does not allow bending (stamp) remains flat though this plane can move and rotate, and the nodes of such a body can move within such a plane.

The **Define an element by a list of nodes** checkbox enables to create a finite element without using the mouse, and simply by listing the numbers of nodes. The numbers of nodes can be specified in the dialog box which is invoked by clicking the **List of nodes** button (Fig. 6.3-2).

Change the Configuration of a Rigid Body

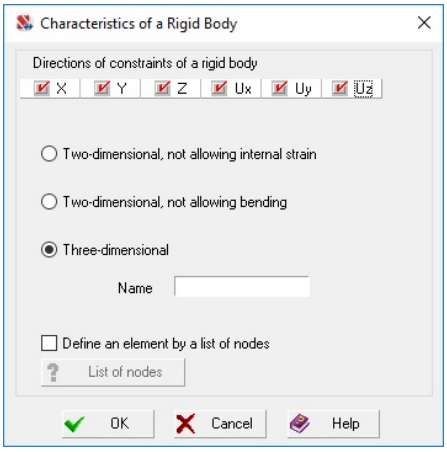


Figure 6.5-4. *The* **Characteristics of a Rigid Body** *dialog box* (correction)

This operation enables to change the type of the rigid body created earlier. The procedure for performing this operation is as follows:

- select the above operation in the menu of control of creating a rigid body;
- the **Characteristics of a Rigid Body** dialog box will appear (Fig. 6.5-4), where you can use the radio buttons to select the type of the rigid body and the checkboxes to specify the codes of constraints (directions of the degrees of freedom) of the nodes of the rigid body;
- close the dialog box by clicking the **OK** button;
- select the element of a rigid body;
- click the **OK** button in the toolbar.

Move the Master Node

Master node can be moved into any node of the rigid body. In order to do it select this operation in the menu, select the existing element of the rigid body and the node it is moved into, click the **OK** button in the toolbar.

Delete Rigid Body

In order to delete a rigid body you have to select the respective operation in the menu, select the element you want to delete, and click the **OK** button in the toolbar.

Add Slave Nodes

In order to add slave nodes you have to select the respective operation in the menu, select the element of the rigid body the nodes are added to, select the nodes of the design model which will be included in the rigid body, and click the **OK** button in the toolbar.

Remove Slave Nodes

In order to remove the slave nodes you have to select the respective operation in the menu, select the element of the rigid body the nodes will be removed from, select the nodes you want to remove, and click the **OK** button in the toolbar. This operation removes the node only from the rigid body but does not delete it from the model.

Move Slave Nodes

In order to move the slave nodes you have to select the respective operation in the menu, select the node you want to move with a cursor (is highlighted in red) and the node it will be moved into (is highlighted in blue), and click the **OK** button in the toolbar.

6.6. Groups of Nodes and Elements

Groups — named sets of nodes or elements that can be used repeatedly to perform various operations (specification of stiffness properties, constraints and loads) when selecting reinforcement and printing the results in the **Documenter**. Groups can be involved in the assembly mode along with the models. Loads from the structural fragment, tilt of the building and the foundation slab, etc. can be determined for the groups of nodes. The principle of combining objects into groups and naming the groups are completely determined by the user.

These can be particular areas of the structure, for example, floor slabs at different elevations, columns of the floor, or other sets of objects. The main thing is that the groups are available in all modes of working with the model, both at the stage of its creation or specification of parameters for processing the data in

postprocessors (for example, the calculation of DCF), and in the process of analysis and documentation of the calculation results. The graphic environment of the complex is designed in such a way so that you can always locate information within the group of nodes or elements necessary for the work.

The automatic combination of elements into groups is performed when importing the data from the **FORUM** preprocessor and in the process of transformation of data given in **.dxf** and **.dwg** formats (each layer of data is included into a separate group).

Groups described in this section can be (conventionally) called *general groups*. There are also groups of reinforced concrete and steel structural members, which are described in Chapters 19 and 20 respectively.

In order to start the work with general groups (not structural ones) switch to the **Groups** tab (Fig. 6.6-1) and invoke the toolbar (Fig. 6.6-2) using the **Working with groups of nodes and elements** operation .

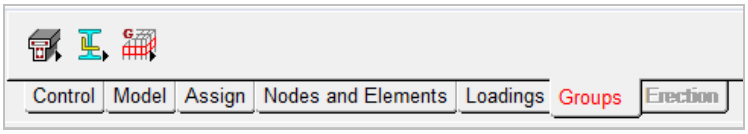


Figure 6.6-1. *The* **Groups** *tab*

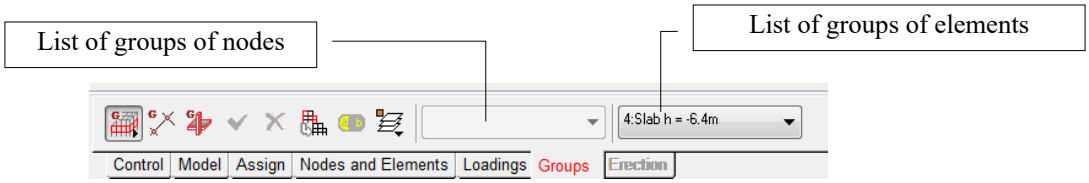


Figure 6.6-2. *The* **Groups** *section of the toolbar*

The operations of creating the groups can be invoked from the toolbar. The section contains buttons for specifying the objects of the group (nodes — or elements —), saving the group — , cancelling the selection — , cancelling the operation , Boolean operations with groups , controlling the groups and two lists for selecting the groups of nodes and elements created earlier. After loading from the list all objects of the active group are displayed as selected in the model.

Create Groups

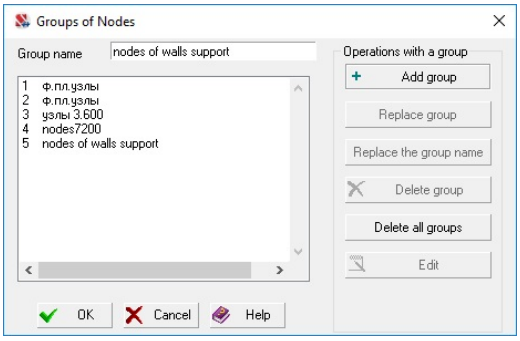


Figure 6.6-3. *The* **Groups of Nodes** *dialog box*

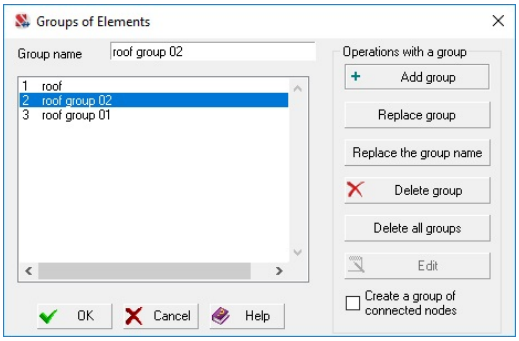


Figure 6.6-4. *The* **Groups of Elements** *dialog box*

In order to create a new group of nodes or elements it is necessary to:

- click the button defining the type of the objects of the group (nodes or elements);
- select the objects of the group in the design model;
- click the **Save/modify group** button ;
- specify the name of the group in the **Groups of Nodes** (Fig. 6.6-3) or **Groups of Elements** (Fig. 6.6-4) dialog box and click the **Add group** button;

– click the **OK** button in the dialog box.

After saving the group, its name is input into the respective list of groups in the toolbar. Before preparing the next group you have to cancel the selection of objects of the previous group by clicking the button .

When creating a group of elements, you can automatically create a group of nodes **connected** with the elements that will get the same name. Such nodes must belong to the elements of the selected group and at the same time to at least one element that is not included in this group. To do this click the **Create a group of connected nodes** button before clicking the **Add group** button.

Correct the Group

It is necessary to perform the following actions in order to change the information in the group:

– select the correct group from the list in the toolbar;
– modify it by including and/or excluding objects from it;
– click the button for saving the group;
– select the number or the name of the group you want to replace from the list;
– click the **Replace group** button;
– close the dialog box by clicking the **OK** button.

Groups can be deleted in a similar way (it should be noted that in this context, deleting a group only means removing it from the list of groups, and not deleting the elements of the group from the design model). In this case, after selecting the group in the list of groups, you have to invoke the **Save/modify group** operation, specify the number of the group in the dialog box and click the **Delete group** button. The **Edit** button enables to make changes in the list of nodes/elements in the text format.

Select Group

To select a group just indicate its name in the respective list of the toolbar. All objects of the active group are highlighted in red in the model.

Thus, operations can be performed with the objects of the group immediately after invoking it. There is one restriction — several groups can not be selected simultaneously from the lists in the toolbar, i.e. the case when both groups of nodes and groups of elements, and objects belonging to different groups are highlighted in the model at the same time is excluded. If it is necessary to obtain a combination of several groups, you have to use the color scale of indication of groups (the **Groups** button in the **Display Filters** toolbar).

Operations with Groups

This operation enables to generate a new group of nodes or elements in the result of performing one of the logical operations of union, intersection, or difference with these groups. The procedure for performing this operation is as follows:

– click the respective button invoking the operation of selection: — for working with groups of nodes or — for working with groups of element;
– invoke this operation;
– select one group involved in the operation in the left and right lists in the **Operations with Groups** dialog box (Fig. 6.6-5);
– click the button with the name of the performed operation;
– specify the name of the resulting group in the **Group Name** dialog box (Fig. 6.6-6) and click the **OK** button.

A new group will be generated in the result and input in the end of the list of groups. These actions can be performed with other groups as well, including those obtained in the result of the logical operations. Once you have completed the work, close the dialog box by clicking the **OK** button.

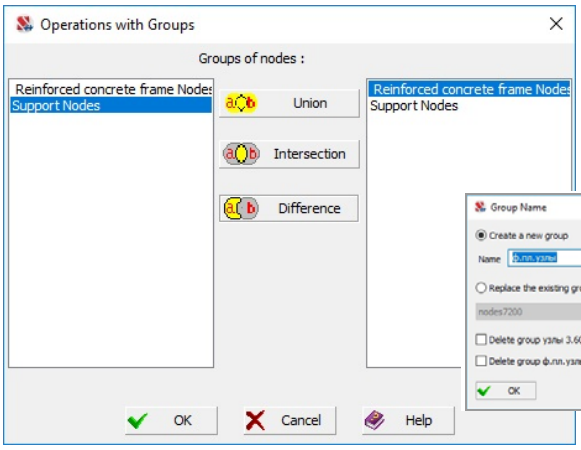


Figure 6.6-5. *The* **Operations with Groups** *dialog box*

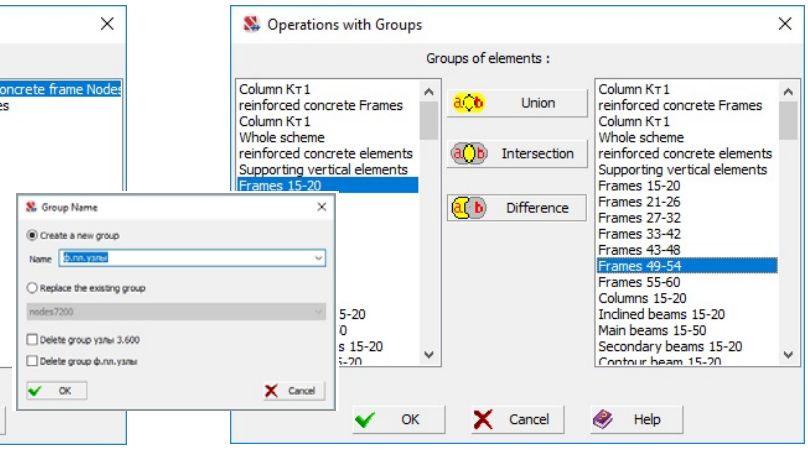


Figure 6.6-6. *The* **Group Name** *dialog box*

Control the Groups

The operation is used to control the attributes of groups and enables to perform the following actions: rename groups, sort groups by a certain criterion, delete groups and change the place of the selected group in the list of groups. If the model has both groups of nodes and groups of elements, the type of groups the actions will be performed with can be selected in the drop-down menu which appears after invoking the operation. Otherwise (there is one type of group) the menu will not appear, and the operation will be performed with the available groups.

Operations are performed in the **Control the Groups of Elements/Nodes** dialog box (Fig. 6.6-7), which contains a table with a list of groups. The columns of the table of groups include:

- the current number of the group;
- **No.** — number of the group in the order of creating;
- **Name** — name of the structural group;
- **Number of elements/nodes** in the group.

In order to rename a group place the cursor over the name of the respective group and double left click. The table line will become available for editing.

Groups in the table are sorted by an attribute, which can be selected in the **Sort by …** drop-down list. There are the following types of sorting:

- by the number (in the order of creating groups — when the **Min/Max** button is pressed; and vice versa — when the **Max/Min** button is pressed);
- by the name of the group (in the alphabetical order when the **Min/Max** button is pressed; and vice versa — when the **Max/Min** button is pressed);
- by the number of elements/nodes in the group (ascending — when the **Min/Max** button is pressed; and descending — when the **Max/Min** button is pressed).

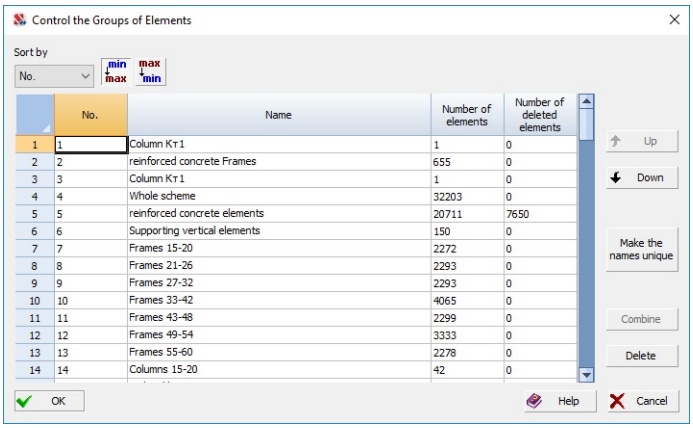


Figure 6.6-7. *The* **Control the Groups of Elements** *dialog box*

The selected groups can be deleted by clicking the respective button.

In order to change the position of one or several groups in the list you have to select these groups and move them in the list by clicking the **Up** or **Down** buttons.

If the design model has groups with the same names (for example, such groups may appear in the result of the work of the **FORUM** preprocessor), clicking the **Make the names unique** button enables to rename them automatically by adding the suffix No. XXX to the name.

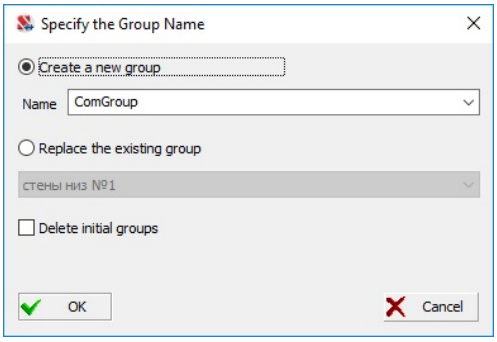


Figure 6.6-8. *Union of groups*

If several groups are selected, the **Combine** button becomes active and enables to create a group that includes all the elements/nodes of the selected groups. You have to specify the name of the new group in the dialog box (Fig. 6.6-8), or indicate that it is necessary to replace the existing group (it should be selected in the drop-down list). The **Delete initial groups** checkbox enables to delete all groups involved in the operation of combining.

All the specifications performed in the dialog box will be saved and executed after closing the dialog box by clicking the **OK** button. The position of groups in the list of the toolbar will correspond to their position in the list of the dialog box.

7. Specifying Characteristics of Nodes and Elements

This chapter considers operations performed during the generation of a design model and dealing with the specification (assignment) of characteristics for nodes and elements. They are included in the **Assign** section of the toolbar (Fig. 7-1). The familiar procedure is used:

- select the operation;
- input the data;
- select the objects;
- assign the specified values (click the **OK** button in the toolbar or press the **Enter** key in the keyboard).

When assigning certain characteristics to the nodes and elements the operation can be performed selectively depending on whether these characteristics were assigned to the selected objects. The set of rules for assigning the characteristics is given in the list in the toolbar on the right from the buttons invoking the operations. The list contains the following rules for performing the operations:

For all the selected ones — the characteristics will be assigned to all the selected objects, regardless of whether they were previously specified or not;

Without the replacement of the earlier specified ones — the characteristics will be assigned only to those selected objects which do not have any specified characteristics;

Only with the replacement of the earlier specified ones — the operation will be performed only for those objects for which the specified characteristics have already been assigned (replacement operation).

When specifying the constraints in the nodes the rules for assigning the characteristics are defined in the respective dialog box.

Figure 7-1. *The* **Assign** *section of the toolbar*

The following operations are invoked from the **Assign** toolbar:

— specify stiffness properties for bar, plate, multilayer, solid, axisymmetric, and special finite elements;

— specify subsoil parameters;

— install constraints in nodes;

— specify the types of finite elements;

— install and remove rigid inserts;

— displacement of the plate midplane;

— install and delete hinges;

— specify the orientation of the local coordinate axes of the elements;

— specify additional points for the calculation of forces/stresses;

— merge displacements in nodes;

— change the direction of the local X_1 axis of bars;

— change the direction of the local Z_1 axis of plates;

— specify the directions of the stress output for plates and solid elements;

— linear resizing of cross sections;

— delete duplicate types of rigidity;

— renumber rigidities;

— specify elements to take into account nonlinearity;

— invokes **Cross** for the calculation of subsoil parameters;

— specify prestressing of elements;

— specify the reinforcement layout of plate elements (see Chapter 19);

— specify the reinforcement layout of bars (see Chapter 19).

— specify the additional data for elements

7.1. Specifying Stiffness Properties

The set of parameters describing the stiffness properties of elements depends on the type of elements. Therefore the software provides operations for assigning such properties to bar, plate, solid and other types of finite elements.

Specifying Stiffness Properties for Bar Elements

Physical and mechanical characteristics of bar elements can be specified in the form of a numerical description, or calculated by the software taking into account the type and sizes of the standard parametric sections or a profile selected from the assortment of rolled steel profiles. These characteristics can be also obtained with the help of **Section Builder**, **Consul** or **TONUS**.

Data are specified in the multi-tab **Rigidity of Bar Elements** dialog box (Fig. 7.1-1). The set of tabs of this dialog box depends on the method of describing the rigidity. You can switch between tabs by clicking on the respective tab. The method of specifying the stiffness properties can be selected in the **General Data** tab by selecting the respective radio button. Tabs corresponding to the selected method will appear. The first tab also contains the **Type of rigidity** table with the following information for all the sections specified earlier: number of the type of rigidity, image of the section, description of the section (sizes of the section – for all sections except for the numerical description), name of the type of rigidity (if it has been specified).

The designation **EF** is used in the table for rigidities the characteristics of which are specified as a numerical description, and icon — for characteristics obtained with the help of **Section Builder**, **Consul** or **TONUS**. Combinations of the designation **EF** and the icon of one of the parametric sections are used for the characteristics specified with the help of the numerical and parametric description.

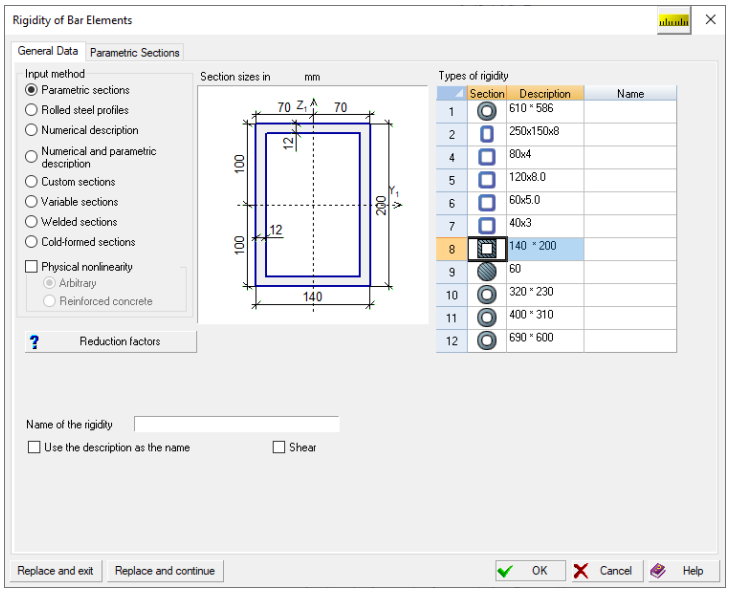


Figure 7.1-1. *The* **Rigidity of Bar Elements** *dialog box*

Name of the type of rigidity can be specified in the respective field. If the **Use the description as the name** checkbox is checked, the contents of the **Description** column of the **Type of rigidity** table will be input into this field.

When working with stiffness properties it is important to distinguish between the modes of the data input and their correction. In the first case the characteristics of the new type of rigidity are input and this type is assigned to the elements of the model, in the second case – the characteristics specified earlier are corrected.

Specifying a New Type of Rigidity

The following procedure is recommended to specify a new type of rigidity:

- select the method of specifying the stiffness properties in the **General Data** tab and specify the name of the type of rigidity;
- open the tab for specifying the stiffness properties by clicking on it;
- specify the necessary data;
- click the **OK** button in the dialog box, the number of the type of rigidity will be assigned and the dialog box will close;
- select the elements in the model this type of rigidity is assigned to;
- click the **OK** button in the **Assign** section or press the **Enter** key in the keyboard.

If the respective filter is active (the button is pressed) when you assign the stiffness properties to elements, the results will be shown in the model (by numbers — number of the type of rigidity is displayed or by color — color identification of elements).

Parametric Sections

The procedure for the data input is not regulated, however, to avoid errors, it is better to work out a certain sequence of actions, such as this:

- select the **Parametric sections** radio button;
- switch to the **Parametric Sections** tab (Fig. 7.1-2);
- select the material in the respective list or select “Other” in the list and specify the characteristics of the material (values of the elastic modulus, Poisson’s ratio, etc.);
- specify the sizes of the section (it should be noted that the units of measurement can be changed in the process of input) and click the **Characteristics of the section** button, which will enable to check the correctness of the specified data in the **Section Properties** dialog box (Fig. 7.1-3);
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements of the model.

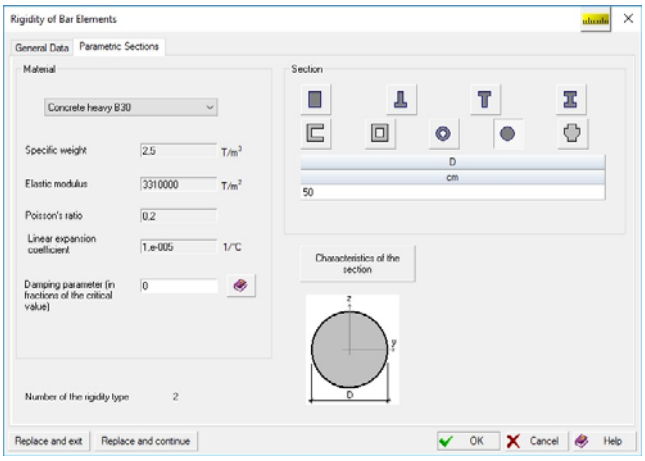


Figure 7.1-2. *The* **Parametric Sections** *dialog box*

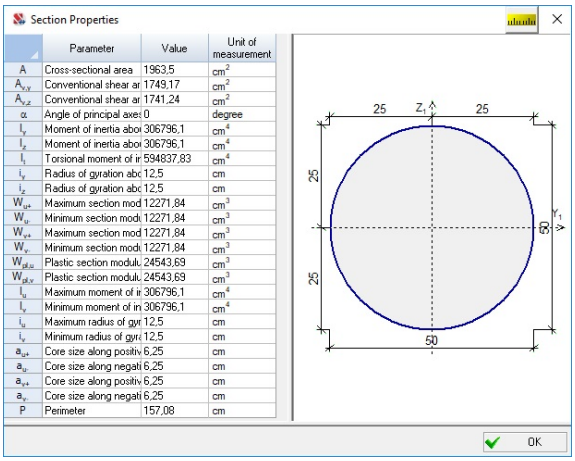


Figure 7.1-3. *The* **Section Properties** *dialog box*

Numerical Description

This tab (Fig. 7.1-4) includes the most necessary set of text fields, which enables to specify the stiffness properties for any type of bar. Therefore, before specifying the values of the properties, you have to select the radio button with the name of the type of the elements this type of rigidity will be assigned to. Only those text fields will remain available for input which must or can be filled for the given type of elements.

The above procedure for the data input and assigning rigidities to the elements of the model is used when working with this tab:

- select the **Numerical description** radio button;
- switch to the **Numerical Description** tab (Fig. 7.1-4);
- select the material in the respective list or select “Other” in the list and specify the characteristics of the material (values of the elastic modulus, Poisson’s ratio, etc.);
- select the radio button with the name of the type of the elements this type of rigidity will be assigned to;
- fill the text fields available for input with data;
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements of the model.

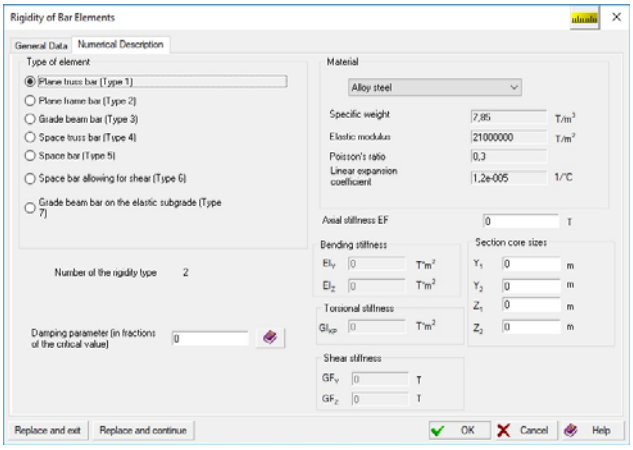


Figure 7.1-4. *The* **Numerical Description** *tab*

Working with the Assortment of Rolled Steel Profiles

The procedure for assigning stiffness properties to bar elements the section of which is taken from the assortment of rolled steel profiles is as follows:

- select the **Rolled steel profiles** radio button;
- switch to the **Rolled Steel Profiles** tab (Fig. 7.1-5);

- select the type of steel in the list of materials or select “Other” and specify the value of the specific weight of steel, elastic modulus, Poisson’s ratio and the linear expansion coefficient in the respective text fields of the **Material** group;
- select the assortment, type and size of the profile in the list;
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements of the model.

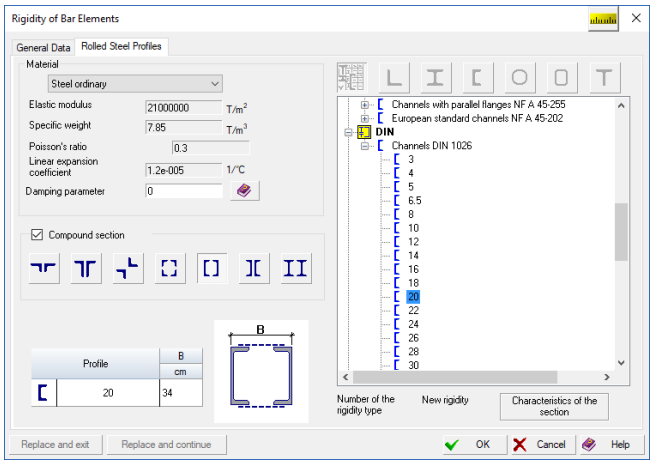


Figure 7.1-5. *The* **Rolled Steel Profiles** *tab*

The buttons above the list of profiles enable to work with the full list of profiles ( )or limit the list to profiles of a certain type (angles, I-beams, ...).

The full list of assortments included in the **SCAD Office** system is given in the **Appendix**. They can be included in or excluded from the list of available assortments in the **Sections** tab of the **Parameters** dialog box, which can be invoked from the ***Settings*** menu.

If a compound section from rolled steel profiles of one of seven types provided in the software is selected, the following procedure will be used:

- check the **Compound section** checkbox in the **Rolled Steel Profiles** tab and click the button with the image of the respective section;
- select the assortment, type and size of the profiles (only the profiles included in the section will be available) in the list of assortments;
- specify the geometric parameters of arrangement typical for the selected section;
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements of the model.

It is recommended to follow the recommendations of codes when specifying the thickness of a gusset plate in sections from double angles. For example, in SNiP the value of the gap is limited by the interval 0.8t…2t, where t is the thickness of the angle.

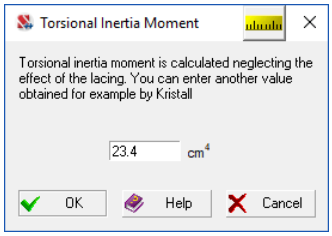


Figure 7.1-6. *The* **Torsional Inertia Moment** *dialog box*

For sections from double channels and I-beams the torsional inertia moment is calculated neglecting the effect of the lacing. If after specifying the parameters of arrangement you click the **Characteristics of the section** button (Fig. 7.1-2), the **Torsional Inertia Moment** dialog box (Fig. 7.1-6) with the respective message will appear. You can specify a clarified value of the moment of inertia obtained for example by **Kristall**.

Variable Sections

The **Variable Sections** operation is used to specify the stiffness properties of the elements with sections varying along the length of the bar. The data are specified in a similar way as for the parametric sections. The

difference is that the parameters should be specified at the beginning and at the end of the element (it is assumed that all parameters change linearly along the bar).

Welded Sections

The **Welded sections** option in the **Input method** group of the **General Data** tab is used to specify the stiffness properties of bar elements made from welded sheet steel.

The following types of welded sections are implemented in the **Welded Sections** tab: welded I-beam (equal and unequal), welded hollow section, welded T-section (with a bottom or with a top flange), and a welded cross-shaped section (see Fig. 7.1-7). **High-quality steel** is selected by default in the **Material** list and all its characteristics are specified (values of the specific weight, elastic modulus, Poisson's ratio etc.). The user can edit them if necessary. This tab also contains a table where you have to specify the sizes of the welded sections: thickness and width of the flanges, height and thickness of the web/webs (the units of measurement can be changed in the process of input). Clicking the **Characteristics of the section** button invokes the **Section Properties** dialog box (Fig. 7.1-8) where you can check the correctness of the specified data.

After closing the dialog box by clicking the **OK** button you have to assign the specified type of rigidity to the bar elements of the design model.

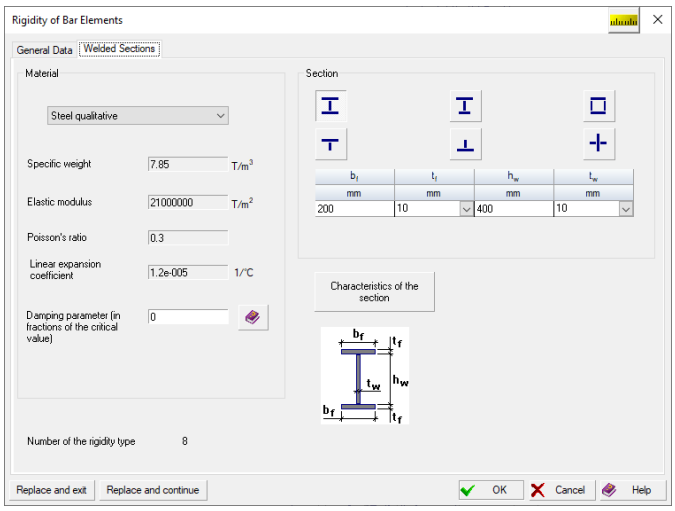


Figure 7.1-7. *The* **Welded Sections** *tab*

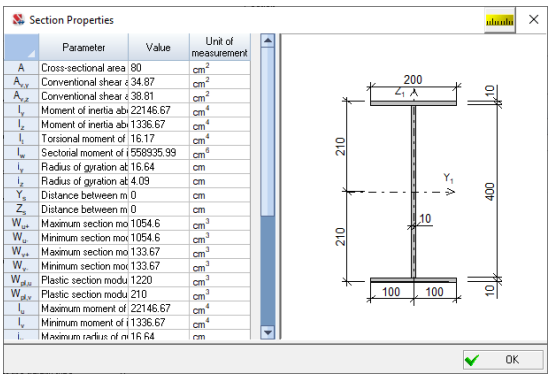


Figure 7.1-8. *The* **Section Properties** *dialog box*

Cold-Formed Profiles

The **Cold-formed profiles** option in the **Input method** group of the **General Data** tab is used to specify the stiffness properties of bar elements made from single or compound cold-formed profiles.

The following types of cold-formed steel sections are implemented in the **Cold-Formed Profiles** tab: cold-formed steel angles, C-, Z-, sigma-shaped sections and hat channels with or without edge fold stiffeners (single or double), various compound and lattice cold-formed sections (see Fig. 7.1-9). It should be noted that compound cold-formed thin-walled sections are assembled by bolting individual profiles with a certain spacing along the length of the bar, which does not require any additional structural members.

High-quality steel is selected by default in the **Material** list and all its characteristics are specified (values of the specific weight, elastic modulus, Poisson's ratio etc.). The user can edit them if necessary. This tab also contains a table where you have to specify the sizes of cold-formed steel sections. Clicking the **Characteristics of the section** button invokes the **Section Properties** dialog box (Fig. 7.1-10) where you can check the correctness of the specified data.

After closing the dialog box by clicking the **OK** button you have to assign the specified type of rigidity to the bar elements of the design model.

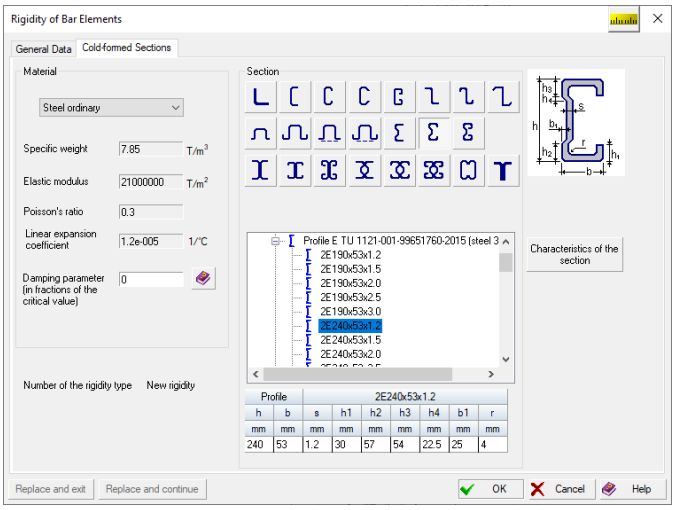


Figure 7.1-9. *The* **Cold-Formed Profiles** *tab*

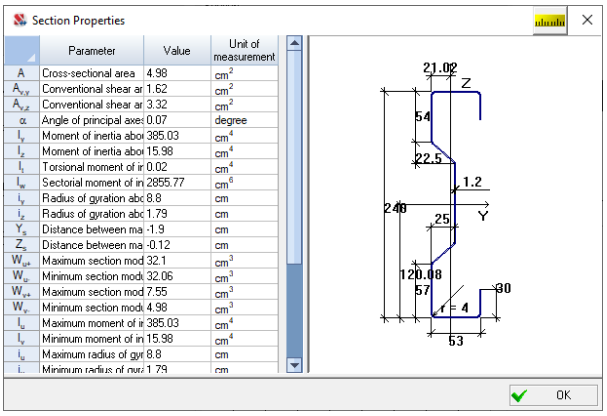


Figure 7.1-10. *The* **Section Properties** *dialog box*

Numerical and Parametric Description

Numerical and Parametric Description

Such a description combines two types of the representation of stiffness properties — in the form of parametric sections and numerical. This enables to both provide information on the shape of the section, and to present the characteristics of the section in the form of a numerical description. Such representation of stiffness properties can be useful when selecting the reinforcement for a custom section, where a numerical description of such a section (for example, obtained in the section builder) is used in the calculation process, and the selection is performed according to the rules of the parametric section selected by the user.

In this case the dialog box contains three tabs **General Data**, **Parametric Sections** and **Numerical Description**. The stiffness properties are specified in the given order of the tabs and according to above rules for the respective tabs.

Custom Sections

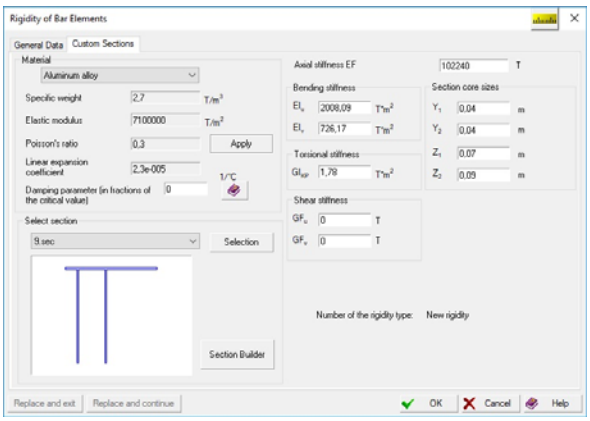


Figure 7.1-11. *The* **Custom Sections** *tab*

The **Custom sections** operation enables to specify the stiffness properties for elements the sections of which have been created with the help of **Section Builder**, **Consul** or **TONUS**. The following procedure is recommended:

- select the **Custom sections** radio button in the **General Data** tab;
- select the material or "Other" and specify the characteristics of the material in the respective text fields of the **Material** group in the respective tab (Fig. 7.1-11);
- click the **Selection** button in the **Select section** group and select the necessary section (files with .sec, .cns or .tns extension);

- once the image of the section appears in the information window of the **Select section** group, click the **Apply** button, and the stiffness properties of the section will be input in the respective text fields;
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements.

Only the maximum (**I_U**) and the minimum (**I_V**) moments of inertia are known for a section of an arbitrary shape created with the help of one of the section builders. After clicking the **Apply** button the value of EI_U is input into the EI_Y text field, and the value of EI_V — into the EI_Z text field. The position of the principal axes of inertia is fixed by the rotation angle of the structural axes described in Sec. 3.3

Stiffness Reduction

Clicking the **Reduction factors** button invokes a dialog box where you can specify the reduction factors for axial and bending stiffness and thereby take into account the effects related to the stiffness reduction due to cracking (see, for example, Sec. A.8.6 of SP 5.03.01-2020).

Correcting the Selected Section

It is necessary to perform the following actions for correcting during the specification of the stiffness properties of the section created earlier by one of the section builders and used to specify the rigidities for the elements of the model:

- click the **Section Builder** button in the **Custom Sections** tab, and the application for generating sections which was used to create the considered section will be invoked;
- make all the necessary changes of the section and save the modified section under the same name as the original one;
- close the application for generating sections, and the control will return to the **Custom Sections** tab (the image of the modified section will appear in the information window);
- after clicking the **Apply** button the stiffness properties of the section will be recalculated taking into account the performed changes;
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements.

Correcting the Characteristics of the Type of Rigidity Specified Earlier

It is necessary to perform the following actions to correct the characteristics of the type of rigidity specified earlier :

- invoke the **Assign rigidities for bars** operation;
- select the line with the number of the type of rigidity you want to change in the **Type of rigidity** table;
- correct the characteristics (you can also change the method of specifying the rigidity, the name of the type of rigidity, etc.);
- click one of the buttons — **Replace and exit** or **Replace and continue**. In the first case the dialog box will close after changing the characteristics, and in the second one you can continue correcting other types of rigidity.

The number of the type of rigidity does not change after performing the above actions. If after correcting the parameters you click the **OK** button, a new type of rigidity will be created, and the characteristics of the type specified earlier will remain without changes.

It should be noted that closing the **Rigidity of Bar Elements** dialog box by clicking the **Replace and exit** button does not allow to assign the modified parameters to the other elements of the model. This operation is available only after closing the dialog box by clicking the **OK** button.

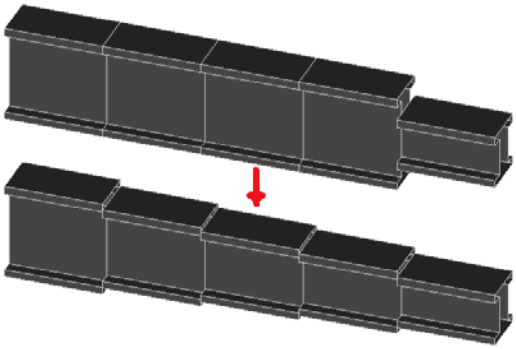
Assigning the Type of Rigidity Specified Earlier to the Elements

It is necessary to perform the following actions to assign the type of rigidity specified earlier to the elements:

- invoke the **Assign rigidities for bars** operation;
- select the line with the number of the respective type of rigidity in the **Type of rigidity** table;
- close the dialog box by clicking the **OK** button;
- assign the selected type of rigidity to the elements of the model.

Linear Resizing of Bar Elements

This operation enables to change the sizes of parametrically defined bar element cross-sections. As a result, the geometric dimensions of intermediate bars in a chain will be adjusted according to linear interpolation of the cross-sectional sizes of its extreme elements, which allows you to model a variable cross-section bar. An example of such a transformation is shown in the figure below.



In order to perform this operation, invoke it in the **Assign** toolbar, select a chain of bars that lie on a straight line, and click the **OK** button.

Specifying Stiffness Properties for Plate Elements

The **Rigidities of Plates** dialog box (Fig. 7.1-12) is used to specify the stiffness properties for shell, plate and deep beam elements.

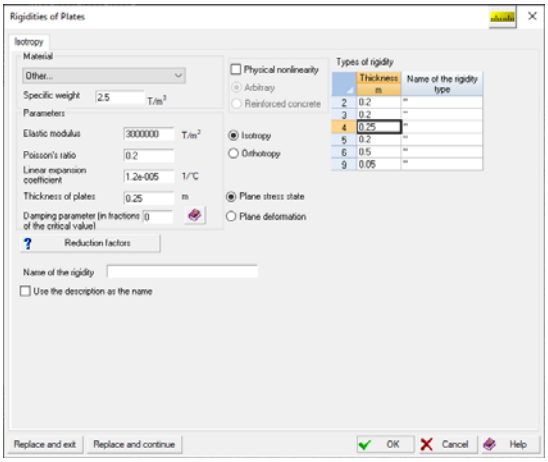


Figure 7.1-12. *The* **Rigidities of Plates** *dialog box*, *the* **Isotropy** *tab*

The procedure for specifying the stiffness properties for plate elements when performing a linear calculation is as follows:

a) isotropic material:

- invoke the **Assign rigidities for plates** operation;

- select the material or "**Other**" in the list of materials and specify the characteristics of the material in the respective text fields of the **Material** group;
- specify the plate thickness;
- select the **Isotropy** radio button;
- select the problem of the theory of elasticity which will be used to perform the analysis — **Plane stress state** (this theory is selected by default) or **Plane deformation**;
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements of the model;

b) orthotropic material:

- invoke the **Assign rigidities for plates** operation;
- select the **Orthotropy** radio button (the **Orthotropy** tab will appear in the dialog box);
- specify the plate thickness;
- select the problem of the theory of elasticity which will be used to perform the analysis — **Plane stress state** (this theory is selected by default) or **Plane deformation**;
- specify the values of the elastic modulus, Poisson's ratio and the linear expansion coefficients in the directions X_1 and Y_1, and the value of the shear modulus in the **Orthotropy** tab (Fig. 7.1-13);
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements of the model.

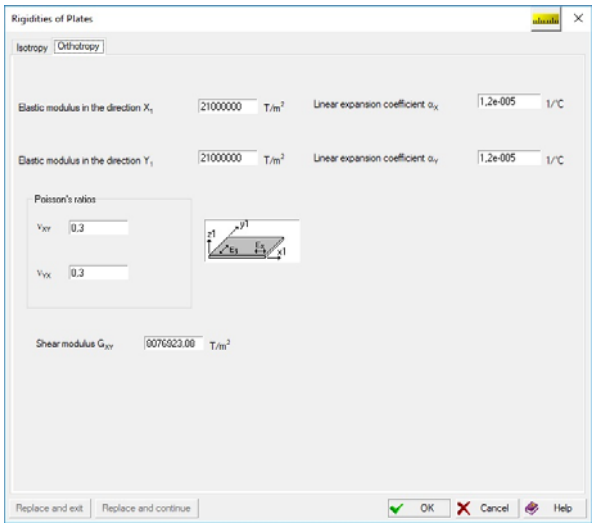


Figure 7.1-13. *The* **Rigidities of Plates** *dialog box*, *the* **Orthotropy** *tab*

It is necessary to perform the following actions to correct the characteristics of the type of rigidity specified earlier :

- invoke the **Assign rigidities for plates** operation;
- select the line with the number of the type of rigidity you want to change in the **Type of rigidity** table;
- correct the characteristics specified earlier;
- click one of the buttons — **Replace and exit** or **Replace and continue**. In the first case the dialog box will close after changing the characteristics, and in the second one you can continue correcting other types of rigidity.

The number of the type of rigidity does not change after performing the above actions.

If after correcting the parameters you click the **OK** button, a new type of rigidity will be created, and the characteristics of the type specified earlier will remain without changes.

Stiffness Reduction

Clicking the **Reduction factors** button invokes a dialog box where you can specify the reduction factors for bending and membrane stiffness and thereby take into account the effects related to the stiffness reduction due to cracking (see, for example, Sec. A.8.6 of SP 5.03.01-2020).

Specifying Stiffness Properties for Solid Elements

Stiffness properties can be specified in the **Rigidity of Solid Elements** dialog box (Fig. 7.1-14). If the material is isotropic, the characteristics include four parameters (elastic modulus, Poisson's ratio, specific weight, and the linear expansion coefficient) and in this case the following procedure can be recommended:

- invoke the **Assign rigidities for solid elements** operation;
- select the material or "**Other**" in the list of materials and specify the characteristics of the material in the respective text fields of the **Material** group in the **Rigidity of Solid Elements** dialog box;
- select the **Isotropy** radio button (it is selected by default);
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements of the model.

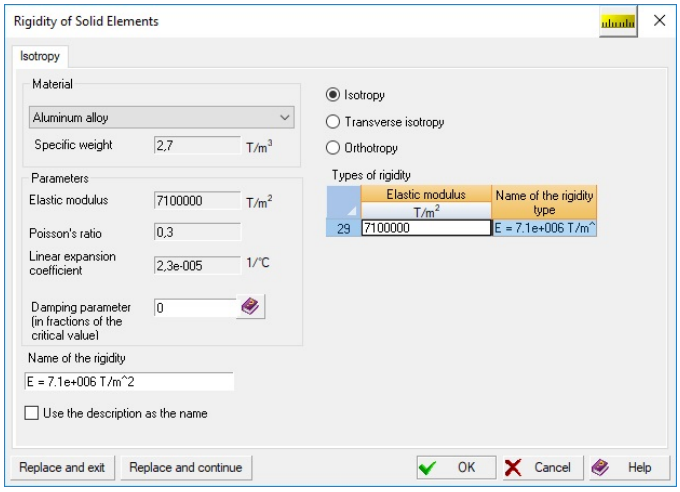


Figure 7.1-14. *The* **Rigidity of Solid Elements** *dialog box*

When the transverse isotropy is considered, the **Transverse Isotropy** tab (Fig. 7.1-15) appears in the dialog box, where you can specify the characteristics of the material.

A similar tab but with the respective set of characteristics (Fig. 7.1-16) appears when you have to specify the orthotropic properties of the material (the specified values should satisfy the symmetry conditions described in Section 3.5).

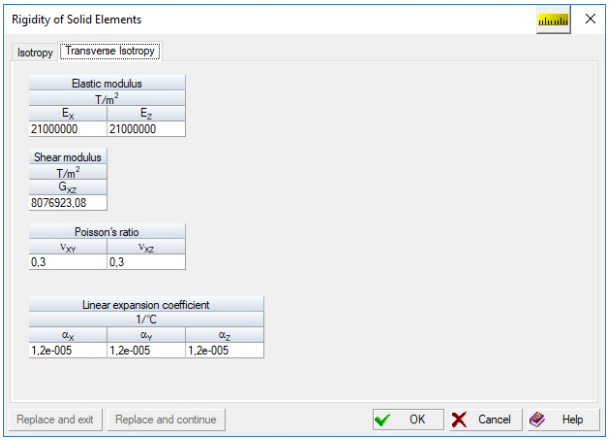


Figure 7.1-15. *The* **Transverse Isotropy** *tab*

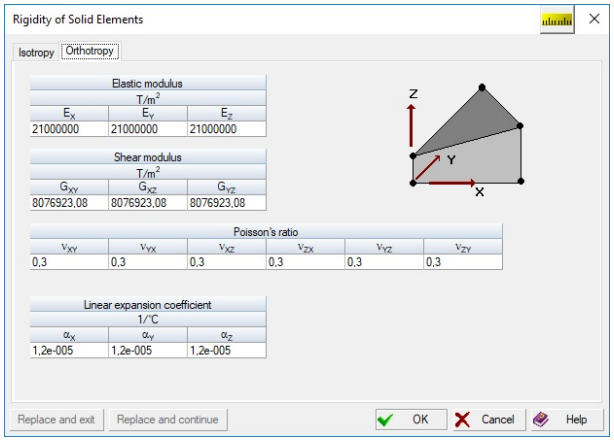


Figure 7.1-16. *The* **Orthotropy** *tab*

It is necessary to perform the following actions to correct the characteristics of the type of rigidity specified earlier :

- invoke the **Assign rigidities for solid elements** operation;
- select the line with the number of the type of rigidity you want to change in the **Type of rigidity** table;

- correct the characteristics specified earlier;
- click one of the buttons — **Replace and exit** or **Replace and continue**. In the first case the dialog box will close after changing the characteristics, and in the second one you can continue correcting other types of rigidity.

The number of the type of rigidity does not change after performing the above actions.

If after correcting the parameters you click the **OK** button, a new type of rigidity will be created, and the characteristics of the type specified earlier will remain without changes.

Specifying Stiffness Properties for Multilayer Elements

This operation enables to specify the stiffness properties for multilayer elements (a model including the multilayer elements must have the type 8 or 9).

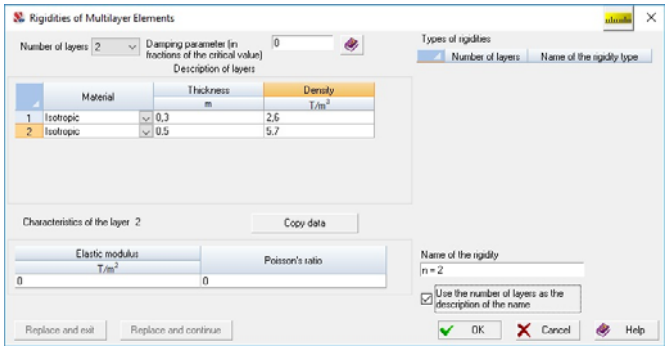


Figure 7.1-17. *The* **Rigidities of Multilayer Elements** *dialog box*

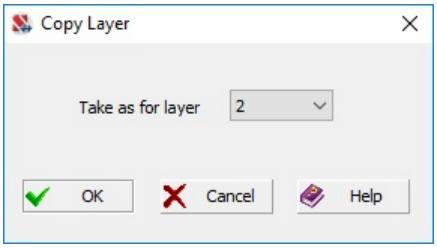


Figure 7.1-18. *The* **Copy Layer** *dialog box*

This operation invokes the **Rigidities of Multilayer Elements** dialog box (Fig. 7.1-17). The procedure for specifying the data in this dialog box is as follows:

- select the number of layers in the elements of the given type of rigidity from the list;
- specify (select from the list) the properties of the material for each layer in the **Material** column, the thickness of the layer in the **Thickness** column, and the density of the material of the layer in the **Density** column of the **Description of layers** table;
- select the line with the number of the layer the stiffness properties are specified for in the **Description of layers** table;
- fill in the table with the characteristics of the selected layer, specifying the values of the elastic modulus and the Poisson's ratio;
- select the number of the next layer in the **Description of layers** table and repeat the above actions of the specification of the characteristics;
- click the **OK** button and assign the specified type of rigidity to the elements of the model.

If the subsequent layer has the same characteristics as another layer specified earlier, you can use the **Copy data** button to assign them. Clicking this button invokes the **Copy Layer** dialog box with a list (Fig. 7.1-18), where you can select the number of the original layer.

It is necessary to perform the following actions to correct the characteristics of the type of rigidity specified earlier :

- invoke the **Rigidities of multilayer elements** operation;
- select the line with the number of the type of rigidity you want to change in the **Type of rigidity** table;
- correct the characteristics specified earlier;
- click one of the buttons — **Replace and exit** or **Replace and continue**. In the first case the dialog box will close after changing the characteristics, and in the second one you can continue correcting other types of rigidity.

The number of the type of rigidity does not change after performing the above actions.

If after correcting the parameters you click the **OK** button, a new type of rigidity will be created, and the characteristics of the type specified earlier will remain without changes.

Specifying Stiffness Properties for Axisymmetric Elements

When the structures modeled by axisymmetric elements are considered (type of model 11), the stiffness properties of elements are specified in the **Rigidities of Axisymmetric Elements** dialog box (Fig. 7.1-19).

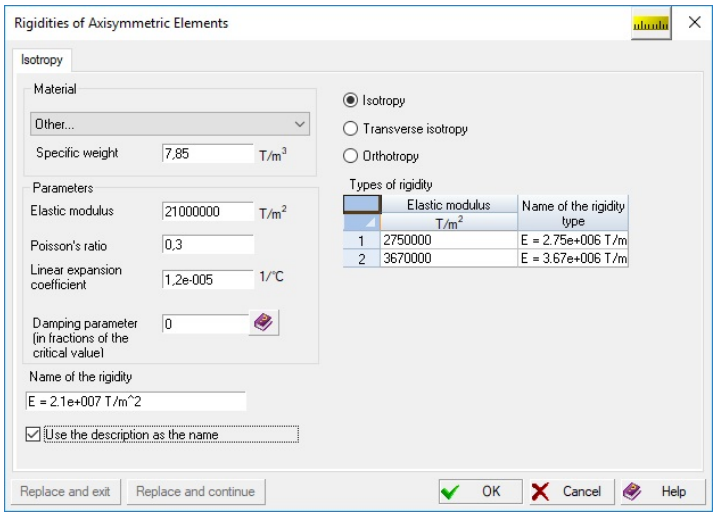


Figure 7.1-19. *The* **Rigidities of Axisymmetric Elements** *dialog box*

The following procedure can be recommended to specify rigidities for axisymmetric elements:

- invoke the **Assign rigidities for axisymmetric finite elements** operation;
- select the material or "**Other**" in the list of materials and specify the characteristics of the material in the respective text fields in the **Rigidities of Axisymmetric Elements** dialog box;
- select the **Isotropy** radio button (it is selected by default);
- close the dialog box by clicking the **OK** button;
- assign the specified type of rigidity to the elements of the model.

When the transverse isotropy is considered, the **Transverse Isotropy** tab (Fig. 7.1-20) appears in the dialog box, where you can specify the characteristics of the material. A similar tab, but with the respective set of characteristics (Fig. 7.1-21), appears when you have to specify the orthotropic properties of the material.

It is necessary to perform the following actions to correct the characteristics of the type of rigidity specified earlier :

- invoke the considered operation;
- select the line with the number of the type of rigidity you want to change in the **Type of rigidity** table of the **Rigidities of Axisymmetric Elements** dialog box;
- correct the characteristics specified earlier;
- click one of the buttons — **Replace and exit** or **Replace and continue**. In the first case the dialog box will close after changing the characteristics, and in the second one you can continue correcting other types of rigidity.

The number of the type of rigidity does not change after performing the above actions.

If after correcting the parameters you click the **OK** button, a new type of rigidity will be created, and the characteristics of the type specified earlier will remain without changes.

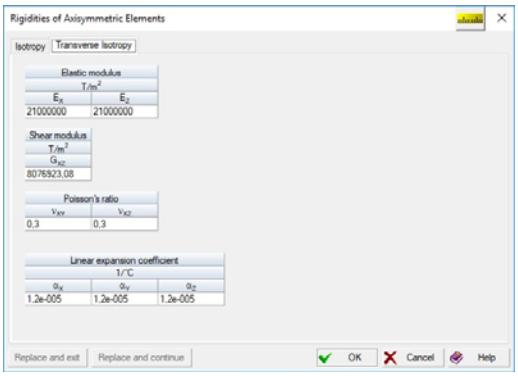


Figure 7.1-20. *The* **Transverse Isotropy** *tab*

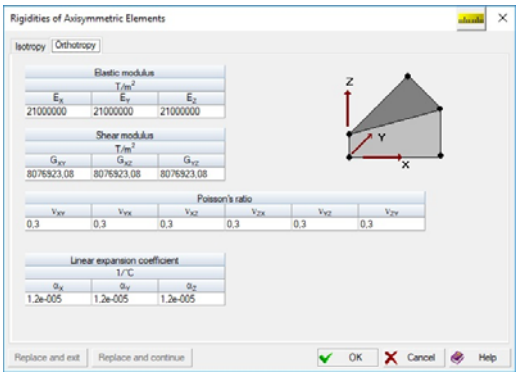


Figure 7.1-21. *The* **Orthotropy** *tab*

7.2. Specifying Stiffness Properties for Special Finite Elements

Stiffness properties are usually specified for special finite elements when they are created. This operation can be used to reassign the stiffness properties of special finite elements created earlier and to specify the respective stiffness properties for the elements after the change of the type of a finite element (for example, after changing the type of a bar element to an element of the elastic constraint).

When specifying the stiffness properties of special finite elements after changing the type of an element the following procedure for performing this operation is recommended:

- invoke the operation, the drop-down list will appear, where you have to select a line with the name of the necessary type of the special element (only the lines containing the names of the elements present in the model will be available in the list);
- the dialog box will appear, where you have to specify the stiffness properties or select them in the **Type of rigidity** list;
- click the **OK** button and assign the rigidity to the elements of the model.

It is necessary to perform the following actions to correct the characteristics of the type of rigidity specified earlier:

- invoke the considered operation;
- select the line with the number of the type of rigidity you want to change in the **Type of rigidity** table of the dialog box;
- correct the characteristics specified earlier;
- click one of the buttons — **Replace and exit** or **Replace and continue**. In the first case the dialog box will close after changing the characteristics, and in the second one you can continue correcting other types of rigidity.

The number of the type of rigidity does not change after performing the above actions.

Color Indication of the Rigidities

Although the color scales are considered in a separate section (see Chapter 4), the brief description of the color scale of rigidities is given here (for reference).

The window of the color scale for the indication of rigidities of the elements is invoked from the **Display Filters** toolbar (see Sec. 10.2) and contains a table with the data and a set of control buttons. The table has seven columns, which contain:

- checkboxes — to enable/disable the color display of the given type of rigidity;
- numbers of the types of rigidity;
- color field showing the color of display of the given type of rigidity in the model;
- designation of the type of element or its section;
- size of the section, thickness of the plate or other parameters which characterize this type of rigidity;
- name of the type of rigidity specified by the user;

− number of elements with the given type of rigidity in the model.

Buttons located at the top of the color scale enable to localize the selection of the types of rigidity displayed in color depending on the type of elements and the method of the rigidity description:

— parametric specification of the sections of bar elements;

— sections from the assortments of rolled steel profiles;

— sections from rolled profiles and sheets created in **Section Builder**;

— numerical and parametric specification of rigidities of bar elements;

— numerical description of rigidities of bar elements;

— plate elements;

— solid elements;

— special finite elements (elastic constraints, constraints of finite rigidity, etc.).

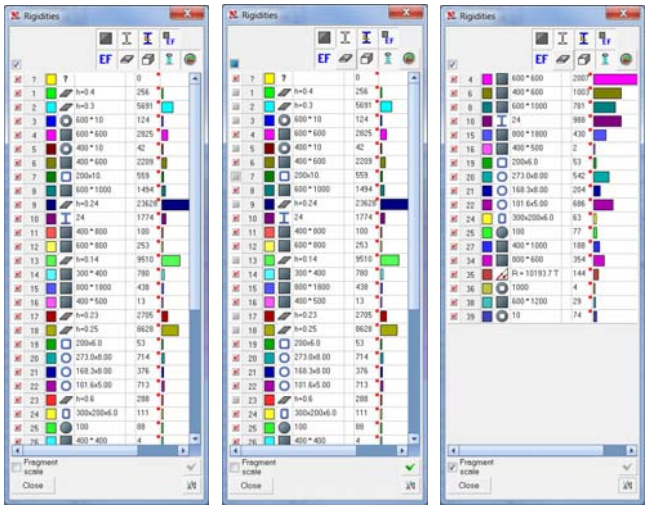
If one or more of these buttons are active, their corresponding numbers (checkboxes) of the types of rigidity will be active in the color scale as well. All buttons are active by default (Fig. 7.2-1, *a*). The button enables or disables all buttons simultaneously. The buttons can be used in any combination.

The checkbox above the checkboxes of the types of rigidity enables to check or uncheck all the checkboxes of the color scale at same time.

Click the **Apply** button — , if you want the elements with the rigidities corresponding to the selected combination of buttons to be displayed in color (Fig. 7.2-1, *b*), and the **Fragment** button— , if you want to obtain a fragment including only these elements.

If the current window displays not the whole model but only its part (the fragmentation of the design model is performed), the set of the types of rigidity in the color scale can be limited only by the numbers included in the current fragment. The **Fragment scale** checkbox is used to switch between the scales (Fig. 7.2-1, *c*).

The button Close is used to finish the work with the color scale.



a) *b*) *c*)

Figure 7.2-1. *The* **Rigidities** *color scale*

Delete Duplicate Types of Rigidity

"Duplicate types of rigidity" may appear when working with the model, i.e. with the same parameters and different only by numbers. After performing this operation all elements that had the same parameters are automatically assigned a common number of the type of rigidity.

Figure 7.2-2. *The operation of deleting the duplicate types of rigidity*

If the duplicate types of rigidity have been given different names, the result of performing the operation will depend on the response to the request in the message window (Fig. 7.2-2). If the answer is positive, they will be reduced to one (minimum) number with its respective name. Otherwise, the compression is not performed.

Duplicate types of rigidity are deleted together with the duplicate types of the subsoil parameters.

Renumber Rigidities

This operation enables to sort/renumber the rigidities in the design model.

When this operation is invoked, the **Renumber Rigidities** dialog box appears (see Fig. 7.2-3) with a list of rigidities. The table has the following columns:

the current number of the rigidity;

- **No**. — number of the rigidity in the order of creating;
- **Type** — icon of the rigidity type;
- **Name** — name of the rigidity;
- **Description** — description of the rigidity;
- **Number of elements**, which have the respective stiffness properties.

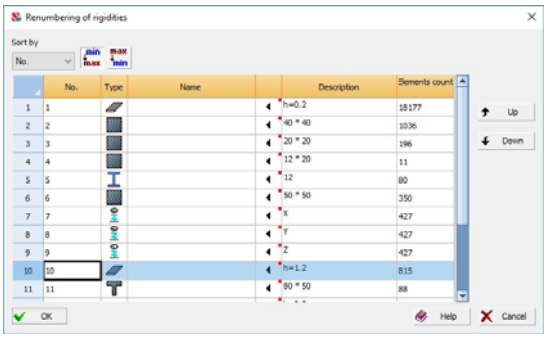


Figure 7.2-3. *The* **Renumber Rigidities** *dialog box*

The characteristics of the rigidity the sorting is based on is selected from the **Sort by list**. There are the following types of sorting:

- by the number (in the order of creating rigidities — when the **Min/Max** button is pressed; and vice versa — when the **Max/Min** button is pressed);
- by the type of the rigidity;
- by the name of the rigidity (in the alphabetical order when the **Min/Max** button is pressed; and vice versa — when the **Max/Min** button is pressed);
- by the description of the rigidity (ascending — when the **Min/Max** button is pressed; and descending — when the **Max/Min** button is pressed);
- by the number of elements (ascending — when the **Min/Max** button is pressed; and descending — when the **Max/Min** button is pressed).

In order to change the position of one or several rigidities in the list you have select these rigidities and move them in the list by clicking the **Up** or **Down** buttons.

Moreover, this dialog box enables to rename the types of rigidity, or you can specify that the description of the rigidity can be used as the name with the help of the button ◀.

All the specifications performed in the dialog box will be saved and executed after closing the dialog box by clicking the **OK** button. The position of rigidities will correspond to their position in the list of the dialog box.

Physical and Mechanical Properties of the Material

When specifying the stiffness properties such properties of materials as the elastic modulus, specific weight, Poisson's ratio can be assigned automatically. The **Material** section of the dialog boxes (Fig. 7.2-5) contains a drop-down list with the list of materials (Fig. 7.2-4). If you select the necessary material, all the data will be taken from the database supplied with the software. The program displays the properties of the selected material for control (elastic modulus, Poisson's ratio, …).

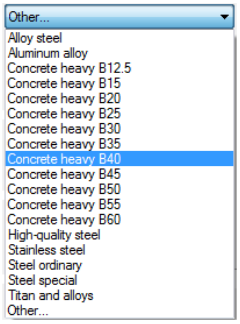


Figure 7.2-4. *The* **Materials** *drop-down list*

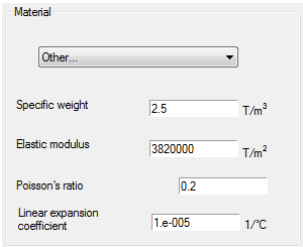


Figure 7.2-5. *The* **Material** *section*

You can add, delete or change the data from the database of materials with the help of **Materials Editor**, which is included in the **SCAD Office** package. If you select the last item in the list "Other...", you will be able to specify the data on the mechanical properties of the necessary material not included in the database in the respective text fields. The software will obviously check the input information before the calculation (for example, you cannot specify the Poisson's ratio greater than 0.5, …).

7.3. Specifying the Properties of Elements

Specifying the Element Type

This operation enables to assign or change the types of finite elements. The **Specify the Element Type** dialog box (Fig. 7.3-1) is used to assign the element type. It is invoked by clicking the respective button in the **Assign** section of the toolbar (see Fig. 7-1).

You can perform the following actions in this dialog box:

− select the radio button with the name of the respective element (bar, shell, etc.);

- select the necessary element type in the list;
- click the **OK** button in the dialog box;
- select the elements in the model the selected type is assigned to, and assign it by clicking the **OK** button in the toolbar.

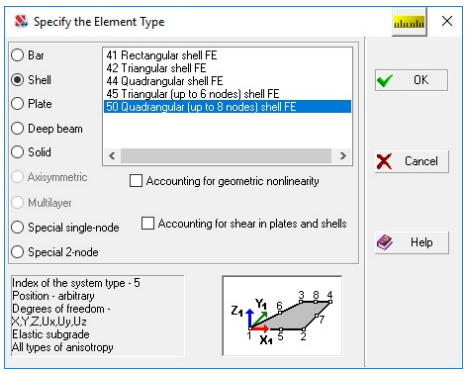


Figure 7.3-1. *The* **Specify the Element Type** *dialog box*

When the **Accounting for geometric nonlinearity** checkbox is checked, the selected elements will be assigned a type which enables to perform the analysis taking into account large displacements.

When assigning a type to shell , deep beam and plate elements you can check the **Accounting for shear in plates and shells** checkbox. Instead of the specified type of element its corresponding type will be assigned (the type number is increased by 100), which enables to allow for shear according to the Mindlin–Reissner theory.

Before assigning the element type look through the short description which is displayed in the information field of the dialog box after you select an element from the list, and on the icon with the image of the local axes of the element.

The element type must correspond to the specified type of the design model. The correct choice of the element type will help to avoid mistakes when solving the problem.

The specified type will be assigned only to the elements with the number of nodes corresponding to the description of the element in Chapter 3.

Specifying Nonlinear Elements

Invoking this operation drops down a menu where you can select type of nonlinear behaviour (geometric or physical nonlinearity). After that select the elements in the model and click the **OK** button in the toolbar. The selected elements are used to allow for large displacements or physical nonlinearity when performing the nonlinear analysis.

Numbers of element types will be formed by adding 300 (in the case of geometric nonlinearity) or 400 (in the case of physical nonlinearity) to the number of type of a linear element. For example, the element of type 44 will be transformed into 344 (or 444) and so on.

Install or Remove Rigid Inserts

This operation is used to install rigid inserts in bar elements. Rigid inserts can be described in two ways: in the global coordinate system of the design model or in the local coordinate system of the element. The **Rigid Inserts** dialog box (Fig. 7.3-2) is used to specify the characteristics of rigid inserts, where you have to specify the method of describing inserts and their lengths.

If you are working in the local coordinate system, the local X_2 axis of the element goes from node 1 to node 2, and the lengths of rigid inserts are specified as the lengths of projections on the respective local axes. If the rigid insert is given in local axes and the value of length of the projection along the X_1 axis is not equal to zero, and zero values are specified along the Y_1 and Z_1 axes, the length of the projection along the X_1 axis (in fact, the length of the rigid insert) must be defined by a positive number. This rule is accepted, because the imposition of a rigid insert on the flexible part of the bar taking place in this case is physically meaningless. When inputting the data in the global coordinate system the software controls the

fact of imposing a rigid insert on the flexible part in the process of calculation and automatically changes the direction of the insert.

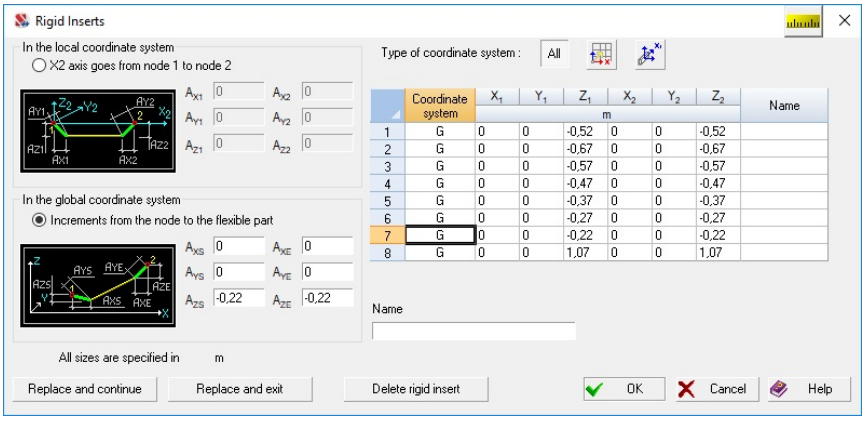


Figure 7.3-2. *The* **Rigid Inserts** *dialog box*

If you are working in the global coordinate system, the lengths of rigid inserts are specified as projections on the axes of the global coordinate system, i.e. as vectors directed from the node of the flexible part of the bar. It is necessary to perform the following actions to create rigid inserts:

- invoke the **Install-remove rigid inserts** operation;
- select the method of describing an insert using the radio buttons — in the local coordinate system of the bar or in the global coordinate system;
- specify the name of the rigid insert if necessary;
- specify the data in the respective text fields and click the **OK** button in the dialog box;
- select the elements in the model the rigid inserts are installed in, and click the **OK** button in the toolbar.

It is necessary to perform the following actions to correct the characteristics of rigid inserts created earlier:

- invoke the considered operation;
- select a line with the parameters of the inserts you want to correct in the table of the **Rigid Inserts** dialog box, and the selected parameters will be input into the respective text fields;
- correct the parameters;
- click one of the buttons — **Replace and exit** or **Replace and continue**. In the first case the dialog box will close after changing the characteristics, and in the second one you can continue correcting other rigid inserts.

Buttons located at the top enable to perform the filtration of the data specified earlier and leave the table lines referring to the data only in the local or only in the global coordinate systems.

Keep in mind that when the display filter of rigid inserts is disabled, a line connecting the nodes is displayed in the model, but not the element. If there are rigid inserts, this line might not reflect the actual position of the element in the model.

Move the Plate Midplane

This operation enables to change the position of plate elements with respect to their snap nodes by moving the midplane along the local Z_1 axis. The value of displacement is specified in the dialog box (Fig. 7.3-3, *a*), which appears after invoking the operation.

In order to perform the operation after specifying the displacement value it is necessary to select the elements you want to move in the model and click the **OK** button in the **Assign** toolbar.

The displacement values specified earlier can be corrected by transferring the data from the selected table line into the text fields with their subsequent correction and fixation with the **Replace and exit** or **Replace and**

continue buttons. The operation can be used for example to align the surface of the slab modeled by finite elements of different thickness along the upper or lower face and in other cases (Fig. 7.3-3, *b*).

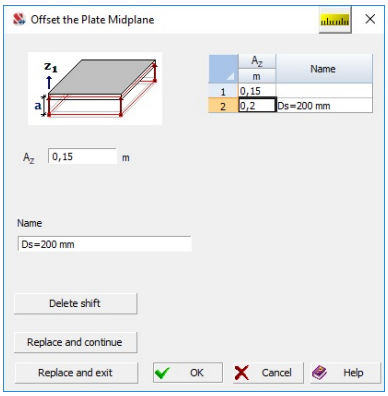


Figure 7.3-3, *a*. *The* **Move the Plate Midplane** *dialog box*

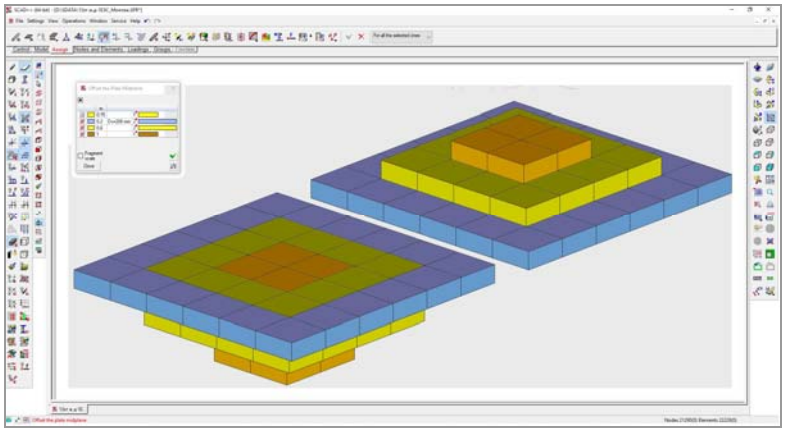


Figure 7.3-3, *b*. The example of using the **Move the plate midplane** operation

Install and Delete Hinges and Sliders

Boundary conditions can be assigned for bar elements as a freedom of mutual rotation about the axes of the local coordinate system (cylindrical hinges) or a freedom of mutual linear displacements along these axes (sliders).

It is assumed by default that such independent displacements are impossible due to the constraints between the element and the node.

Boundary conditions in each node can be specified in the **Boundary Conditions of Bars** dialog box (Fig. 7.3-4) with the help of the respective buttons. This dialog box is also used when it is necessary to change the boundary conditions specified earlier (but not to cancel all of them).

In order to perform this operation you have to:

- click the respective button in the toolbar;
- the dialog box will appear where you have to specify the boundary conditions: to release the degree of freedom completely (a perfect hinge), set the compliance to 0;
- select the places where the hinges should be installed (**near the node** in the coordinate system $X_1Y_1Z_1$ or **at the transition to the flexible part** in the coordinate system $X_2Y_2Z_2$ presented on the figure 3.3-12);
- click the **OK** button in the dialog box;
- select the necessary elements in the model;
- click the **OK** button in the toolbar.

A special operation **Delete hinges** is used to cancel the specified boundary conditions for all directions. It invokes the **Delete Hinges** dialog box (Fig. 7.3-5). The concept of a “hinge” is generalized and is used for both actual hinges and sliders.

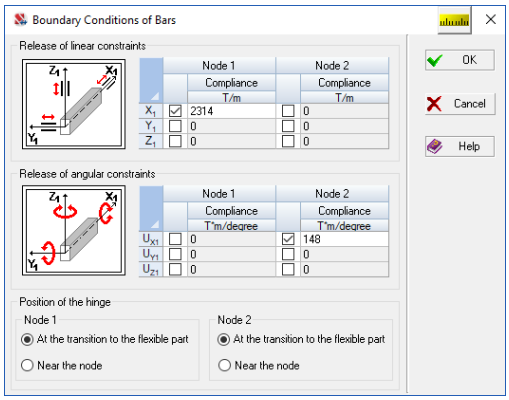


Figure 7.3-4. *The* **Boundary Conditions of Bars** *dialog box*

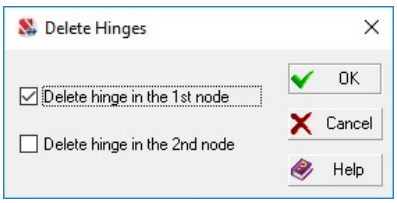


Figure 7.3-5. *The* **Delete Hinges** *dialog box*

Depending on the active buttons the deleting can be performed in one or in both nodes of the bar.

The actions of deleting are similar to the actions of assigning — select elements and click the **OK** button in the toolbar.

The filters button is used to display the hinges in the model.

Specifying the Orientation Angle of the Axes of the Bar Section

This operation is used to:

- change the orientation of the principal or structural axes of the bar section;
- specify the local coordinate system for a two-node elastic constraint of non-zero length.

The detailed information about the local coordinate system of the bar elements is given in the respective section of the "Finite Element Library" chapter. Orientation can be defined by indicating the rotation angle of the axes, specifying the coordinates of the point with respect to which the local Z1 or Y1 axes of the selected elements are rotated (depending on the selected option) or by moving with respect to the first node. The operation allows you to orient both the principal axes of inertia and the structural axes.

The control of the operation is performed with the help of the **Orientation of the Axes of Inertia** dialog box (Fig. 7.3-6).

It is necessary to select the method for specifying the angles in the dialog box (rotation angle of axes is selected by default) and specify the necessary data. All the other actions are familiar — click the **OK** button in the dialog box, select elements and confirm the operation by clicking the **OK** button in the toolbar.

In order to restore the values of angles assumed by default it is necessary to repeat the operation of specification specifying the angle of 0°.

The values of the rotation angles of the axes for the given element can be corrected in the **Local axes** mode in the **Information on the Element** dialog box. The dialog box is invoked by clicking the button in the **Display Filters** toolbar.

It is necessary to perform the following actions to correct the characteristics of the angles specified earlier:

- invoke the considered operation;
- select a line with the parameters you want to correct in the table of the **Orientation of the Axes of Inertia** dialog box, and the selected parameters will be input into the respective text fields;
- correct the parameters;
- click one of the buttons — **Replace and exit** or **Replace and continue**. In the first case the dialog box will close after changing the characteristics, and in the second one you can continue correcting other angles of orientation.

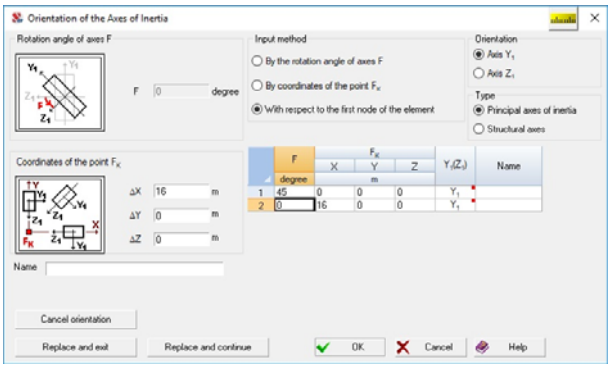


Figure 7.3-6. *The* **Orientation of the Axes of Inertia** *dialog box*

The **Cancel orientation** button enables to restore the initial position of the structural axes of elements. After closing the dialog box by clicking this button select the elements in the model, the initial position of the structural axes of which you want to restore and click the **OK** button in the toolbar.

Specifying Intermediate Sections for the Calculation of Forces

This operation is used to calculate forces in the intermediate points along the length of the bar or in the nodes of plate and solid elements (it should be noted that the forces are calculated by default only in the beginning, middle and end of the bar and in the center of the plate). It is necessary to know the forces in the intermediate sections of the bars, for example, to perform the selection of the reinforcement.

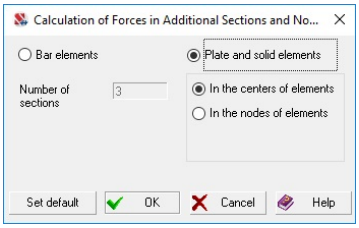


Figure 7.3-7. *The* **Calculation of Forces in Additional Sections and Nodes** *dialog box*

The control of the operation is performed in the **Calculation of Forces in Additional Sections and Nodes** dialog box (Fig. 7.3-7).

It is necessary to select the elements (bars or plates), and specify the parameters in the dialog box: for bars — number of sections (general, including the sections in the beginning and in the end of the bar), for plates and solid elements — type of the output information. All the other actions are standard — click the **OK** button in the dialog box, select the elements, click the **OK** button in the toolbar.

In order to change the parameters of specification it is necessary to repeat the specification with the new data.

If in the bar elements it is necessary to specify the number of sections of the output of forces determined by default, i.e. specified in the **Elements** tab of the **Calculation Parameters** dialog box (see Sec. 11.9), you have to click the **Default** button in the considered dialog box before clicking the **OK** button, and then select the necessary elements and perform the operation.

Change the Direction of the Local X_1 Axis of Bars to the Opposite One

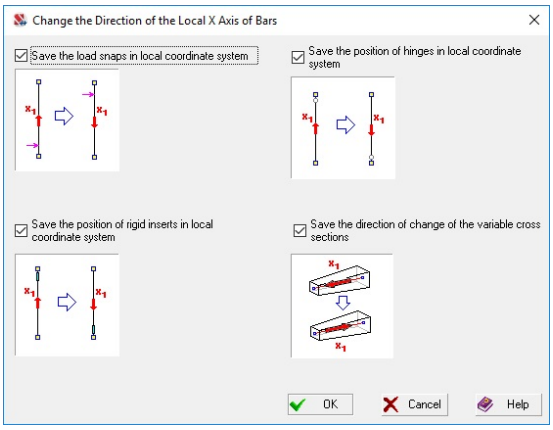


Figure 7.3-8. *The* **Change the Direction of the Local X Axis of Bars** *dialog box*

This operation enables to change the direction of the local X_1 axis of the bar to the opposite one, i.e. basically to swap the first and the second nodes of the element.

If a concentrated load is acting on the bar, it will also change its position in accordance with the direction of the local axis. A similar problem may occur with hinges and rigid inserts. Keep in mind that if the load on the bar is specified in the local coordinate system, it will also change its direction in accordance with the direction of the local axis. Therefore, when this operation is invoked, the dialog box appears (Fig. 7.3-8), where you can specify options which enable to automatically change the snaps of loads, position of hinges and rigid inserts. Having clicked the **OK** button in the dialog box, you have to select the necessary bar elements in the model and click the button in the toolbar.

In the case of bars of variable cross-section you can save or reverse the direction of the section variation.

Change the Direction of the Local Z_1 Axis of Plate Elements

This operation enables to change the direction of the local Z_1 axis of plate elements to the opposite one.

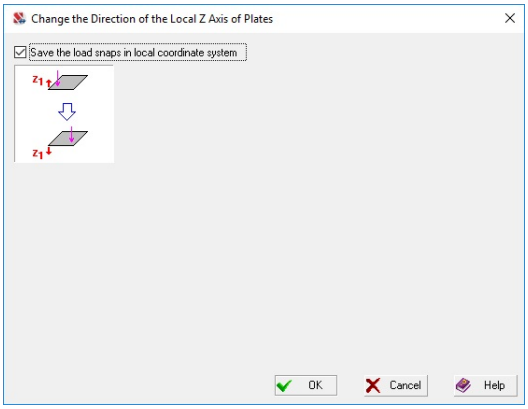


Figure 7.3-9. *The* **Change the Direction of the Local Z Axis of Plates** *dialog box*

If a concentrated load is acting on the plate, it will also change its position in accordance with the direction of the local axis. Keep in mind that if the load on the bar is specified in the local coordinate system, it will also change its direction in accordance with the direction of the local axis. Therefore, when this operation is invoked, the dialog box appears (Fig. 7.3-9), where you can specify the option which enables to automatically change the snaps of loads. Having clicked the **OK** button in the dialog box, you have to select the necessary plate elements in the model and click the button in the toolbar. In order to select elements you can use the color or vector indication of the direction of axes with the help of the filters button.

Stresses along the Given Direction

The operation enables to specify the direction of force and stress output in plate elements different from the default one — along the axes of the local coordinate system of the element. This is important when analyzing the results of calculation of force factors given in the form of isolines and isofields, since the disorder of the directions of local axes of the elements with irregular meshes (for example, after meshing) does not allow to plot the isolines and isofields of force factors correctly (they simply lose their meaning). Moreover, when selecting the reinforcement it is recommended to agree the directions of the stress output with the orientation of reinforcement meshes.

Clicking the button invokes the menu, where you can select two variants: user defined directions or the automatic specification (Fig. 7.3-10).

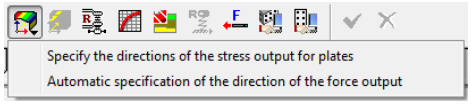


Figure 7.3-10

In the first case the operation enables to specify the following data for plates:

- A. direction of the X_N axis of the stress analysis which is defined by a line going from the first node of the element to the point specified by the increments of the coordinates of this node in the global coordinate system. The obtained line is projected onto the plane of the element;
- B. a point specified by the coordinates in the global coordinate system; the projection of the line connecting this point with the first node of the element onto the plane of the element defines the X_N axis of the stress analysis;
- C. direction of the X_N axis of the stress analysis coinciding with the direction of the given vector;
- D. direction of the X_N axis of the stress analysis by an explicit indication of the axis of the global coordinate system.

If the automatic specification is selected, a direction along the global X axis is specified for all horizontal plates, and along the global Z axis — for all vertical ones.

The Z_N axis will coincide with the Z_1 axis of the local coordinate system of the element, and the Y_N axis will lie in the plane of the element and will form a right-hand system together with the X_N and Z_N axes.

The axes of the analysis of stresses in plate elements are specified in the **Specify Axes of the Stress Analysis** dialog box (Fig. 7.3-11).

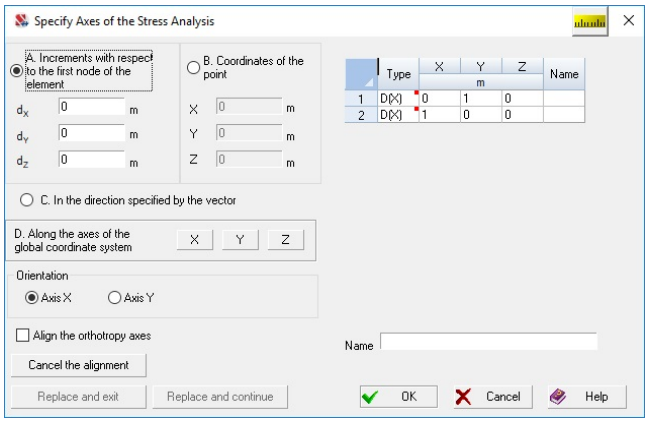


Figure 7.3-11. *The* **Specify Axes of the Stress Analysis** *dialog box*

In order to perform this operation click the button invoking the operation in the toolbar and select the method of specifying axes in the menu. If the automatic specification is selected, the operation will be performed after the selection of this item in the menu. Otherwise, the **Specify Axes of the Stress Analysis** dialog box will appear, where you can select the rule for specification of the force output axes using the respective radio buttons.

The following actions are performed depending on the selected variant:

- if variant A or B is selected, the increments or the coordinates of the point are specified respectively;
- if variant D is selected, the marker button with the name of the axis of the global coordinate system defining the direction of X_N is activated.

After closing the dialog box by clicking the **OK** button you have to select the necessary elements in the model and click the **OK** button in the **Assign** section of the toolbar.

If variant C is selected as the direction of the stress output, it is recommended to perform the operation in the following order:

- close the dialog box by clicking the **OK** button;
- select the elements in the model the operation is performed for;

- click the **OK** button in the toolbar, the message window will appear (Fig. 7.3-12) with a suggestion to select two nodes in the model which define a vector;
- select two nodes in the model with a cursor with a target (the vector will be directed from the first node to the second one);
- click the **OK** button in the toolbar again.

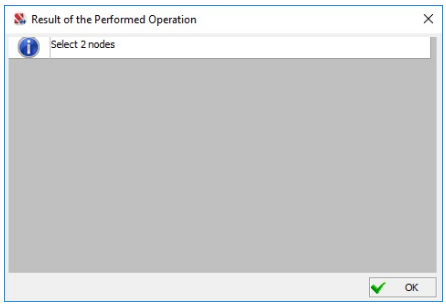


Figure 7.3-12. *The message window*

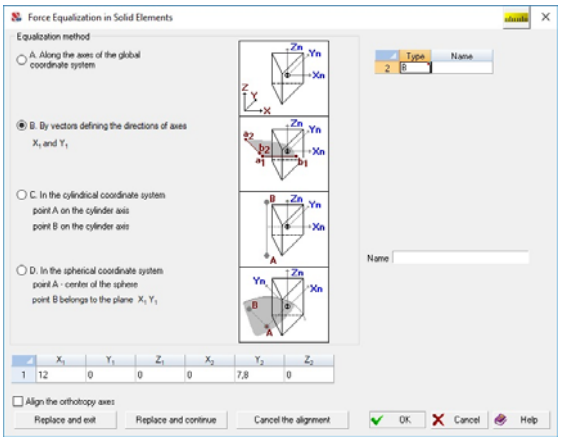


Figure 7.3-13. *The* **Force Equalization in Solid Elements** *dialog box*

In order to cancel the performed specifications (the direction of the force output will coincide with the axes of the local coordinate system of the elements) it is necessary to activate the mode of the specification of axes and click the **Cancel the alignment** button in the dialog box. After closing the dialog box select the necessary elements in the model and click the **OK** button.

You can also select the option in the dialog box when the direction of the Y_N axis will be specified instead of that of the X_N axis. In order to do it select the **Y axis** option in the **Orientation** group.

The direction of the stress output in solid elements is specified in a similar way — . The data are specified in the dialog box shown in Fig. 7.3-13.

The following variants of the specification of the stress output axes are provided in the software for solid elements:

- along the axes of the global coordinate system;
- by a vector defining the direction of the force output axes;
- in the cylindrical coordinate system;
- in the spherical coordinate system;
- in the local coordinate system of the element.

Coordinates of two points lying on the axis of the cylinder and defining the direction of the Z_N axis are specified in the cylindrical coordinate system. The X_N axis is orthogonal to the Z_N axis and passes through the center of gravity of the element. The Y_N axis is orthogonal to the X_N and Z_N axes. Coordinates of the center of the sphere (X_1, Y_1, Z_1) and additional points (X_2, Y_2, Z_2) are specified in the spherical coordinate system. The X_N axis goes from the center of the sphere through the center of gravity of the element (X_3, Y_3, Z_3). The Y_N axis lies in the plane defined by the three given points, it is orthogonal to the X_N axis and is directed in the direction of the point X_2, Y_2, Z_2. The Z_N axis is orthogonal to the X_N and Y_N axes.

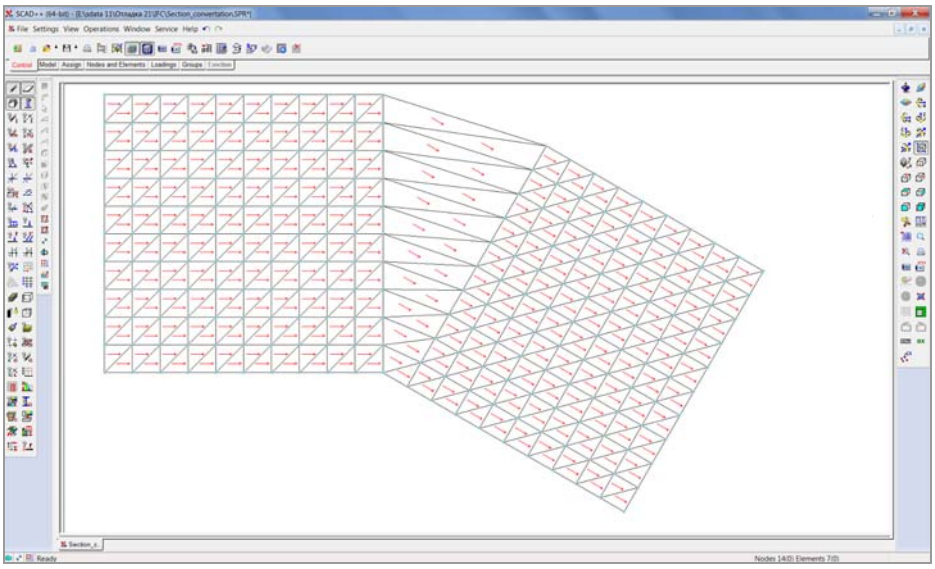


Figure 7.3-14. *Example of a design model*

Fig. 7.3-14 shows an example of a design model for the left part of which the X axes of the force output are specified along the direction of the X axis of the global coordinate system, and for the right one (rotated part of the model) the axes are defined by the increments with respect to the first node of the element (1, –0.577, 0), which corresponds to an angle of 30°.

The display of the direction of axes of the calculation of forces in the design model is enabled/disabled by clicking the filters button .

It is necessary to perform the following actions to correct the directions of the force and stress output specified earlier:

- invoke the considered operation;
- select a line with the parameters you want to correct in the table of the dialog box, and the selected parameters will be input into the respective text fields;
- correct the parameters;
- click one of the buttons — **Replace and exit** or **Replace and continue**. In the first case the dialog box will close after changing the characteristics, and in the second one you can continue correcting the directions.

Specifying the Direction of the Orthotropy Axes

This operation enables to specify the direction of the orthotropy axis X_1 for the selected elements according to the same rules as for the direction of the force and stress output axes. Therefore the dialog boxes of the specification of axes of the force and stress output for plate and solid elements have the **Align the orthotropy axes** checkbox, which enables to indicate that the specifications performed in the dialog boxes relate to the orthotropy axes.

Specifying Subsoil Parameters

Subsoil parameters are considered as a separate type of data and are not included in the description of stiffness properties of elements. Thus, elements with the same stiffness properties can have one type of rigidity irrespective of the subsoil parameter specified for them.

When this operation is invoked, the menu appears (Fig. 7.3-15), where you have to select the elements (bars or plates) the subsoil parameters will be assigned for.

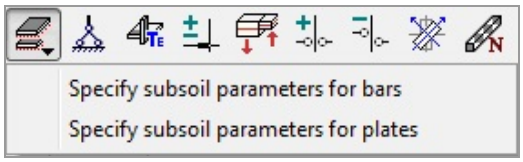


Figure 7.3-15. *Selecting the elements*

If there are only plates in the design model, the menu does not appear and the dialog box for the input of data appears at once.

The values of subsoil parameters for plate and bar elements are specified in the dialog boxes shown in figures 7.3-16 and 7.3-17 respectively. The operation of assigning the subsoil parameters to the selected elements of the model is performed by clicking the **OK** button in the toolbar.

You can use **Cross** to determine the subsoil parameters $\mathbf{C_1}$ (Winkler model) under the foundation slab. The **Calculation of subsoil parameters** button — in the **Assign** toolbar enables to invoke this application. The **Calculation of subsoil parameters** or **Calculation of subsoil strain factors** buttons in the dialog boxes can be used to calculate the subsoil parameters $\mathbf{C_1}$ and $\mathbf{C_2}$. The Pasternak model is used for calculating subsoil parameters, and the model given in the work of V.G.Piskunov and Y.M.Fedorenko [1] — for determining the subsoil strain factors. Calculations are performed both for plate and for bar elements.

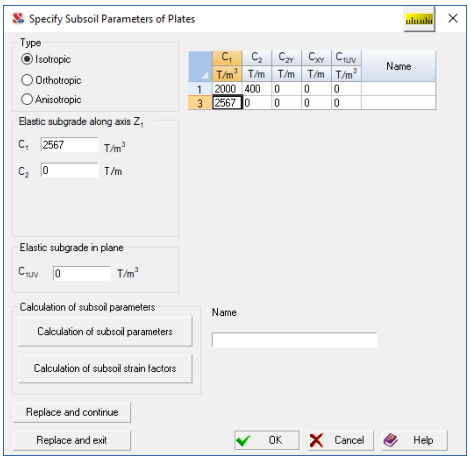


Figure 7.3-16. *The* **Specify Subsoil Parameters of Plates** *dialog box*

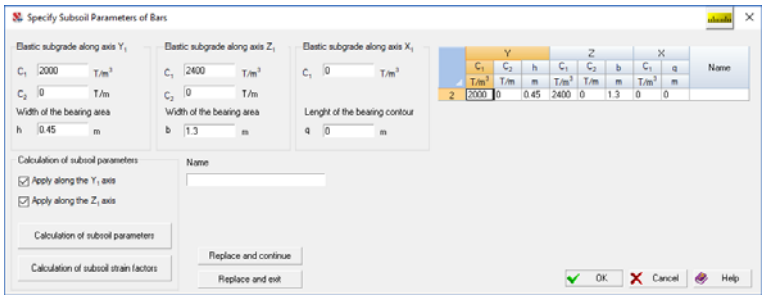


Figure 7.3-17. *The* **Specify Subsoil Parameters of Bars** *dialog box*

The obtained values of the subsoil parameters are automatically transferred into the respective text fields. For bar elements this transfer is performed depending on the checked **Apply along axis ...** checkboxes (Fig. 7.3-17).

The values of the subsoil parameters are corrected according to the same rules as other parameters, for example, rigid inserts or the angles of orientation of the bar axes, i.e. by transferring the data from the selected table line into the text fields with their subsequent correction and fixation with the **Replace and exit** or **Replace and continue** buttons.

The **Information on the Element** dialog box has the **Subsoil parameters** button, which enables to control the values of the subsoil parameters. Clicking it invokes the **Specify Subsoil Parameters** dialog box (Fig. 7.3-16, 7.3-17). In order to change a parameter related to a particular element in the processor you have to specify its new value and close the dialog box by clicking the **OK** button.

Specifying the Prestressing of Elements of the Design Model

The operation enables to specify the local (internal) prestressing for bar, cable-stayed, plate and solid elements. The initial data for the operation are specified in the **Prestressing** dialog box (Fig. 7.3-18), where you have to use the respective radio buttons to select the elements the prestressing is specified for,

and to indicate the set of force factors which will be used for the specification. If the **Extended mode** checkbox is not checked, the parameters of prestressing will be specified using a reduced set of force factors. Otherwise, the maximum set of factors will be used for this type of elements. For example, a reduced set of factors for bar and cable-stayed elements includes the specification only of a longitudinal force, and the extended one — the specification of eight factors including a longitudinal force, torque, two bending moments, two shear forces, and two values of reactions. The values of force factors are specified in the table, the columns of which depend on the selected type of elements. Each set of parameters of prestressing can have its own name specified in the respective field.

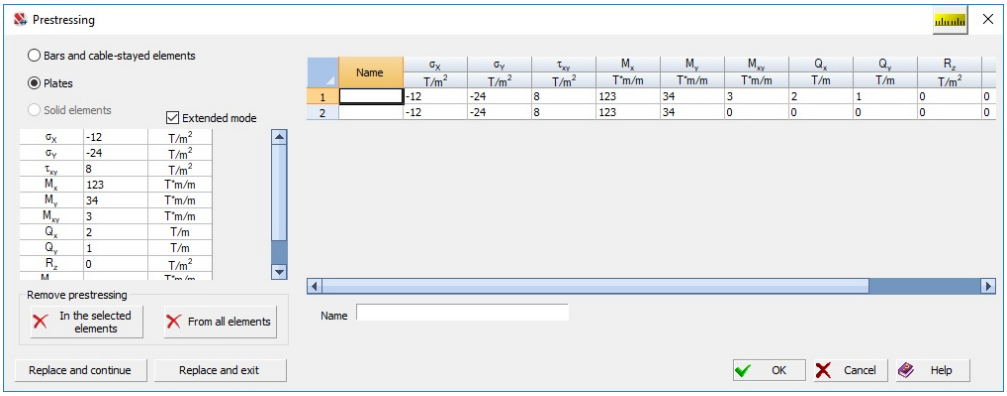


Figure 7.3-18. *The* **Prestressing** *dialog box*

In order to specify the prestressing for the elements of the model you have to specify the parameters in the dialog box, close the dialog box by clicking the **OK** button, select the elements in the model and click the **OK** button in the **Assign** toolbar.

The values of prestressing are corrected by altering the table and saving these changes by clicking the **Replace and exit** or **Replace and continue** buttons.

In order to delete the prestressing from the elements it is necessary to select the line with the parameters you want to delete in the table of the **Prestressing** dialog box and select the rule for deleting using the respective buttons — **From all elements** or only **In the selected elements**, and then close the dialog box by clicking the **OK** button. If the first rule is selected, the values of prestressing specified earlier will be deleted from all the elements where they were specified. In the second case the values will be deleted from the elements selected in the model after clicking the **OK** button in the toolbar.

The **Information on the Element** dialog box has the respective button which enables to control the values of the parameters of prestressing. The **Prestressing** button — in the **Display Filters** toolbar enables to obtain both digital and color indication of the specified parameters.

Specification of the Additional Data for Elements

This operation enables to assign the elements some properties which are used quite rarely. The following properties can be specified in the current version of the software:

- limit relative inelastic deformation μ according to KMK 2.01.03-96, KMK 2.01.03-19 (Uzbekistan);
- meshing density of the bar section at the physically nonlinear analysis;
- number of layers along the plate thickness at the physically nonlinear analysis;
- boost factor f_{vk} of the floors according to SP RK 2.03-30-2017 (Kazakhstan).

Clicking this button drops down a menu, where you have to select a parameter which will be specified. Then a dialog box will appear (Fig. 7.3-19), where you can specify the name and the value of the characteristic.

The further procedure is identical to that described for the specification of other properties (rigidities, subsoil parameters, ...).

The color scale of the additional properties can be enabled/disabled by clicking the filters button .

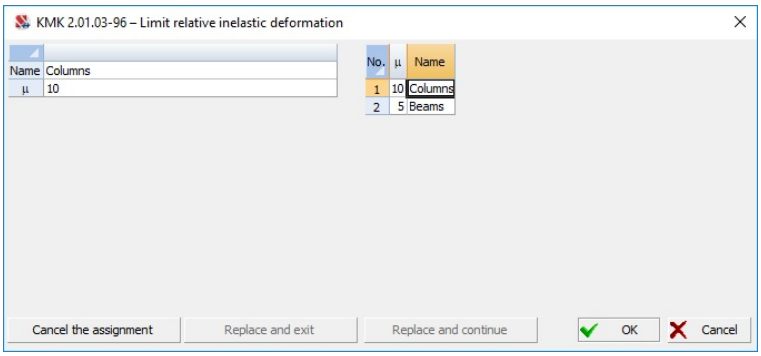


Fig. 7.3-19. Dialog box for the specification of the additional data

7.4. Specifying Characteristics of Nodes

Installing Constraints in Nodes

Constraints in nodes of the design model are specified in the dialog box (Fig. 7.4-1), which is invoked by clicking the respective button in the **Assign** section of the toolbar.

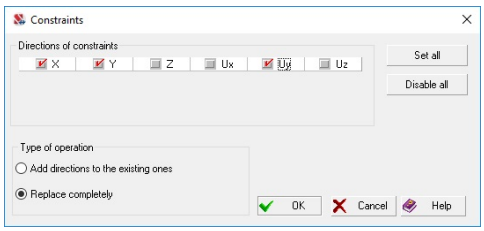


Figure 7.4-1. *The* **Constraints** *dialog box*

In order to install constraints in the node it is necessary to select the directions of degrees of freedom on which the constraints are imposed using the respective checkboxes, click the **OK** button in the dialog box, select the nodes in the model and perform the operation by clicking the **OK** button in the **Assign** section.

Depending on the type of operation selected in the dialog box the constraints in nodes will be replaced completely (deleted if all the checkboxes of directions are not checked) or added to those specified earlier.

The **Constraints** dialog box is different for design models of type 8, 9 and 11 and depends on the number of the degrees of freedom in nodes of the respective type of the model (12 degrees for type 8, 9 and two degrees for type 11). Fig. 7.4-2 shows the **Constraints** dialog box for the models of types 8 and 9, and figure 7.4-3 — for the model of type 11.

You can use the filters button to display the constraints in the model.

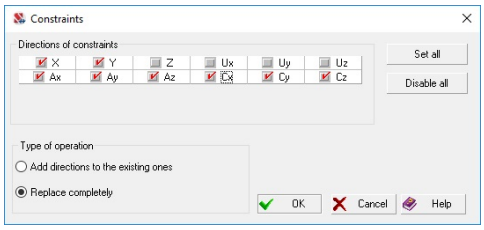


Figure 7.4-2. *The* **Constraints** *dialog box*
for structures from multilayer shells

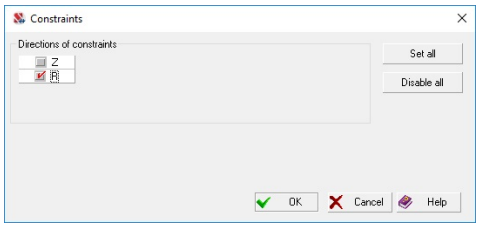


Figure 7.4-3. *The* **Constraints** *dialog box*
for an axisymmetric problem

Merged Displacements

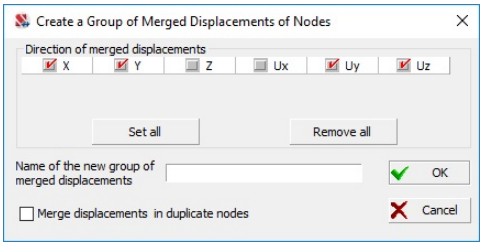


Figure 7.4-4. *The* **Create a Group of Merged Displacements** *dialog box*

If the groups of merged displacements have not been created yet, clicking the button invokes the **Create a Group of Merged Displacements** dialog box (Fig. 7.4-4), where you can specify the directions of the degrees of freedom in which the displacements of the group are merged and specify the names of the groups.

In order to perform the operation it is necessary to select the directions of degrees of freedom in which the displacements are merged using the respective checkboxes, specify the name of the group (the groups should have unique names), confirm the specified data by clicking the **OK** button in the dialog box, select the nodes in the model which are included in the group and perform the operation by clicking the **OK** button in the toolbar. If the **Merge displacements in duplicate nodes** checkbox is checked, the displacements are merged for the specified directions for all selected pairs of nodes.

If the design model already has groups of merged displacements created earlier, clicking the button drops down the menu where you can select one of the following actions:

- **Create a group of merged displacements** (described above);
- **Control the groups of merged displacements**.

In the second case the dialog box is invoked (Fig. 7.4-5), which is used to control the attributes of groups and enables to perform the following actions:

- rename groups;
- sort groups by a certain criterion;
- delete groups;
- change the place of the selected group in the list of groups (**Up**, **Down** buttons);
- change the set of the degrees of freedom.

The columns of the table of groups include:

- the current number of the group;
- **No.** — number of the group in the order of creating;
- **Name** — name of the structural group;
- **Number of nodes** in the group;
- list of the **degrees of freedom**.

In order to rename a group place the cursor over the name of the respective group and double left click. The table line will become available for editing.

The characteristics of the group the sorting is based on is selected from the **Sort by** list. There are the following types of sorting:

- by the number (in the order of creating groups — when the **Min/Max** button is pressed; and vice versa — when the **Max/Min** button is pressed);
- by the name of the group (in the alphabetical order when the **Min/Max** button is pressed; and vice versa — when the **Max/Min** button is pressed);
- by the number of nodes in the group (ascending — when the **Min/Max** button is pressed; and descending — when the **Max/Min** button is pressed).

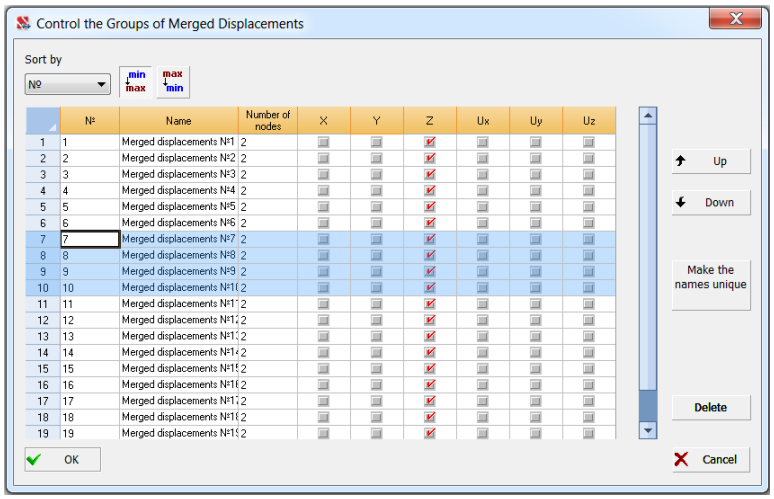


Figure 7.4-5. *The* **Control the Groups of Merged Displacements** *dialog box*

In order to change the position of one or several groups in the list you have to select these groups and move them in the list by clicking the **Up** or **Down** buttons.

If when specifying the data the user has created several groups with the same names, the **Make the names unique** operation enables to rename them automatically (by adding the suffix № XXX to the name) and obtain unique names.

You can delete the groups of merged displacements selected in the list with the help of the respective button.

All the specifications performed in the dialog box will be saved and executed after closing the dialog box by clicking the **OK** button. The position of groups in the list of the toolbar will correspond to their position in the list of the dialog box.

You can use the filters button to display the groups in the model.

7.5. Specifying Subsoil Parameters for the Elements of the Design Model with the Help of Cross

Cross enables to calculate the subsoil parameters under the foundation slab using the results of geological surveys represented as characteristics of boreholes. The initial data for the calculation include the information on the construction site including its overall dimensions, locations of boreholes and their characteristics, location and geometry of existing buildings and structures on the site, and other data provided in **Cross** [2]. This information must be prepared in advance in **Cross** and saved as a file with the extension **.crs**. After this every time you invoke **Cross** you can use this information by opening the respective file.

In order to determine the subsoil parameters it is usually necessary to perform several iterations using the values of pressure on soil under the foundation slab obtained in the previous step for each subsequent iteration. The value of the reaction of soil $\mathbf{R_z}$ is used as the pressure on soil. Before performing the first calculation the subsoil parameters (they are usually the same for all elements of the slab) are specified by the user. If the construction site is prepared, it is necessary to perform the following actions to establish the connection with **Cross**:

- click the **Calculation of subsoil parameters** button in the **Assign** toolbar;
- select a fragment of the design model the subsoil parameters are calculated for (elements included in the fragment must lie in one plane and form a united area);
- click the button in the toolbar;
- if the current results of the calculation include the values $\mathbf{R_z}$, the **Select Loading** dialog box appears (Fig. 7.5-1). It has a drop-down list, where you have to select the number of the loading or the combination of loadings defining $\mathbf{R_z}$ (or use the respective button to reject using the values of reactions);

– the **Select Construction Site** dialog box (Fig. 7.5-2) will appear, which enables to:
  - load one or more files with the extension **.crs** containing the prepared data on the construction site. In order to do it, click the **Add** button and select the respective files in the standard **Open** dialog box of the Windows environment;
  - remove the loaded files from the **Construction site file** list by checking their checkboxes and clicking the **Delete** button;
  - check the checkbox in the line with a file of the construction site you want to use in the calculation and click the **Continue with the selected construction site** button;
  - if the **Continue without a construction site** option is selected, the message window will appear with a suggestion to open the existing construction site (Fig. 7.5-3). If the response is positive, a construction site is selected in the standard **Open** dialog box.

After performing the last action **Cross** with the site prepared earlier is invoked.

The main advantage of selecting the construction site with the help of the respective dialog box is the fact that the software remembers the location of the foundation slab on the site after the first calculation, and you do not need to specify it again at subsequent iteration steps.

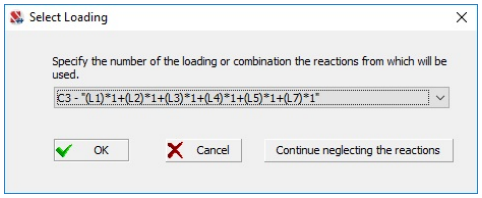


Figure 7.5-1. *The* **Select Loading** *dialog box*

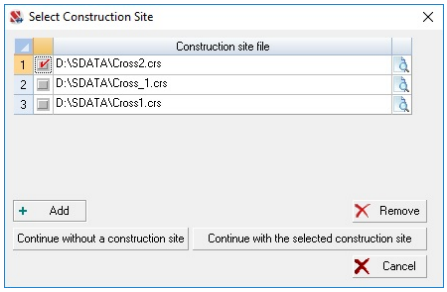


Figure. 7.5-2. *The* **Select Construction Site** *dialog box*

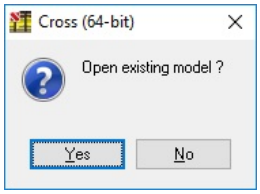


Figure 7.5-3. *The message window*

In the initial state the fragment of the design model (foundation slab) the subsoil parameters are determined for is located in the lower left corner of the site (Fig. 7.5-4), and its contour selected by a cursor in the form of an intersection of arrows can move together with the mouse. After moving the contour into the right position, left click and the position of the slab will be fixed within the dimensions of the site (Fig. 7.5-5).

It should be noted that the fragment of the design model can not be located outside the dimensions of the site, and intersect the lines bounding these dimensions as well as contours of other foundations (for example, the existing buildings), located in the site earlier. When placing the slab within the dimensions of the site it is enough to maintain the accuracy of positioning measured by several meters which almost does not affect the results of the calculation.

If the existing and the analyzed foundations are close enough, the accuracy of positioning must be higher.

At the first calculation the pressure on soil (load) and the footing bottom level are specified by the user according to the rules described in the manual to **Cross**. After obtaining the value $\mathbf{R_z}$ only the footing bottom level is specified for each subsequent iteration step (the additional load can be specified, if necessary).

Vertical displacements of nodes of the slab can be considered as the comparative evaluation of the result obtained at the current step with the result obtained at the previous step. The decision to terminate the iteration process and to take the obtained subsoil parameters as the design ones is made by the user depending on the nature of the considered problem.

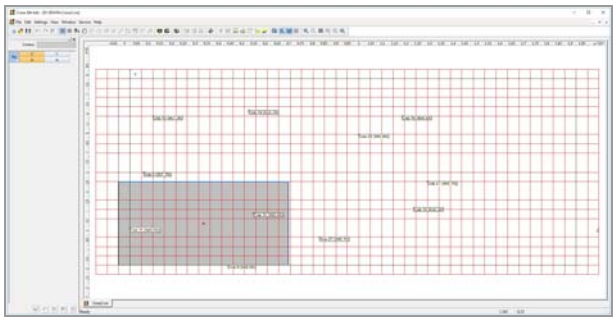


Figure 7.5-4. *Initial position of the foundation slab after loading it into* **Cross**

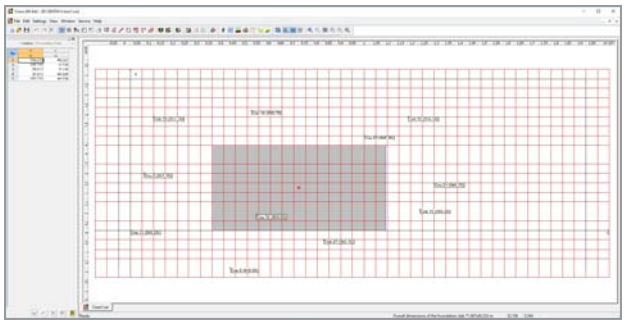


Figure 7.5-5. *Final position of the foundation slab after the fixation*

After performing the analysis in **Cross** the rules for specifying the subsoil parameters for each finite element are determined by the rules selected by the user in the dialog box (Fig. 7.5-6)

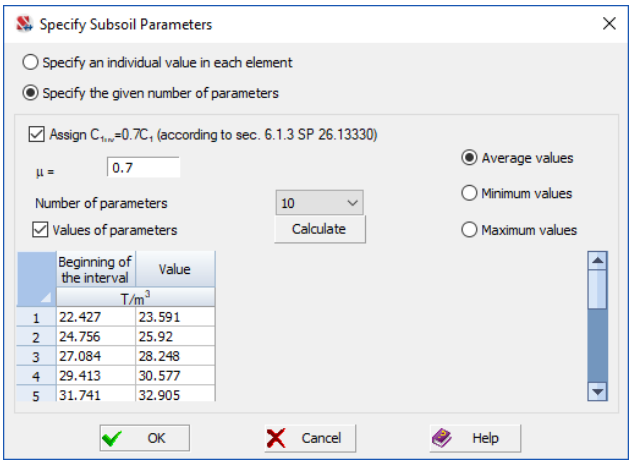


Figure 7.5-6. *The* **Specify Subsoil Parameters** *dialog box*

The rules for specifying the subsoil parameters for the elements of the design model can be selected with the help of the respective radio buttons in this dialog box. There are the following two variants:

- specifying an "individual" value of the subsoil parameter for each finite element included in the considered fragment (on the basis of the value calculated by **Cross** in the center of the element);
- specifying the given number of intervals of the values of parameters according to the specified rules (the used value can be taken as the minimum, maximum or average value of the limits of the interval).

Finally, the user can specify the values of the limits of the intervals, which must be used by the software. In order to do it, check the **Values of parameters** checkbox and specify the necessary values of the parameters in the table.

A special checkbox enables to use a mode where the elements are assigned the coefficient of elastic uniform compression C_1 together with the coefficient of elastic uniform shear C_{1uv} equal to μC_1. By default μ=0.7 (according to the recommendations of Sec. 6.1.3 of SP 26.13330), but this value may be changed by the user.

References to Chapter 7

1. V.G. Piskunov, Y.M. Fedorenko, A dynamic method for monitoring layered slabs on elastic subsoil. – Architecture and Construction in Belarus, No.5-6, 1994, p.19-22.
2. V.S. Karpilovsky, Kriksunov E.Z., A.A. Malyarenko, M.A. Mikitarenko, A.V. Perelmuter, M.A. Perelmuter, V.G. Fedorovsky, V.V. Yurchenko, – SCAD Office. Implementation of SNiP in Computer-Aided Design Applications. – M.: SCAD SOFT, 2014 – 480p.

8. Specifying Loadings

This chapter considers operations of specifying loads and generating loadings for static and dynamic analysis. The control of the operations is performed with the help of the **Loadings** section of the toolbar (Fig. 8-1).

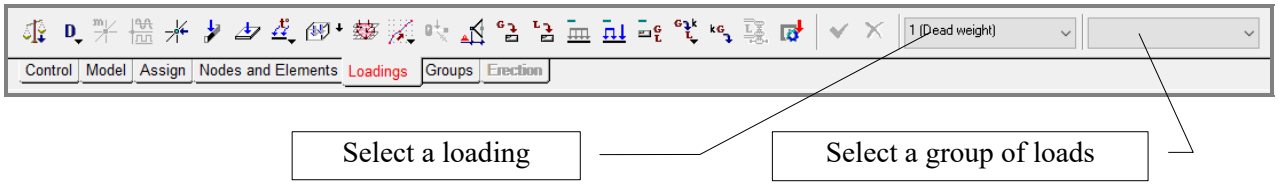


Figure 8-1. *The* **Loadings** *section of the toolbar*

Loads acting on the considered structure are specified as nodal forces and moments, and as concentrated, trapezoid and distributed forces and moments on the elements.

Loadings can include any set of loads and are characterized by a number and a name. If some sets of loads are included in several loadings, **Groups of loads** including these sets can be created in the process of generating loadings. Groups are saved under the indicated names and can be added to any loadings.

The **Loadings** section includes the following operations:

— automatic specification of the self-weight;

— parameters of dynamic loadings;

— inertial properties;

— parameters of dynamic loads;

— characteristics of nodal loads;

— characteristics of loads on bars;

— characteristics of loads on plate elements;

— characteristics of thermal loads;

— characteristics of loads on solid elements;

— characteristics of loads on regions;

— route;

— characteristics of loads of the “specified displacement” type described as a 0-element;

— characteristics of loads of the “displacements of constraints” type;

— save a group of loads;

— save a loading;

— remove all loads from the model;

— delete loads;

— delete loadings and groups of loads;

— assemble loadings from the groups of loads;

— specify the coefficients of the groups of loads;

— pack (delete “empty”) loadings;

— change parameters (names) of loadings.

8.1. Specifying Static Loadings

The following procedure is recommended to specify static loadings:

- select the type of the load (nodal or on the elements) using the buttons in the toolbar;
- specify the type, direction and value of the load in the dialog box with the parameters of the load;
- select nodes or elements the specified load is snapped to;
- click the **OK** button in the **Loadings** section of the toolbar or press the **Enter** key in the keyboard;
- repeat the above operations for other loads included in the current loading;

- click the **Save/Add loading** button after specifying all the loads of the current loading;
- specify the name of the loading and its characteristics in the **Save Loading** dialog box, and click the **OK** button;
- click the **Remove all loads** button in the toolbar before specifying a new (next) loading.

The **Save/Add loading** operation — must finish the specification of loads of the current loading. Otherwise the specified loads will not be taken into account in the calculation. Characteristics of loadings specified in the **Save Loading** dialog box are automatically taken into account when generating design combinations of forces (DCF). It is recommended to give loadings unique names characterizing the purpose of loads.

The **Remove all loads** operation clears the temporary (buffer) memory where the loads of the current loading are accumulated and enables to prepare for creating a new loading. This operation must obviously be performed after saving the loading.

The rules for working with groups of loads and correcting loadings are described below.

Automatic Specification of the Self-weight

The operation is performed for bars, plates and solid elements — and only when the specific weight of the material is specified in the stiffness properties. The self-weight of elements per unit length/area is calculated as the product of the cross-sectional area of bars or the plate thickness and the specific weight of the material. When the operation is performed for bars and the numerical or the numerical and parametric description of rigidities is used, **SCAD** works according to the following algorithm — the value of EF is taken from the numerical description, it is divided by the Young's modulus E (we obtain an analogue of the cross-sectional area) and multiplied by the specific weight of the material.

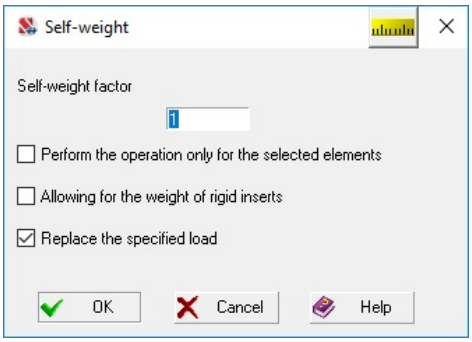


Figure 8.1-1. *The* **Self-weight** *dialog box*

Once the operation is invoked the **Self-weight** dialog box appears, where you can specify the self-weight factor, and the rules for specifying the loads:

- specify the loads for all the elements or only for those selected in the model which enables to specify different factors for different groups of elements;
- include or not include rigid inserts in the length of the element, i.e. allow for the "weight" of rigid inserts;
- replace the self-weight factor at the reactivation of the operation or sum the new value of the factor and the previous one.

If in the process of performing the operation elements the self-weight load cannot be assigned to are detected, a message window with the list of such elements appears. This message is duplicated into the list of performed operations in the bottom of the window, where the operations of selecting the indicated elements are provided.

Loads from the self-weight are modeled as distributed forces acting on the elements in the direction of the Z axis of the global coordinate system. The filters button is used to display the distributed loads on the model. When the filters button is pressed, the value of the load is displayed next to the image.

Nodal Loads

Nodal loads are specified in the **Input Nodal Loads** dialog box, which appears after clicking the respective button in the **Loadings** section of the toolbar.

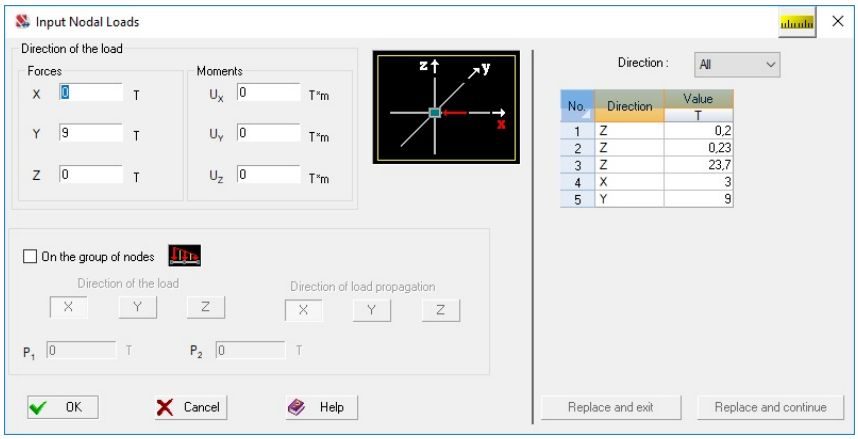


Figure 8.1-2. *The* **Input Nodal Loads** *dialog box*

If the loads in different directions are acting on nodes in one loading, such a set of loads can be specified at once. The icon displaying the positive direction of the respective load will show the direction of the load which has a text field with a cursor in it. The following procedure is recommended to input the nodal loads:

- invoke the **Nodal loads** operation;
- specify the values of the loads acting on one node at the same time in the respective dialog box;
- close the dialog box by clicking the **OK** button;
- select the nodes in the model the specified set of loads will be acting on;
- click the **OK** button in the toolbar or press the **Enter** key in the keyboard.

Characteristics of all the nodal loads specified earlier in the current loading are displayed in the table of the **Input Nodal Loads** dialog box. These data can be used to specify new loads or correct those already specified. The **Direction** list is used to select loads by the direction of their action (all the loads are displayed by default).

If when selecting one node, there are several nodes in the cursor target at once, their list will be output in the **Nodes in the Cursor Target** dialog box (Fig. 8.1-3). The node selected in the list of this dialog box is displayed in red in the model, and its coordinates are output in the respective text fields of the dialog box. Click the **Select** button to select the node, the dialog box will close and you will be able to use this node to specify the load.

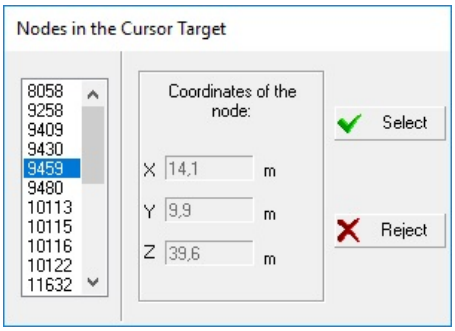


Figure 8.1-3. *The* **Nodes in the Cursor Target** *dialog box*

Correcting Loads Specified Earlier

It is necessary to perform the following actions to correct the loads specified earlier:

- select the loading which includes the nodal loads you want to correct from the list of the toolbar of the **Loadings** mode;

- invoke the **Nodal loads** operation;
- select a line with the load you want to correct in the table with the list of the specified loads in the **Input Nodal Loads** dialog box, and the characteristics of the load (direction and value) will be input into the respective text field;
- change the characteristics of the load;
- click the **Replace and exit** button (the dialog box closes) if only this load changes, or **Replace and continue** button if other nodal loads in the current loading will change as well.

  There is one more method to correct/delete nodal loads implemented in the mode of obtaining information on the node (filters button).

After correcting all the loads it is necessary to save the loading.

Using the Characteristics of the Specified Load to Create New Ones

If when creating a new load it turns out that such a load (type, value, direction, …) has been already used in the current loading, you can perform the following actions to apply it again:

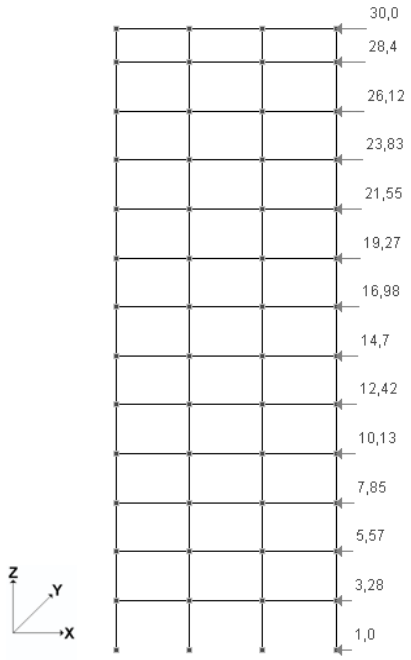
- invoke the **Nodal loads** operation;
- select a line with the characteristics of the necessary load in the table with the list of the specified loads in the **Input Nodal Loads** dialog box (the characteristics of the load will be input into the respective text field);
- click the **OK** button (the dialog box closes) and apply the selected load to the nodes of the model.

  After specifying all the loads it is necessary to save the loading.

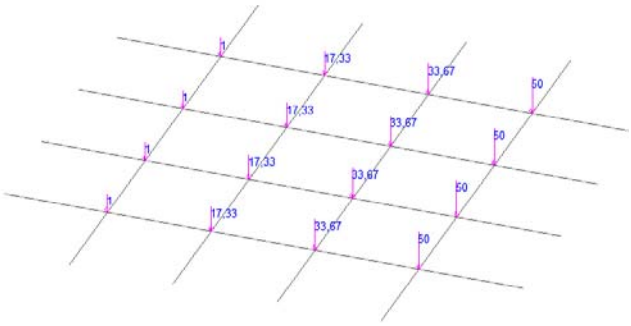
Specifying Loads for a Group of Nodes

When creating nodal loads you can specify loads of the hydrostatic type for a group of selected nodes. The law of their variation with the spatial coordinate corresponding to the "depth of the water body" is linear. "The depth of the water body" can be specified in any direction.

When specifying such loads their value will change linearly from the specified initial value to the final one depending on the coordinate of the node to which the load is applied in the direction of propagation (direction of the increase of “depth” is specified by the user).



Example 1
Direction of load propagation Z, direction of the load X, range of the load variation $P_1 = 1$, $P_2 = 30$.



Example 2
Direction of load propagation X, direction of the load Z, range of the load variation $P_1 = 1$, $P_2 = 50$.

It is necessary to perform the following actions to specify loads for a group of nodes:

- invoke the **Nodal loads** operation;
- the **Input Nodal Loads** dialog box will appear, where you have to check the **On the group of nodes** checkbox;
- select the direction of the load (the operation is performed only for the directions along the axes of the global coordinate system);
- select the direction of load propagation;
- specify the initial (P_1) and final (P_2) values of load and click the **OK** button in the dialog box (the initial load value P_1 will be assigned to the node (nodes) with the minimum value of the coordinate in the direction of propagation);
- select the nodes in the design model and apply the loads to them by clicking the **OK** button in the **Loadings** section of the toolbar or pressing the **Enter** key in the keyboard.

Loads on Bar Elements

Type, direction and value of loads on bar elements are specified in the **Specify Loads on Bar Elements** dialog box (Fig. 8.1-4), which appears after clicking the respective button in the **Loadings** section of the toolbar.

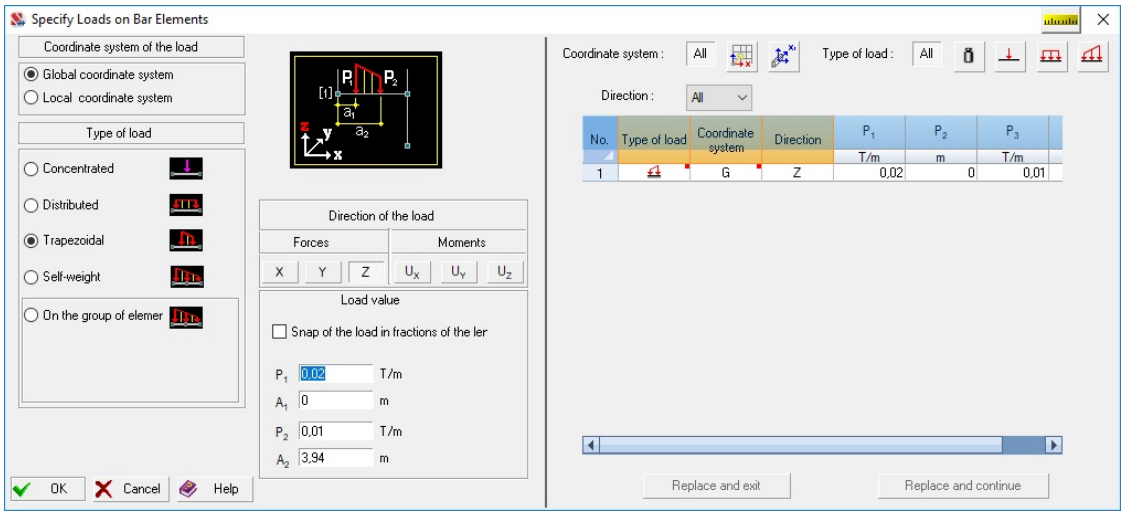


Figure 8.1-4. *The* **Specify Loads on Bar Elements** *dialog box*

The following types of loads are provided in the software:

- concentrated load;
- load distributed over a part of the element;
- uniformly distributed load on the whole element ;
- self-weight.

All types of loads can be specified in the global or in the local coordinate system. Moreover, a load of the "hydrostatic" type can be applied to a group of elements.

The following procedure is recommended to specify loads on bars:

- invoke the **Loads on bars** operation;
- specify the coordinate system, type, direction, value, and, if necessary, — snap of the load in the **Specify Loads on Bar Elements** dialog box (when the type of the load is selected, the dialog box displays an icon showing the positive direction of its action);
- close the dialog box by clicking the **OK** button;
- select the elements in the model the specified load is applied to;
- click the **OK** button in the toolbar or press the **Enter** key in the keyboard.

Characteristics of all loads on bars specified earlier in the current loading are displayed in the table of the dialog box. These data can be used to specify new loads or correct those already specified. A set of marker buttons with an image of the type of load and the **Direction** drop-down list are used to select loads

by their type, direction of action and the coordinate system (all loads for all directions including the loads from self-weight are displayed by default).

If when selecting one element, there are several elements in the cursor target at once, their list will be output in the **Elements in the Cursor Target** dialog box (Fig. 8.1-5). The bar selected in the list of this dialog box is displayed in red in the model. Click the **Select** button to select the element, the dialog box will close and you will be able to use this element to specify the load.

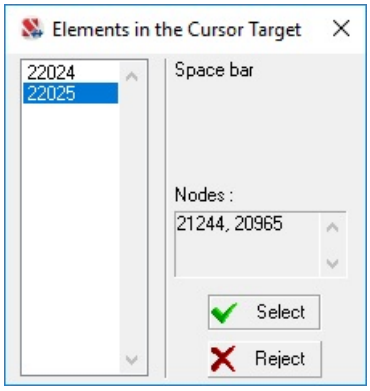


Figure 8.1-5. *The* **Elements in the Cursor Target** *dialog box*

When specifying concentrated and trapezoidal loads the control of snapping loads along the length of the element is performed and if the load is outside the element, a message with the list of elements the load is not applied to is generated. This message is duplicated into the list of performed operations in the bottom of the window.

Correcting Loads Specified Earlier

It is necessary to perform the following actions to correct the loads specified earlier:

- select the loading which includes the loads on bars you want to correct from the list of the toolbar of the **Loadings** mode;
- invoke the **Loads on bars** operation;
- select a line with the load you want to correct in the table with the list of the specified loads in the **Specify Loads on Bar Elements** dialog box, and the characteristics of the load (type, direction, value, snap) will be input into the respective text fields;
- change the characteristics of the load;
- click the **Replace and exit** button if the characteristics of only one load change (the specified changes will be applied after closing the dialog box), or the **Replace and continue** button if there are several such loads in the current loading (in this case the changes will be displayed in the table after clicking this button).

The correction mode is not used for creating new loads, because the characteristics of loads are changed locally and are not saved. After finishing the work with the current loading it is necessary to save it.

Using the Characteristics of the Specified Load to Create New Ones

If when creating a new load it turns out that such a load (type, value, direction, …) has been already used in the current loading, you can perform the following actions to apply it again:

- invoke the **Loads on bars** operation;
- select a line with the characteristics of the necessary load in the table with the list of the specified loads of the **Specify Loads on Bar Elements** dialog box (the characteristics of the load will be input into the respective text fields);
- click the **OK** button (the dialog box closes);
- apply the selected load to the elements.

After specifying and correcting all the loads it is necessary to save the loading.

Specifying Loads for a Group of Elements

When creating loads on bars you can specify loads of the hydrostatic type for a group of elements. The law of their variation with the spatial coordinate corresponding to the "depth of the water body" is linear. "The depth of the water body" can be specified in any direction.

When specifying such loads (in the form of trapezoidal loads along the length of the element) will change linearly from the specified initial value to the final one in the direction of propagation (direction of the increase of "depth" is specified by the user).

It is necessary to perform the following actions to specify loads for a group of elements:

- invoke the **Loads on bars** operation;
- the **Specify Loads on Bar Elements** dialog box will appear, where you have to check the **On the group of elements** checkbox;
- select the direction of the load;
- select the direction of load propagation;
- specify the initial (P_1) and final (P_2) values of load and click the **OK** button in the dialog box (the initial load value P_1 will be assigned to the node/nodes of the elements with the minimum value of the coordinate in the direction of propagation);
- select the elements in the design model and apply the loads to them by clicking the **OK** button in the **Loadings** section of the toolbar or pressing the **Enter** key in the keyboard.

Loads on Plates

Type, direction and the value of loads on plate elements (shells, plates, deep beams) are specified in the **Specify Loads on Plate Elements** dialog box (Fig. 8.1-6), which appears after clicking the respective button in the **Loadings** section of the toolbar.

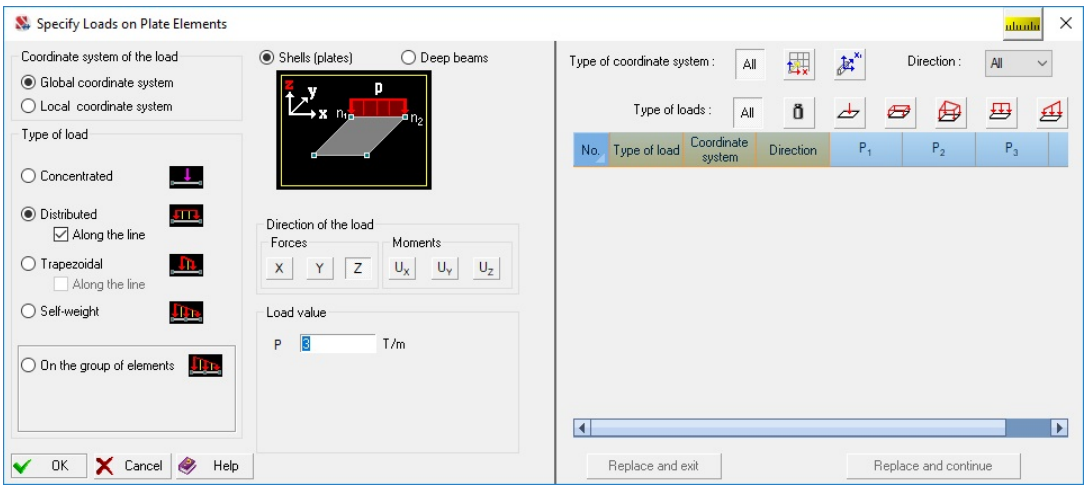


Figure 8.1-6. *The* **Specify Loads on Plate Elements** *dialog box*

The following types of loads are provided in the software:

- concentrated load;
- uniformly distributed load on the element;
- uniformly distributed load between two nodes of the element (load along the line);
- trapezoidal load on the element;
- trapezoidal load between two nodes of the element (load along the line);
- self-weight.

All types of loads can be specified in the global or in the local coordinate system. Moreover, a load of the "hydrostatic" type can be applied to a group of elements.

The following procedure is recommended to specify loads on plates:

- invoke the **Loads on plates** operation;
- specify the coordinate system, type, direction, value, and, if necessary, — snap of the load in the **Specify Loads on Plate Elements** dialog box (when the type of the load is selected, the dialog box displays an icon showing the positive direction of its action);
- close the dialog box by clicking the **OK** button;
- select the elements in the model the specified load is applied to;

− click the **OK** button in the toolbar or press the **Enter** key in the keyboard.

Characteristics of all loads on plates specified earlier in the current loading are displayed in the table of the dialog box. These data can be used to specify new loads or correct those already specified. A set of marker buttons with an image of the type of load and the **Direction** drop-down list are used to select loads by their type, direction of action and the coordinate system (all loads for all directions including the loads from self-weight are displayed by default).

The dialog box displays an icon showing the positive direction of the load. Since a system for describing local axes assumed for deep beams differs from that for plates and shells, the **Deep beams** radio button should be selected when specifying loads on deep beams in the local coordinate system.

If when selecting one element, there are several elements in the cursor target at once, their list will be output in the **Elements in the Cursor Target** dialog box (Fig. 8.1-5). The element selected in the list of this dialog box is displayed in red in the model. Click the **Select** button to select the element, the dialog box will close and you will be able to use this element to specify the load.

When specifying concentrated loads the control of snapping loads within the element is performed and if the load is outside the element, a message with the list of elements the load is not applied to is generated. This message is duplicated into the list of performed operations in the bottom of the window.

Loads along the Line

The load on plate elements can be distributed along the **line** connecting any two nodes of the element. It is necessary to perform the following actions to specify this load:

− invoke the **Loads on plates** operation;
− select the type of load (uniformly distributed or trapezoidal) and check the **Along the line** checkbox in the **Specify Loads on Plate Elements** dialog box;
− specify the coordinate system, direction and value of the load (when the type of the load is selected, the dialog box displays an icon showing the positive direction of its action);
− click the **OK** button in the dialog box;
− select the elements in the model to the nodes of which the load is snapped;
− select the nodes in the model belonging to the selected elements the load is snapped to;
− click the **OK** button in the **Loadings** section.

The operation will be performed correctly if each selected element will have only two selected nodes. Otherwise an error message with the numbers of elements is generated.

Correcting Loads Specified Earlier

It is necessary to perform the following actions to correct the loads specified earlier:

− select the loading which includes the loads you want to correct from the list of the toolbar of the **Loadings** mode;
− invoke the **Loads on plates** operation;
− select a line with the load you want to correct in the table with the list of the specified loads in the **Specify Loads on Plate Elements** dialog box, and the characteristics of the load (type, direction, value, snap) will be input into the respective text fields;
− change the characteristics of the load;
− click the **Replace and exit** button if the characteristics of only one load change (the specified changes will be applied after closing the dialog box), or the **Replace and continue** button if there are several such loads in the current loading (in this case the changes will be displayed in the table after clicking this button).

There is one more method to correct/delete loads on an element implemented in the mode of obtaining information on the element (filters button).

After finishing the work with the current loading it is necessary to save it.

Using the Characteristics of the Specified Load to Create New Ones

If when creating a new load it turns out that such a load (type, value, direction, …) has been already used in the current loading, you can perform the following actions to apply it again:

− invoke the **Loads on plates** operation;

- select a line with the characteristics of the necessary load in the table with the list of the specified loads in the **Specify Loads on Plate Elements** dialog box (the characteristics of the load will be input into the respective text fields);
- click the **OK** button (the dialog box closes);
- apply the selected load to the elements.

After specifying and correcting all the loads it is necessary to save the loading.

Specifying Loads for a Group of Elements

When creating loads on plates you can specify loads of the hydrostatic type for a group of elements (or one element). The law of their variation with the spatial coordinate corresponding to the "depth of the water body" is linear. "The depth of the water body" can be specified in any direction.

When specifying such loads (in the form of trapezoidal loads over the field of the element) will change linearly from the specified initial value to the final one in the direction of propagation (direction of the increase of “depth” is specified by the user).

It is necessary to perform the following actions to specify loads for a group of elements:

- invoke the **Loads on plates** operation;
- the **Specify Loads on Plate Elements** dialog box will appear, where you have to check the **On the group of elements** checkbox;
- select the direction of the load;
- select the direction of load propagation;
- specify the initial (P_1) and final (P_2) values of load and click the **OK** button in the dialog box (the initial load value P_1 will be assigned to the node/nodes of the elements with the minimum value of the coordinate in the direction of propagation);
- select the elements in the design model and apply the loads to them by clicking the **OK** button in the **Loadings** section of the toolbar or pressing the **Enter** key in the keyboard.

Specifying Loads on Solid Elements

The software enables to specify the following types of loads on solid elements:

- load from the self-weight;
- uniformly distributed load over the volume of the element;
- uniformly distributed load on the face of the element;
- concentrated load.

The type of load is selected from the drop-down menu which appears after clicking the “right arrow”. An exception is the load from the self-weight which is specified in the same way as for other types of finite elements by invoking the operation .

It is necessary to perform the following actions to specify a uniformly distributed load for any type of elements:

- invoke the operation by clicking the **Loads on solid elements** button (“right arrow” is not used for this type of load);
- the **Distributed Loads on Solid Elements** dialog box will appear (Fig. 8.1-7), where you have to select the coordinate system in which the loads will act and the direction of their action;
- specify the load value;
- close the dialog box by clicking the **OK** button and select the elements in the model for which the load is specified;
- click the **OK** button in the toolbar.

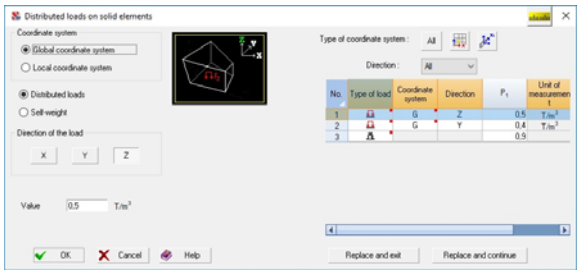


Figure 8.1-7. *The* **Distributed Loads on Solid Elements** *dialog box*

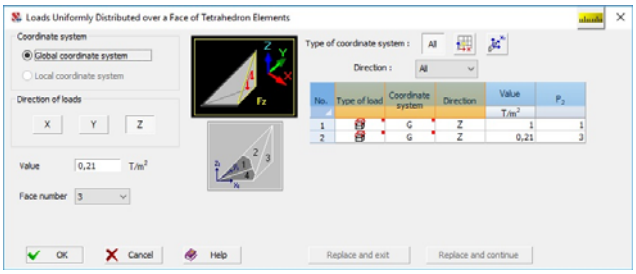


Figure 8.1-8. *The* **Loads Uniformly Distributed over a Face of …** *dialog box*

You should keep in mind that the distributed loads in the local coordinate system can be specified only for the elements of type 31 (parallelepiped) and 33 (triangular prism).

A uniformly distributed load on the faces of the elements is specified depending on the type of the elements and is selected in the drop-down menu. The rules of the numbering of the faces are given in Figure 3.5-1 and are duplicated in the icons in the respective dialog boxes.

The following rules for specifying the loads on the faces of the elements are recommended if the faces are specified by their numbers:

- invoke the operation by clicking the “right arrow” and selecting a section from the drop-down menu corresponding to the type of the elements;
- the **Loads Uniformly Distributed over a Face of …** dialog box will appear (Fig. 8.1-8), where you have to select the coordinate system in which the loads will act and the direction of their action;
- specify the load value;
- select the number of the face in the drop-down list for which the loads will be specified;
- close the dialog box by clicking the **OK** button and select the elements in the model for which the load is specified;
- click the **OK** button in the toolbar.

Since the numbering of the faces is performed in the local coordinate system of each element, you might not be able to specify the loads for all the necessary elements at once, because the directions of their local axes do not coincide, and therefore the positions of the respective faces in the model do not coincide as well. In this case you can use the operation of specifying a load which enables to select the faces not by their number but by the (selected) nodes belonging to this face. This operation is invoked from the **Load on the faces of the selected solid elements** section of the drop-down menu.

In order to perform the last operation you have to:

- invoke the operation by clicking the “right arrow” and select the above section of the drop-down menu;
- a dialog box will appear (Fig. 8.1-9), where you have to select the coordinate system in which the loads will act and the direction of their action;
- specify the load value;
- close the dialog box by clicking the **OK** button and select the elements in the model for which the load is specified;
- click the **OK** button in the toolbar;
- once the **Select nodes** message appears, select the nodes belonging to the loaded faces;
- click the **OK** button in the toolbar.

After performing the operation the number of the face to which the load is applied is specified in the P_2 column of the table of loads of the respective dialog box.

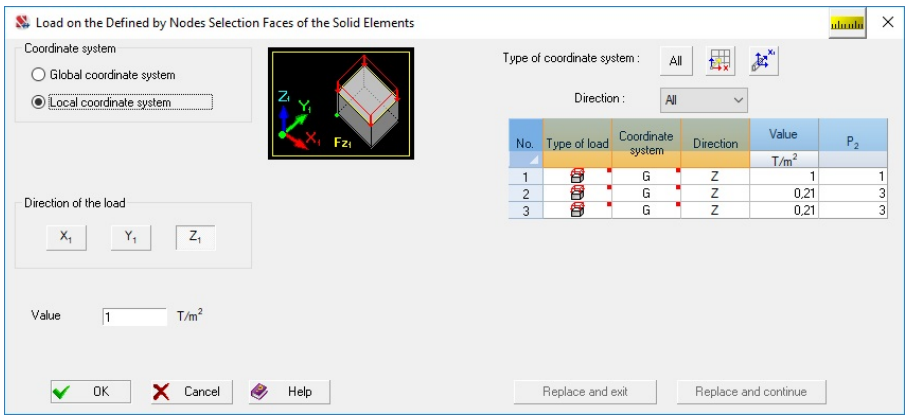


Figure 8.1-9. *The* **Load on the Faces of the Selected Solid Elements** *dialog box*

The correction of the parameters of the load specified earlier is performed according to the same rules as for the other types of elements, i.e. using the **Replace and exit** or **Replace and continue** operations. The following procedure is recommended for performing the correction operations:

- select the number of the loading which includes the loads you want to correct from the list of the **Loadings** toolbar;
- click the right arrow and select the section of the menu with the type of load;
- select a line with the load you want to correct in the table of loads in the dialog box;
- change the parameters of the load and save the new values by clicking the **Replace and exit** or **Replace and continue** buttons;
- after completing the correction save the new state of the loading using the respective operation and the **Replace** option or create a new one on the basis of the previous loading.

When correcting the value of the load you should keep in mind that not the value of the load from the self-weight but the coefficient with which this load is included in the current loading is specified in the P_1 column of the tables of the dialog boxes. When changing the value of the coefficient (by selecting the respective line in the table of loads of the dialog box) the marker of the type of load will automatically switch into the **Self-weight** state.

Thermal Loads

Thermal loads are specified depending on the type of the element. The software enables to specify the following types of thermal load:

- loads on bars:
  - along the bar (in the direction of the local X_1 axis);
  - along the local Y_1 axis;
  - along the local Z_1 axis;
- loads on plates:
  - over the whole field of the element;
  - bending about the local X_1 axis;
  - bending about the local Y_1 axis;
- loads on deep beams:
  - over the whole field of the element;
  - along the local X_1 axis;
  - along the local Z_1 axis (the Z_1 axis of the elements of the deep beam lies in the plane of the element);

- loads on shells:
  - over the whole field of the element;
  - along the local X_1 axis;
  - along the local Y_1 axis;
  - bending about the local X_1 axis;
  - bending about the local Y_1 axis;
- loads on solid elements:
  - thermal load throughout the whole volume of the element.

The following procedure is recommended to specify thermal loads:

– click the **Thermal loads** button in the **Loadings** section of the toolbar;
– select the type of elements the loads are applied to in the drop-down menu;
– select a radio button defining the type of load in the **Thermal Loads …** dialog box, which depends on the selected type of element (for example, **thermal load on bars** is specified in the respective dialog box (Fig. 8.1-10);
– specify the parameters of the load;
– close the dialog box by clicking the **OK** button;
– select the elements in the model the specified type of load is applied to;
– click the **OK** button in the toolbar or press the **Enter** key in the keyboard.

If when specifying the loads the **Thermal loads** filters button is pressed, the specified loads will be displayed in the model.

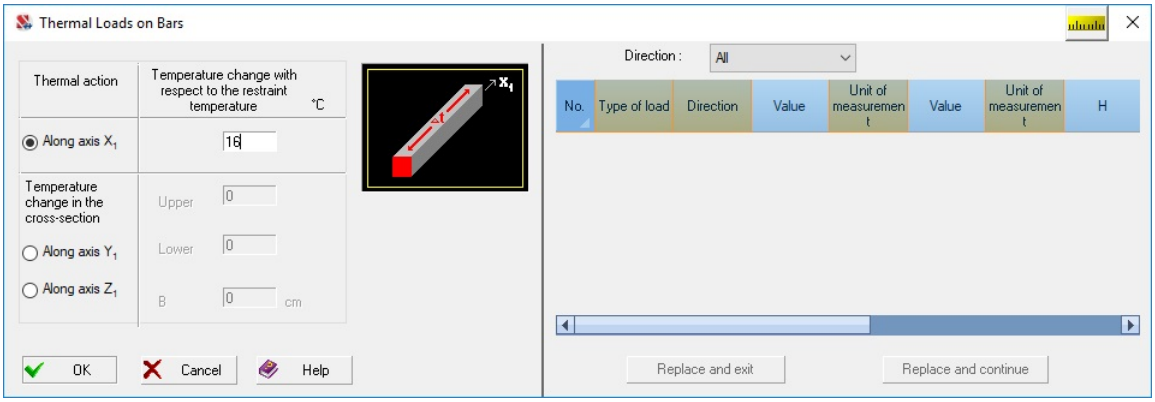


Figure 8.1-10. *The* **Thermal Loads on Bars** *dialog box*

Correcting Loads Specified Earlier

It is necessary to perform the following actions to correct the loads specified earlier:

– select the loading which includes the loads you want to correct from the list of the toolbar of the **Loadings** mode;
– invoke the **Thermal Loads** operation;
– select the type of elements the loads are applied to in the drop-down menu;
– select a line with the load you want to correct in the table with the list of the specified loads in the **Thermal Loads** dialog box, and the characteristics of the load will be input into the respective text fields (for example, a **thermal load on bars** is specified in the respective dialog box shown in Fig. 8.1-10);
– change the characteristics of the load (including its type if necessary);
– click the **Replace and exit** button if the characteristics of only one load change (the specified changes will be applied after closing the dialog box), or the **Replace and continue** button if there are several such loads in the current loading (in this case the changes will be displayed in the table after clicking this button).

There is one more method to correct/delete loads on an element implemented in the mode of obtaining information on the element (filters button).

After finishing the work with the current loading it is necessary to save it.

Using the Characteristics of the Specified Load to Create New Ones

If when creating a new load it turns out that such a load (type, value, direction, …) has been already used in the current loading, you can perform the following actions to apply it again:

- invoke the **Thermal loads** operation;
- select the type of elements the loads are applied to in the drop-down menu;
- select a line with the characteristics of the necessary load in the table with the list of the specified loads in the **Thermal Loads** dialog box (the characteristics of the load will be input into the respective text fields);
- click the **OK** button (the dialog box closes);
- apply the selected load to the elements.

After specifying and correcting all the loads it is necessary to save the loading.

Loads on Region

When creating loads on region you can specify loads on lines and areas "unrelated" to the finite element mesh. The load lines are segments or polylines with vertices coinciding with the nodes of the design model. The load area is a polygon with vertices coinciding with the nodes of the design model.

Type, direction and value of loads are specified in the **Specify Loads on Region** dialog box (Fig. 8.1-11), which appears after clicking the respective button in the **Loadings** section of the toolbar.

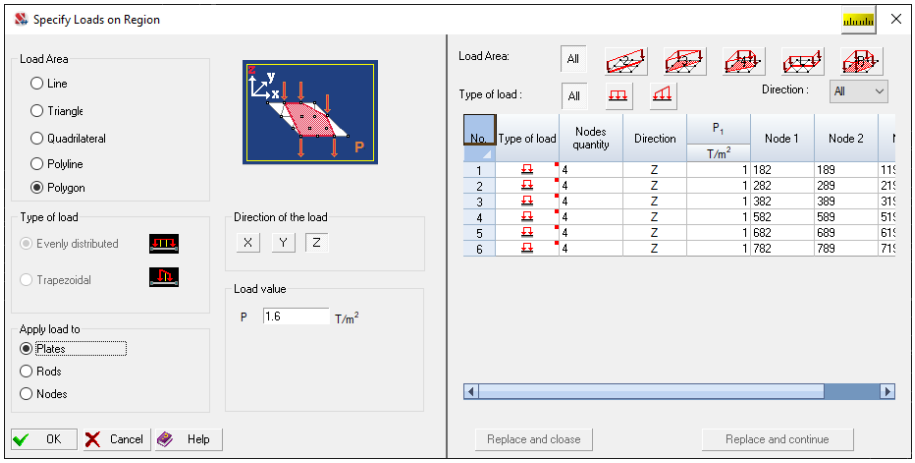


Figure. 8.1-11. *The* **Specify Loads on Region** *dialog box*

The following types of loads are provided in the software:

- uniformly distributed and trapezoidal loads along the line;
- uniformly distributed and trapezoidal loads on a triangle;
- uniformly distributed and trapezoidal loads on a quadrangle;
- uniformly distributed loads on a polyline;
- uniformly distributed loads on an arbitrary polygon.

All types of loads are specified in the global coordinate system. The load is transferred only to the plate elements if the load area is a line or polyline. The distance from the nodes of the load area to the plane a certain plate element lies in should not exceed the value of the **Accuracy of the region's load position** which is specified in the **Error! Hyperlink reference not valid.** (the **Special** tab).

In cases where the load area is a polygon (in particular, a triangle or a quadrilateral), the load can be transferred to plates, nodes or rods (the desired option must be selected by the user in the dialog box).

The following procedure is recommended to specify area loads:

- invoke the **Loads on Region** operation;
- specify the type, direction, value (values – in the case of a trapezoidal load) of the load in the **Specify Loads on Region** dialog box (when the type of the load is selected, the dialog box displays an icon showing the positive direction of its action);

- close the dialog box by clicking the **OK** button;
- select the nodes in the model which define a line/polyline/area;
- a polyline can be completed by double clicking after specifying the last node or by clicking the **OK** button in the toolbar or pressing the **Enter** key in the keyboard. An area can be closed by pointing at the first node.

Characteristics of all loads on region specified earlier in the current loading are displayed in the table of the dialog box. These data can be used to specify new loads or correct those already specified. A set of marker buttons with an image of the type of load and the **Direction** drop-down list are used to select loads by their type and direction of action (all loads for all directions are displayed by default).

Correcting Loads Specified Earlier

It is necessary to perform the following actions to correct the loads specified earlier:

- select the loading which includes the area loads you want to correct from the list of the toolbar of the **Loadings** mode;
- invoke the **Loads on Region** operation;
- select a line with the load you want to correct in the table with the list of the specified loads in the **Specify Loads on Region** dialog box, and the characteristics of the load (type, direction, values) will be input into the respective text fields;
- change the characteristics of the load;
- click the **Replace and exit** button if the characteristics of only one load change (the specified changes will be applied after closing the dialog box), or the **Replace and continue** button if there are several such loads in the current loading (in this case the changes will be displayed in the table after clicking this button).

The correction mode is not used for creating new loads, because the characteristics of loads are changed locally and are not saved. After finishing the work with the current loading it is necessary to save it.

Using the Characteristics of the Specified Load to Create New Ones

If when creating a new load it turns out that such a load (type, value, direction, …) has been already used in the current loading, you can perform the following actions to apply it again:

- invoke the **Loads on Region** operation;
- select a line with the characteristics of the necessary load in the table with the list of the specified loads of the **Specify Loads on Region** dialog box (the characteristics of the load will be input into the respective text fields);
- close the dialog box by clicking the **OK** button;
- select a new line or load area.

After specifying and correcting all the loads it is necessary to save the loading.

The tabular representation of the information on the loads on region can be obtained for the selected loading.

8.2. Analysis for the Specified Displacements

The program provides two variants of the analysis for the specified displacements — by using the so-called null elements, which enable to specify the displacement the direction of which does not coincide with the direction of the axes of the global coordinate system, and by specifying the displacement of a constraint in the direction of the axes of the global coordinate system.

Loads through “Null Elements”

When performing the analysis for the specified displacements using the “null elements” (type 154) the displacements are specified in accordance with the index of the model type as loads with respect to the local axes of the “null element”.

A “null element” actually plays the role of a constraint specified in the coordinate system coinciding with the local coordinate system of the element in the directions for which the nonzero stiffness is specified

in the element. Thus specifying the displacement with the help of a “null element”, you specify the displacement of the constraint.

The value of the displacement is specified for linear displacements in the units of length indicated in the dialog box (e.g., in meters), likewise for angular displacements — in the units of angles. The positive direction of the linear displacements coincides with the direction of the local axes of the element, and that of the angular ones — clockwise when viewed from the end of the respective axis.

Direction and value of the displacement are specified in the **Analysis for the Specified Displacements** dialog box (Fig. 8.2-1).

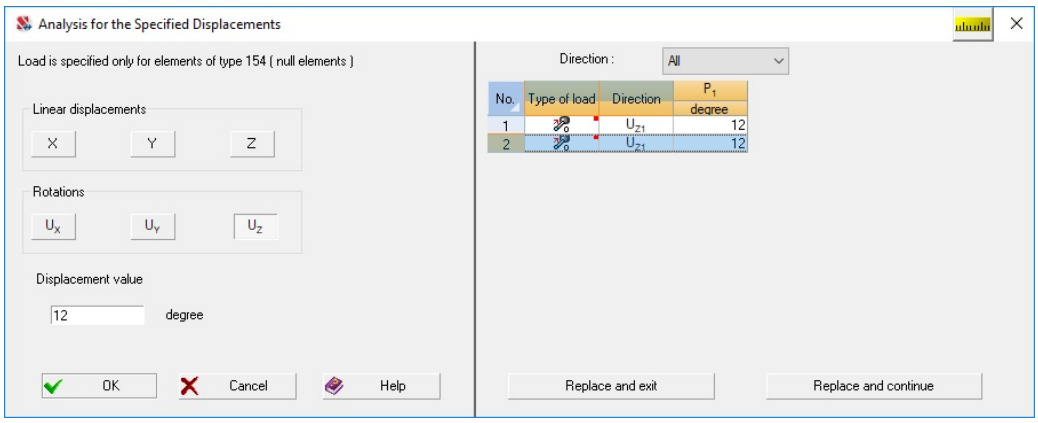


Figure 8.2-1. *The* **Analysis for the Specified Displacements** *dialog box*

The following procedure is recommended to specify loads:

- click the **Load of the "specified displacement" type (described as a 0-element)** button in the **Loadings** section of the toolbar;
- select the marker defining the direction of displacement in the **Analysis for the Specified Displacements** dialog box;
- specify the value of the displacement;
- close the dialog box by clicking the **OK** button;
- select the “null elements” in the model the specified displacement is applied to;
- click the **OK** button in the toolbar or press the **Enter** key in the keyboard.

If when specifying the loads the **Display specified displacements** filters button is pressed, the specified loads will be displayed in the model.

The specified displacements can be applied only in the directions for which the nonzero stiffness is specified in the “null element”.

Correcting Loads Specified Earlier

It is necessary to perform the following actions to correct the loads specified earlier:

- select the loading which includes the loads you want to correct from the list of the toolbar of the **Loadings** mode;
- invoke the **Load of the "specified displacement" type (described as a 0-element)** operation;
- select a line with the load you want to correct in the table with the list of the specified loads in the dialog box, and the characteristics of the load will be input into the respective text fields;
- change the characteristics of the load (including the direction if necessary);
- click the **Replace and exit** button if the characteristics of only one load change (the specified changes will be applied after closing the dialog box), or the **Replace and continue** button if there are several such loads in the current loading (in this case the changes will be displayed in the table after clicking this button).

There is one more method to correct/delete loads on an element implemented in the mode of obtaining information on the element (filters button).

After finishing the work with the current loading it is necessary to save it.

Using the Characteristics of the Specified Load to Create New Ones

If when creating a new load it turns out that such a load (type, value, direction, …) has been already used in the current loading, you can perform the following actions to apply it again:

- invoke the **Load of the "specified displacement" type (described as a 0-element)** operation;
- select a line with the characteristics of the necessary load in the table with the list of the specified loads in the dialog box (the characteristics of the load will be input into the respective text fields);
- click the **OK** button (the dialog box closes);
- apply the selected load to the “null elements”.

After specifying and correcting all the loads it is necessary to save the loading.

Specifying a Load by the Displacement of a Constraint

In order to perform the analysis for the specified displacements you can specify a nodal load of the special type — displacement of the constraint in the coordinate system of the problem.

The load can be specified only in the nodes where a constraint is imposed in the direction of the specified displacement The value of the displacement is specified for linear displacements in the units of length selected for the problem (e.g., in meters), likewise for angular displacements — in radians or degrees. The positive direction of the linear displacements coincides with the direction of the axes of the global coordinate system, and that of the angular ones — clockwise when viewed from the end of the respective axis. Direction and value of the displacement are specified in **Analysis for the Specified Displacements** dialog box (Fig. 8.2-2).

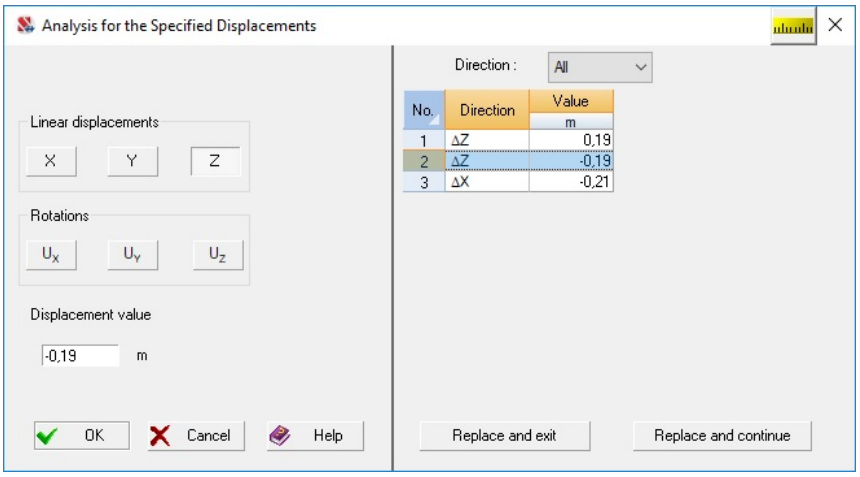


Figure 8.2-2. *The* **Analysis for the Specified Displacements** *dialog box*

The following procedure is recommended to specify loads:

- click the **Load of the "specified displacement" type (described as the displacement of the constraint)** button in the **Loadings** section of the toolbar;
- select the marker defining the direction of displacement in the **Analysis for the Specified Displacements** dialog box;
- specify the value of the displacement;
- close the dialog box by clicking the **OK** button;
- select the nodes in the model the specified displacement is applied to;
- click the **OK** button in the toolbar or press the **Enter** key in the keyboard.

If when specifying the loads the **Display specified displacements** filters button is pressed, the specified loads will be displayed in the model.

Correcting Loads Specified Earlier

It is necessary to perform the following actions to correct the loads specified earlier:

- select the loading which includes the loads you want to correct from the list of the toolbar of the **Loadings** mode;
- invoke the **Load of the "specified displacement" type (described as the displacement of the constraint)** operation;
- select a line with the load you want to correct in the table with the list of the specified loads in the dialog box, and the characteristics of the load will be input into the respective text fields;
- change the characteristics of the load (including the direction if necessary);
- click the **Replace and exit** button if the characteristics of only one load change (the specified changes will be applied after closing the dialog box), or the **Replace and continue** button if there are several such loads in the current loading (in this case the changes will be displayed in the table after clicking this button).

There is one more method to correct/delete loads on a node implemented in the mode of obtaining information on the node (filters button).

After finishing the work with the current loading it is necessary to save it.

Using the Characteristics of the Specified Load to Create New Ones

If when creating a new load it turns out that such a load (type, value, direction, …) has been already used in the current loading, you can perform the following actions to apply it again:

- invoke the **Load of the "specified displacement" type (described as the displacement of the constraint)** operation;
- select a line with the characteristics of the necessary load in the table with the list of the specified loads in the dialog box (the characteristics of the load will be input into the respective text fields);
- click the **OK** button (the dialog box closes);
- apply the selected load.

After specifying and correcting all the loads it is necessary to save the loading.

8.3. Saving Loadings

After specifying all the loads it is necessary to save the current loading. It can be done with the help of the **Save/Add loading** operation. After invoking this operation the **Save Loading** dialog box appears (Fig. 8.3-1).

If the loading is new, it is necessary to specify its name, type and load type (these parameters can be selected from the respective lists). You also have to specify the safety factor for load and the fraction of duration for the loading. If the values of the loads are specified on the basis of their characteristic values, you have to check the respective checkbox.

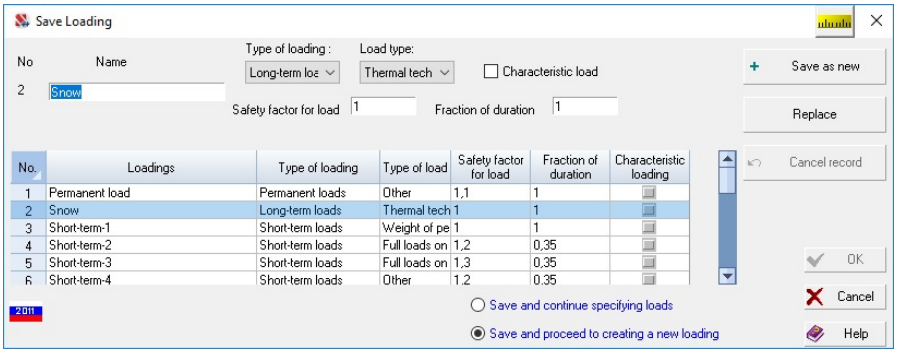


Figure 8.3-1. *The* **Save Loading** *dialog box*

After specifying the characteristics of the new loading, it can be saved by clicking the **Save as new** button. If it is saved after correcting the existing loading (saved earlier), the **Replace** button is used. Before closing the dialog box by clicking the **OK** button it is necessary to specify whether you will continue working with the current loading — the **Save and continue specifying loads** radio button is selected, or with the new one — the **Save and proceed to creating a new loading** radio button is selected.

The peculiarity of the operation is its direct connection with the operation of preparing the initial data for the generation of design combinations of forces (DCF) and displacements (DCD) (see Chapter 18). If the parameters of the loading are specified correctly, they are used as the characteristics of the loading when generating DCF. The changes of the characteristics of loading during the preparation of the initial data of DCF are displayed in the **Save Loading** dialog box.

If the initial data are prepared in the formats of the so-called “old DCF” of versions 11.x and earlier, then before generating or correcting the loadings it is necessary to invoke the operation of preparing the data for the selection of DCF from the project tree, and thus convert the data into the new format. It usually turns out that converting the format is not enough and it will be necessary to correct the data.

8.4. Groups of Loads

Groups of loads are similar to loadings and differ from them in that, firstly, they are not considered in the calculation as loadings but are used only at the stage of generating loadings and, secondly, can be added to any loading. Groups of loads are used in cases when a certain set of loads on nodes and/or elements is included into several loadings in its original form or with different factors. They can be also used as an "addition" to the accumulated loading in the structural analysis taking into account the sequence of erection (the **Erection** mode), and for specifying the weight of the elements of the destroyed structures when modeling the progressive collapse.

Groups of loads are created from the sets of loads or from the loadings created earlier.

It is necessary to perform the following actions to create a group of loads:

- specify the loads that have to be included in the group (the **Remove all loads** button — is used to clear the buffer memory before creating a new group of loads);
- click the **Add a group of loads** button ;
- the **Save Group of Loads** dialog box will appear (Fig. 8.4-1), where you have to specify a unique name of the group (it has to be a name, because groups do not have numbers) and click the **OK** button.

After performing these operations the name of the created group will appear in the **Select a group of loads** list in the toolbar.

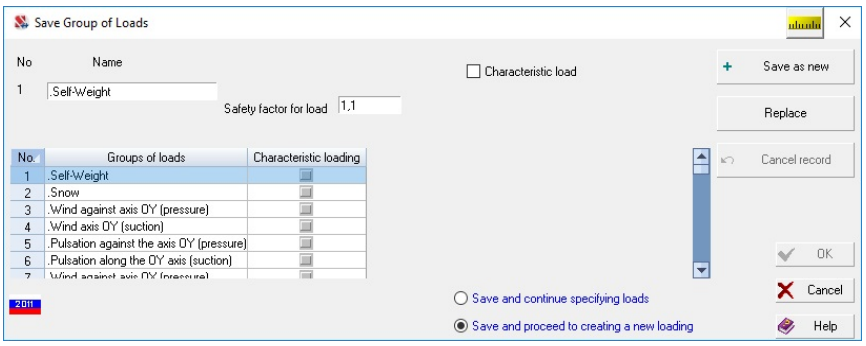


Figure 8.4-1. *The* **Save Group of Loads** *dialog box*

In order to include a group in the loading you just have to select a necessary group in the list. One loading may include any number of groups.

Groups are added to the loading according to the following rules:

1. If the loading is described as a “characteristic load”, and the added group is described as a “characteristic load”:

the load values of the group do not change.

2. If the loading is described as a “characteristic load”, and the added group is described as a “design load”:

before being added to the loading the load values of the group are divided by the safety factor for load specified when the **group** was created**.**

3. If the loading is described as a “design load”, and the added group is described as a “characteristic load”.

before being added to the loading the load values of the group are divided by the safety factor for load specified when the **group** was created and multiplied by the safety factor for load of the **loading.**

4. If the loading is described as a “design load”, and the added group is described as a “design load”. the load values of the group do not change.

5. If the load buffer storage is empty, then when the group is loaded, the loads in the storage will be described in the same way as this group with the safety factor for load specified for the group.

If a whole loading is considered as a group, it can be saved both as a loading and as a group.

It is necessary to clear the buffer memory from the loads with the help of the **Remove all loads** operation — before creating a *new* loading or a group of loads. If it turns out that the buffer memory contains loads which are not saved as a loading or a group, the respective warning with a suggestion to save the loads or reject their saving is generated.

Specifying Loads Using Groups of Nodes and Elements

Groups of nodes and elements can be used efficiently to specify loads. If when preparing the design model the same loads are applied to some elements in different loadings, it is convenient to combine these elements into groups. Before specifying the load you have to select the necessary group, specify the parameters of the load and click the **OK** button in the **Loadings** section of the toolbar.

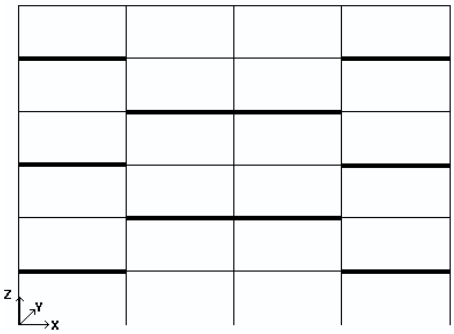


Figure 8.4-2. *Design model* (selected elements are shown with a thick line)

Let’s consider the design model given in Fig. 8.4-2 as an example. If in the process of describing loads on the selected elements (they are shown with a thick line in the figure) load of the same type is specified in different loadings, it makes sense to combine these elements into a group. All the further work will be performed as follows:

- select the necessary group from the **Select a group of elements** list (the **Groups** section of the toolbar) before specifying the loads;
- specify the type, direction and value of the load in the **Loadings** section;
- apply the load to the elements of the group.

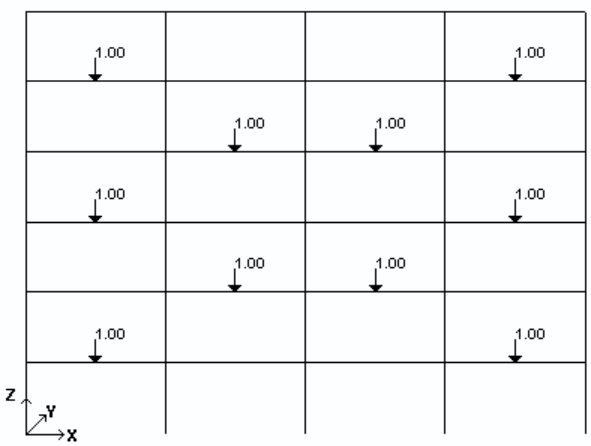


Figure 8.4-3. A concentrated force is applied to all the elements of the group

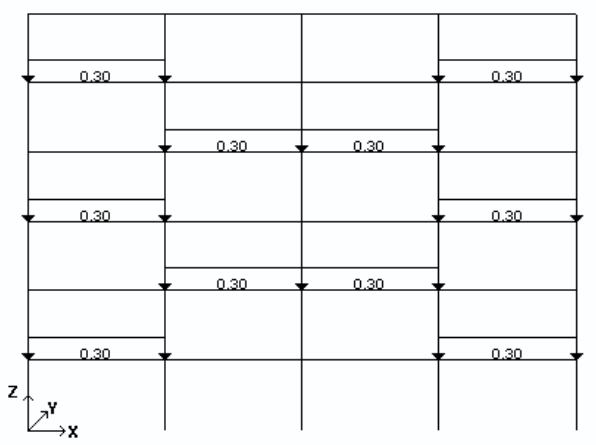


Figure 8.4-4. A distributed load is applied to all the elements of the group

In the result the specified load will be applied to all the elements, for example, the concentrated force 1 T in the Z direction (Fig. 8.4-3). Repeat the above actions for the same group applying the distributed load 0.3 T to the elements (Fig. 8.4-4).

Thus if the same elements are used when specifying loads in one or several loadings, it is convenient to combine such elements into named groups. You can work with groups of nodes in a similar way.

Generating Loadings from Groups of Loads

The operation is used to generate loadings from groups of loads. A group can be included into the existing loading and a new loading can be generated from groups. A dialog box with the respective name appears after selecting the type of operation in the drop-down menu.

The **Include a group into loadings** operation enables to add a group of loads with the given factor to different loadings (Fig. 8.4-5). In order to perform this operation you have to:

- select a group from the **Groups of loads** list which is included into one or several loadings;
- in the table check the checkboxes of loadings the selected group is included into;
- specify the factors with which the group is included in the loadings;
- click the **OK** button.

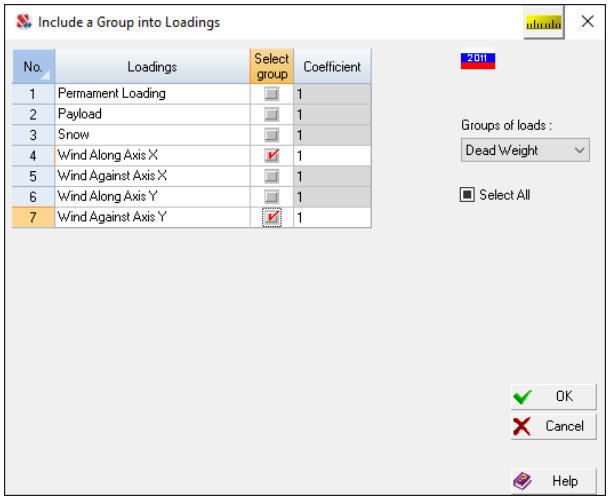
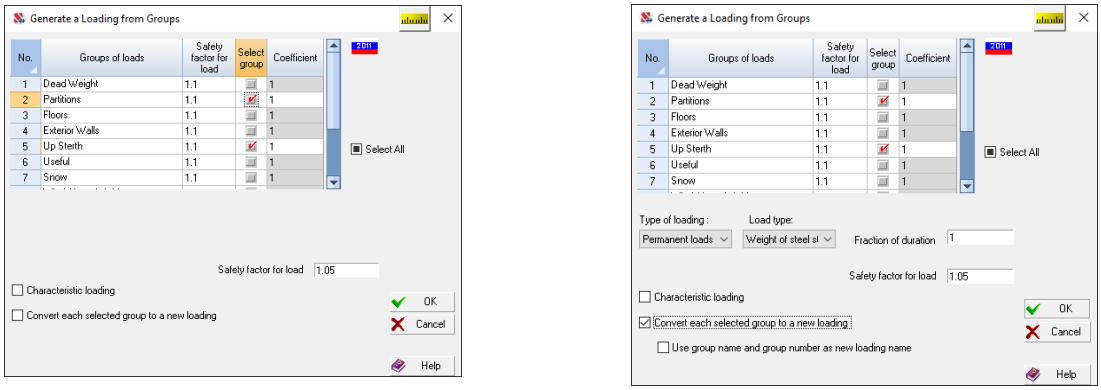


Figure 8.4-5. *The* **Include a Group into Loadings** *dialog box*



a *b*

Figure 8.4-6. *The* **Generate a Loading from Groups** *dialog box*

The **Generate a loading from groups** operation (Fig. 8.4-6 *a*) enables to generate a new loading from the groups of loads. Each group can be included in the loading with its factor. In order to perform this operation you have to:

- in the table check the checkboxes of groups the loading is created from;

- specify the factors for these groups;
- check the **Characteristic loading** checkbox, if the loads included in it should be saved as characteristic values (if EN 1990 is selected as the standard for the calculation of load combinations, then this checkbox is not available);
- click the **OK** button.

After closing the dialog box it is necessary to save the new loading using the **Save/Add loading** button .

If it is necessary to generate loadings from each selected group of loads, then you should check the **Convert each selected group to a new loading** checkbox (Fig. 8.4-6 *b*). Obviously, for the correct execution of this operation, the groups must have certain properties, namely: the loads combined into groups must belong to the same type of loading, have the same fraction of duration and safety factor for load. These parameters are selected from the respective lists. To make the name of the loading different from the name of the group, check the **Use the group name and group number as the name of the new loading** checkbox. For example, the group of loads "Floors", which is the third in the list of groups (Fig. 8.4-6 *b*), will be converted to a loading "3_Floors".

Specifying Factors for Groups of Loads

This operation enables to specify the factors for groups which are added to the loadings in a "traditional" way, i.e. by selecting them from the list in the toolbar. You can select the order of specification in the **Specify the Scale Factors** dialog box (Fig. 8.4-7) — use the specified value of the factor for all included groups or ask for the factor for each group. In the second case after selecting the group from the list the **Load Scale Factor** dialog box appears (Fig. 8.4-8), where you have to specify the factor for the selected group.

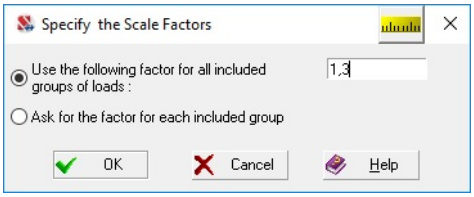


Figure 8.4-7. *The* **Specify the Scale Factors** *dialog box*

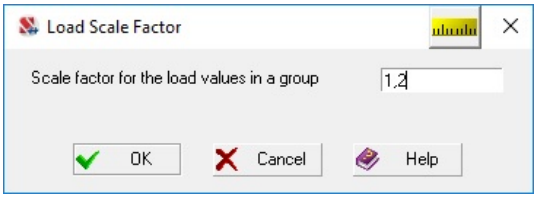


Figure 8.4-8. *The* **Load Scale Factor** *dialog box*

8.5. Deleting Loads, Loadings and Groups of Loads

Deleting Loads

The control of deleting the loads from the current loading is performed with the help of the **Delete Loads** dialog box (Fig. 8.5-1), which is invoked by clicking the respective button in the **Loadings** section of the toolbar. It is necessary to perform the following actions to delete all loads from the selected nodes or elements:

Figure 8.5-1. *The* **Delete Loads** *dialog box*

- select the option with the name of the performed operation **All loads** in one of the groups **Delete nodal loads from the selected nodes** or **Delete loads from the selected elements** in the dialog box;
- click the **OK** button (the dialog box will close);
- select elements or nodes in the model the loads are deleted from;
- click the **OK** button in the **Loadings** section.

Loads can be deleted selectively by specifying their type and direction:

- select the option with the name of the performed operation **Selected loads** and **By the type, direction** depending on the type of loads (nodal or on elements);
- specify the parameters of the deleted load with the help of the buttons or checkboxes;
- click the **OK** button (the dialog box will close);
- select elements or nodes in the model the loads are deleted from;
- click the **OK** button in the **Loadings** section.

You can also delete loads of a certain value:

- select the option with the name of the performed operation **Selected loads** and **By the value** depending on the type of loads (nodal or on elements);
- indicate the values of the deleted loads in the table;
- click the **OK** button (the dialog box will close);
- select elements or nodes in the model the loads are deleted from;
- click the **OK** button in the **Loadings** section.

There is one more option for deleting loads implemented in the mode of obtaining information on the node or element (filter buttons and , respectively). In this case loads can be deleted from a particular object.

Deleting Loadings and Groups of Loads

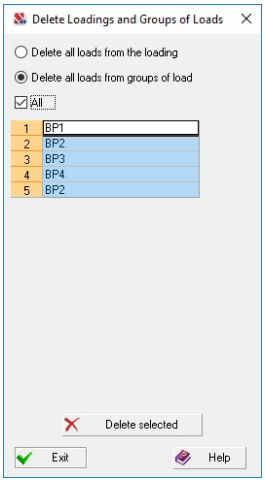


Figure 8.5-2. *The* **Delete Loadings and Groups of Loads** *dialog box*

This operation enables to delete one or several loadings and groups of loads. Operations of deleting are performed in the respective dialog box (Fig. 8.5-2). Its list will contain the names of loadings or groups of loads depending on the selected radio button.

In order to perform the operation select the items you want to delete in the list and click the **Delete selected** button. All the loads specified in the loadings you are deleting are removed from them, but the loadings technically remain. This is due to the possible references to these loadings in the initial data for the calculation of design combinations of forces, combinations of loadings, etc.

Pack Loadings

This operation enables to delete all the loadings which do not have any loads from the project.

After packing the loadings it is necessary to check the initial data for the calculation of DCF and combinations of loadings (Chapters 18 and 17), since the logical connections between loadings may be disrupted.

8.6. Preparing Data for the Analysis for Dynamic Actions

Unlike the analysis for static actions, the initial data for the analysis for dynamic actions includes the characteristics of the selected action, and the description of dynamic loads that generate vibrations of the system.

Dynamic loadings must take inertial forces into account. These forces are associated with the nodal masses and the masses located on the elements of the system. Masses can be specified explicitly similarly to static loads, or based on the specified static loadings. All loads of such loadings (acting in the direction of the Z axis of the global coordinate system) taking into account the coefficients specified for each loading are interpreted as the weights of masses reduced to the nodes of the system. Different methods of describing masses can be used simultaneously in one dynamic loading. Thus, three types of data necessary to perform the analysis can be used for dynamic loadings: characteristics of the loading, description of the load and description of masses. The load is not specified for some types of dynamic actions, for example seismic ones. However, the respective loads have to be specified for actions of the pulse, impact and some other types.

When analyzing the systems different inertial characteristics can be used in different loadings (for example, you can study the motion of the system taking into account temporary loads and the motion of the same system without the temporary loads).

The software implements the analysis for the following types of dynamic actions :

– *seismic actions in accordance with recommendations of the following codes:*

- SNiP II-7-81* — Russian Federation;
- SNiP II-7-81* — with modifications from 01.01.2000 Russian Federation;
- SP 31-114-2004 — Russian Federation;
- SP 14.13330.2011 — Russian Federation;
- SP 14.13330.2014 — Russian Federation;

- SP 14.13330.2018 — Russian Federation;
- SP 14.13330.2018 with Change No. 1 — Russian Federation;
- MGSN 4.04-94 — Moscow, Russian Federation;
- NP 031-01 — (design codes for earthquake-resistant NPP) Russian Federation;
- DBN B.1.1.-12-2006 — Ukraine;
- DBN B.1.1.-12-2014 — Ukraine;
- SNiP B.1.2-4-98 — Republic of Kazakhstan;
- SNiP RK 2.03-30-2006 — Republic of Kazakhstan;
- SP RK 2.03-30-2017 — Republic of Kazakhstan;
- NTP RK 08-01.1-2017 — Republic of Kazakhstan;
- SNRA II-2.02.94 — Republic of Armenia;
- SNRA II-6.02.2006 — Republic of Armenia;
- SNT 2.01.08-99 — Turkmenistan;
- Az. DTN 2.3-1 — Republic of Azerbaijan;
- KMK 2.01.03-96 — Republic of Uzbekistan;
- KMK 2.01.03-19 — Republic of Uzbekistan;
- SNiP KR 20-02:2009 — Kyrgyz Republic;
- GNiP RN 22-07-2015 — Tajikistan Republic;
- GNiP RN 22-07-2018 — Tajikistan Republic;
- SP 358.1325800.2017— Russian Federation;
- EN1998 — European Union;
- codes of CIS (project);

− *seismic actions specified by accelerograms or plots of the dynamic amplification factor:*

- single-component accelerogram;
- three-component accelerogram;
- six-component action;

− *wind actions in accordance with recommendations of the following codes:*

- SNiP 2.01.07-85* — Russian Federation;
- SNiP 2.01.07-85* (SP 20.13330.2011) — Russian Federation;
- MGSN 4.04-94 — Moscow, Russian Federation;

− *other actions:*

- pulse;
- pulse with history tracking;
- impact;
- impact with history tracking;
- impact accounting for the effect of the mass of the impact body;
- harmonic oscillations taking into account the starting resonance;
- harmonic oscillations (analysis for the specified frequency);
- harmonic oscillations (taking into account the possible error in the determination of natural frequencies);
- modal analysis;
- direct integration of the equations of motion.

The considered direction of the action of inertial forces (direction of nodal masses) must correspond to the dynamic translational degrees of freedom defined by the boundary conditions and the index of the model type. It should be noted that some inertial forces can be neglected, therefore only the "directions of masses" which should be considered in the calculation are specified. Masses on the elements are specified in the same way as the distributed and concentrated loads.

The following procedure is recommended to prepare the dynamic loadings:

− invoke the **Dynamic actions** operation in the **Loadings** toolbar and select the **Create a new loading** item in the drop-down menu;

- the **Parameters of Dynamic Actions** dialog box will appear where you have to specify the type and parameters of the action and close the dialog box by clicking the **OK** button, and the new loading will be automatically created;
- if it is necessary to specify the dynamic loads for the analysis of the action (for example, for the impact action), invoke the **Parameters of dynamic loads** operation, specify the parameters of the load corresponding to the dynamic action in the respective dialog box and after closing this dialog box specify the places of the load application in the design model (it is necessary to save the loading after performing the above actions);
- in the case of "manual" specification of masses their description is performed in the **Masses** dialog box which appears after invoking the **Inertial properties** operation; after closing the dialog box it is necessary to snap the masses to nodes and/or elements of the model.

Table 8.6-1 below provides a list of currently implemented types of dynamic actions. The table shows the *Module Number* with the internal identification of the type of action. In some cases, this internal number may appear in diagnostic messages. The last column provides the force summation rules and the symbols are defined in Table 8.6-2. Several rules are specified for some types of actions, which can be used depending on the options selected by the user.

Table 8.6-1. Types of dynamic actions

Dynamic actions	Module number	Code	Name/Purpose	Summing rules
Seismic actions	20	SNiP II-7-81*	Construction in Seismic Regions	SRSS
	50	Project for CIS	Construction in Seismic Regions	
	40	SNiP II.7-81* (01.01.2000)	Construction in Seismic Regions	
	54	SP 31-114-2004	Rules for the design of residential and public buildings for the construction in seismic regions	SRSS M10%
	57	SP 14.13330.2011	Construction in Seismic Regions (Updated edition of SNiP II-7-81*)	
	61	SP 14.13330.2014	Construction in Seismic Regions (Updated edition of SNiP II-7-81*)	
	68	SP 14.13330.2018	Construction in Seismic Regions (Updated edition of SNiP II-7-81*)	
	56	MGSN 4.19.05	MGSN	SRSS
	51	NP-031-01	Design codes for earthquake-resistant nuclear plants	SRSS M10%
	53	DBN B.1.1-12:2006	Construction in Seismic Regions of Ukraine	CQC
	63	DBN B.1.1-12:2014	Construction in Seismic Regions of Ukraine	
	30	SNiP B.1.2-4-98 (Republic of Kazakhstan)	Construction in Seismic Regions	SRSS
	55	SNiP RK 2.03-30-2006 (Republic of Kazakhstan)	Construction in Seismic Regions. Design codes	
	67	SP RK 2.03-30-2017, SP RK 2.03-30-2017* (Republic of Kazakhstan)	Construction in Seismic Regions. Design codes	
	52	SNRA II-2.02.94	Seismic analysis. Building codes of the Republic of Armenia	CQC
	59	SNRA II-6.02.2006	Seismic analysis. Building codes of the Republic of Armenia	
	60	AzDTN 2.3-1 (Republic of Azerbaijan)	Construction in Seismic Regions	SRSS
	62	SNT 2.01.08-99 (Turkmenistan)	Construction in Seismic Regions	M10%
	64	KMK 2.01.03-96 (Uzbekistan)	Construction in Seismic Regions	
	65	SNiP KR 20-02:2009 (Kyrgyzstan)	Earthquake engineering	SRSS
	66	CHT 2.01.08-99* (Turkmenistan)	Construction in Seismic Regions	M10%

Dynamic actions	Module number	Code	Name/Purpose	Summing rules
	69	GNiP 22-07-2015 GNiP 22-07-2018 (Tajikistan)	Earthquake engineering	SRSS
	70	SP 358.1325800.2017	Hydraulic structures	SRSS
	71	EN1998	Design of structures for earthquake resistance	DSM+SRSS DSM+Newmark(0.3) DSM+Newmark(0.4) SRSS+Newmark(0.3) SRSS+Newmark(0.4) CQC+SRSS CQC+Newmark(0.3) CQC+Newmark(0.4) SRSS
	72	SP 14.13330.2018 with change No. 1	Construction in Seismic Regions (Updated edition of SNiP II-7-81*)	SRSS CQC
	73	NTP RK 08-01.1-2017	Construction in Seismic Regions	SRSS
	74	KMK 2.01.03-19 (Uzbekistan)	Construction in Seismic Regions	M10%
	27		Seismic actions according to the accelerogram	ENV
	47		Seismic actions according to the accelerograms (3 components)	
	58		Seismic Analysis Based on the Six-Component Action	SRSS M10% CQC
Wind pulsations	21	SNiP 2.01.07-85*	Loads and Actions	SRSS
	31	MGSN 4.19-05	Multifunctional high-rise buildings and complexes	CQC
	32	SP 20.13330.2011, SP 20.13330.2016	Loads and Actions (Updated edition of SNiP 2.01.07-85*)	SRSS
Pulse	22		Pulse	SD
	42		Pulse with history tracking	
Impact	23		Impact	
	43		Impact with history tracking	
	33		Impact accounting for the effect of the mass of the impact body	
Harmonic oscillations	24		Analysis taking into account the starting resonance	SRRI
	28		Analysis for the specified frequency	
	34		Analysis for the specified frequency taking into account the possible error in the determination of natural frequencies	
Modal analysis	100		Modal analysis	
Direct integration of the equations of motion	101		Direct integration of the equations of motion	ENV

Table 8.6-2. Summing rules

Notation of the summation method	Comments
SRSS	square root of the sum of the squares of the components
CQC	complete quadratic combination method [21]
M10%	summation allowing for the proximity of frequencies (rule 10%)
SRRI	square root of the sum of the squares of the real and imaginary components
SD	summation allowing for damping
ENV	envelope (in time) of the calculated values
DSM	double sum method [21]

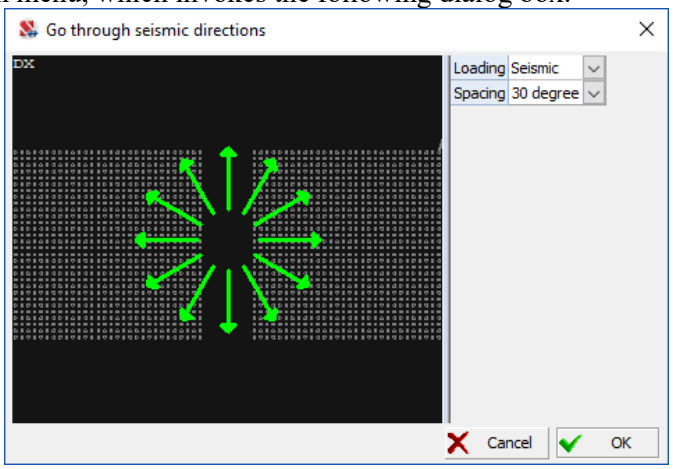
GRP	grouping method [21]
Newmark(λ)	±X±λY±λZ ±λX±Y±λZ ±λX±λY±Z

8.7. Specifying the Parameters of Dynamic Actions

The parameters of dynamic actions can be specified with the help of the **Dynamic actions** operation, which is invoked from the **Loadings** toolbar by clicking the respective button . If the first dynamic loading is created, the **Parameters of Dynamic Actions** dialog box appears where you have to specify the parameters. Otherwise a drop-down menu appears which includes two sections. The first section — **Create a new loading** — is used in cases when a new dynamic loading is created, and the second one — **Correct the existing loading** — to modify the existing dynamic loadings.

It is stated in seismic codes that seismic actions can have any direction, and the most dangerous direction can be predicted only in the simplest cases. Various methods have been proposed to search for the most dangerous direction. For example, it was assumed that the direction with the maximum potential strain energy is the dangerous one. It is easy to see that the system can have structural elements accumulating a relatively small amount of strain energy, and a different seismic direction will be dangerous for them.

Therefore, **SCAD** can automatically go through the possible directions with a given angle step. If a seismic loading has been created, the **Go through seismic directions** option will be available in the drop-down menu, which invokes the following dialog box:



You can select a seismic loading (if there is more than one) and an angle step in this dialog box. A number of mutually exclusive seismic loadings with different horizontal directions will be created after clicking the **OK** button. The analysis of design combinations of forces will then find the directions dangerous for different elements and parts of the structure.

Creating a New Loading

After selecting this section in the menu the two-tab **Parameters of Dynamic Actions** dialog box appears (Fig. 8.7-1). You can specify the following data in its first tab:

- type of action;
- name of the loading;
- indicator of conversion of static loads into masses and a set of parameters of conversion;
- variants of implementation of the selected type of action (table);
- method of determining the natural modes and frequencies.

The name and contents of the second tab depend on the selected type of action.

The following procedure is recommended to specify the parameters of a dynamic action:

– specify the name of the loading in the respective text field;
– select the type of action with the help of the radio buttons;

- if you want to convert static loads into masses, check the respective checkbox and perform the following actions:
  - select the number (name) of the static loading from the drop-down list;
  - specify the value of the factor its loads will be multiplied by when generating the masses in the **Conversion factor** field;
  - click the **Save** button;
  - repeat the above actions for other loadings involved in the generation of masses;
  - in order to delete a loading from the generated list you just have to select a line with this loading in the list and click the **Delete** button;
- select a variant of implementation of the specified dynamic action in the table by clicking on the respective line;
- specify the method for determining the natural modes and frequencies with the help of the radio buttons (if the Lanczos method is selected, you have to specify additional parameters necessary to use this method);
- switch to the second tab of the dialog box (after selecting the type of dynamic action the second tab of the dialog box becomes accessible, where you can specify the parameters of this action) and specify the parameters necessary for the calculation of the selected variant of implementation of the action;
- close the dialog box by clicking the **OK** button.

Static loads are converted into masses according to the following rules:

- all local loads on the elements are converted into nodal loads;
- loads acting in the direction of the Z axis are left from the nodal forces thus obtained (they are considered as masses for all linear degrees of freedom);
- sign of the calculated load is ignored.

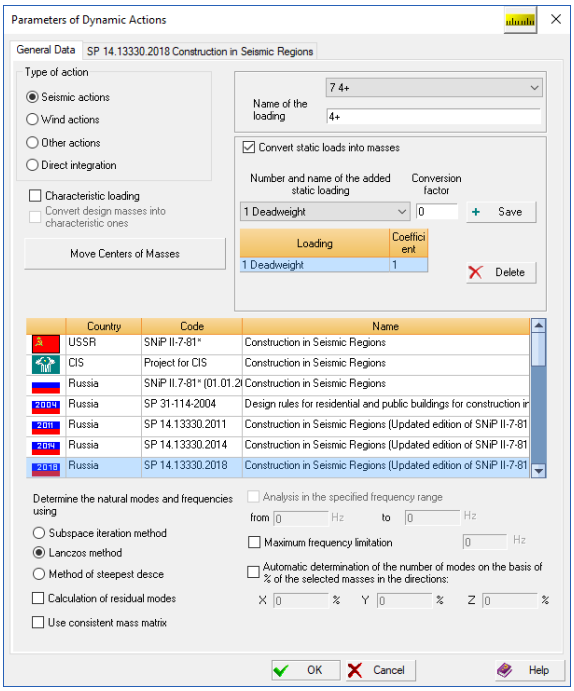


Figure 8.7-1. *The* **Parameters of Dynamic Actions** *dialog box*

It should be noted that the division of the weight of mass (static loading) by the gravitational acceleration is performed automatically and the conversion factor does not play any role. It is used for other purposes, for example to specify the design values of static loads for the calculation of characteristic values of masses.

The **Characterisric load** checkbox enables to indicate that the parameters of the current dynamic action (for example, the nodal masses) are specified not by the design values but by the characteristic ones. The **Convert design masses into characteristic ones** checkbox duplicates the information which is

specified in the 11.7 and is given here for information purposes (if EN 1990 is selected for calculating load combinations, this control is not available).

Seismic analysis in **SCAD** enables to determine the inertial characteristics of the system by recalculating the static loads. You should take into account the following:

a) Not every static load has a mass. For example, the static pressure of gas should not be recalculated into a mass since it contradicts the physical meaning of such a loading (it does not create an inertial force when an acceleration appears in the system).
b) If the value of the load from the self-weight is determined automatically, only the weight of the bearing frame is taken into account. However even in this case it might be necessary to use an enlarged value obtained in the manual calculation by multiplying by the structural weight factor (weight of node connections, weight of the fire protection, etc.). In order to do this, you have to specify this factor.
c) Seismic SNiP (see, for example, Table 2 of SP 14.13330.2011) considers that the design combination will include not the full values of some loads, but multiplied by the combination factors:
   - weight of structures — 0,90;
   - values of temporary long-term loads — 0,80;
   - values of short-term loads — 0,50.

Applying these factors to permanent loads is beyond question. For if we believe that the reduced values of loads are considered (the characteristic value instead of the design one is used in practice), the inertial component should naturally be considered in the same way.

Temporary loads should, however, be converted into masses more carefully. The thing is that when selecting DCF after having created the respective mass and allowing thus for the inertial component of this mass, such load might not be included in the combination. Therefore when the value of the temporary load is significant, it is necessary to create several mutually exclusive variants of the seismic analysis. For example, the variant of an “empty silo” without the load from the backfill, and the variant of a “filled silo”.

It should be noted that neglecting a load when converting it into a mass leads to the decrease in the periods of the natural oscillations, which in turn increases the spectral dynamic amplification factor β for most structures with a period of natural oscillations greater than 0,1 sec, and this circumstance increases the reliability.

If most parameters of the new loading coincide with those of the loading specified earlier, you can use the **Take the values of the parameters from the loading** operation. In this case the new loading will get the parameters of the dynamic loading selected from the list, and then the necessary parameters can be changed.

Natural modes and frequencies can be determined using different methods: subspace iteration method, method of steepest descent, or Lanczos method. A method can be selected using the respective radio buttons. The Lanczos method enables depending on the type of the problem solved to specify the range of frequencies the analysis is performed in, or to use the mode of an automatic determination of the necessary number of modes of natural oscillations depending on the specified percent of the selection of modal masses for each direction (for the analysis for seismic actions). It should be noted that the Lanczos method can be used only together with a multifrontal or PARFES processor.

The selection of the percent of the modal mass is defined by the following considerations. If the structure was infinitely rigid, it would be easy to calculate the value of the kinetic energy $K=a^2M/2$ arising in the process of movement of its mass M under the action of the specified ground acceleration $\boldsymbol{a}$. If you determine the potential energy for each mode of natural oscillations, the sum of such energies must be equal to the value of K, and for an incomplete set of natural modes — to a certain fraction of K. Design codes of some countries (USA, France) recommend to specify the fraction of the selected modal mass as 90% for horizontal and 75% — for vertical direction. These values are taken by default. The values of the frequency range or the percent of modal masses are specified in the text fields which become accessible after checking the respective checkboxes.

If the Lanczos method is used, the software additionally provides an automatic selection of the number of natural oscillation modes based on the restrictions on frequency (all modes the frequencies of which lie in the range from zero to the number specified by the user will be selected). If both the restrictions on frequency and on the percent of modal masses are specified, both conditions will be used.

If the method of steepest descent or Lanczos method is selected, you can use either a diagonal or consistent mass matrix (**Use consistent mass matrix** checkbox).

Correcting Parameters of the Existing Loading

If the **Correct the existing loading** item is selected in the menu of the **Dynamic actions** operation, the **Parameters of Dynamic Actions** dialog box will additionally contain a drop-down list with dynamic loadings specified earlier (Fig. 8.7-2). After selecting a loading you want to correct from the list its parameters are restored automatically. You can modify several loadings successively selecting them from the list.

All the input information is checked and recorded after closing the dialog box by clicking the **OK** button. If the control detects errors in the specified parameters, a warning window appears.

If you use the OK button to close the dialog box in the correction mode, regardless of whether changes have been made to the data or not, the project will be considered as modified and a recalculation of the problem will be required.

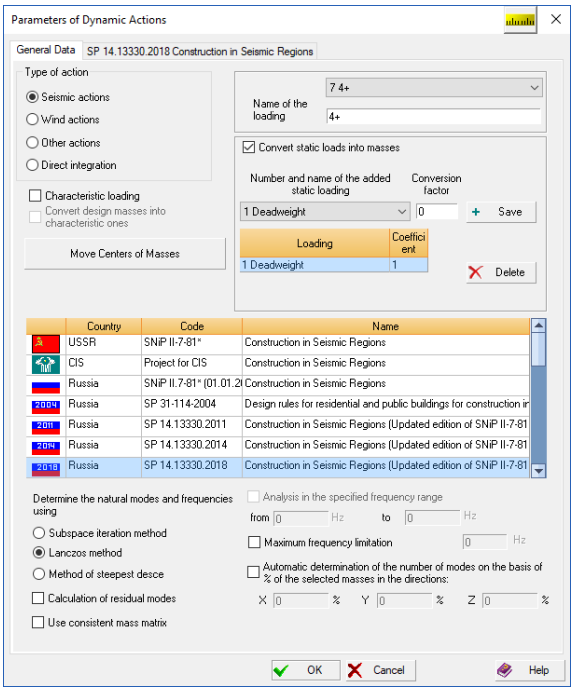


Figure 8.7-2. *The* **Parameters of Dynamic Actions** *dialog box* (the operation of correcting)

Peculiarities of Specifying Data on Seismic Actions

Correction factor is specified in the data of all seismic loadings for the correction of initial data if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it. The following cases of using the value of the correction factor different from one can be considered as examples:

- the soil category requires to change the seismicity of the site (for example, decrease it by one magnitude), which makes it necessary to specify magnitude 6 (see, for example, Table 1 of SP 14.13330.2014). Taking into account that the increase of seismicity by one magnitude doubles the result, you can specify magnitude 7 for the site and take the correction factor as 0,5;

- it is necessary to perform the check calculation of the actual existing structure for the impact of the earthquake with an intensity of 8,5. It is sufficient to specify magnitude 8 for the site and take the correction factor as $2^{0,5}$ = 1,414, since the value of the ground acceleration is calculated by the formula $K*2^N$ and $(K*2^{8,5})/(K*2^8)$ = 1,414.

Requirement for considering the seismic torque in the updated edition of Russian codes "Construction in Seismic Regions" (SP 14.13330.2014) is as follows:

> *"5.16 When calculating buildings and structures with the length or width greater than 30 m using a cantilever DDM apart from the seismic load determined according to 5.5, it is necessary to take into account torque about the vertical axis of the building or structure passing through its shear center. The value of the design eccentricity between the shear center and the center of mass of buildings or structures in the considered level should be not less than 0,1B, where B is the size of the building or structure in plan in the direction perpendicular to the force action S_k".*

There are similar requirements in the codes of Ukraine — DBN B.1.1-12:2006, Kazakhstan — RK 2.03-30-2006 and a number of other countries.

A design dynamic model (DDM) of plane or spatial types are usually used in **SCAD** and in this case the above recommendation concerning a cantilever DDM is not relevant. However, a calculation for the conditional torque may be required in cases when a regular structural model experiences only translational displacements under the seismic action.

A typical example is a spatial frame with rows of identical transverse frames which bend simultaneously and all discs of floors receive linear displacements.

If the considered object is like that and there is no torsion of floor discs, it is recommended to artificially introduce the asymmetry into the model. This can be done by taking the self-weight (and hence the mass) in one half of the building with a safety factor for load, and in the other — without this factor. Then the natural torsion will appear and there will be no need to use the recommendation of Sec. 5.15 of SNiP.

You can also move the centers of masses of storeys as described below

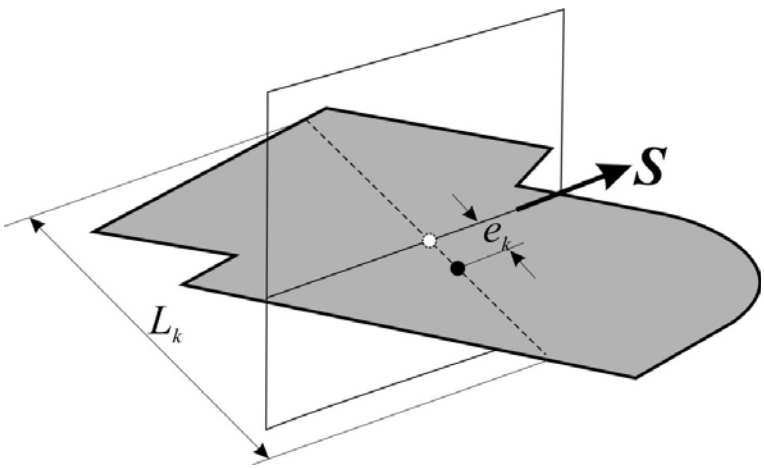
Move Centers of Masses

A real seismic action differs in that the soil accelerations under the structure are not strictly synchronous and this unevenness can cause twisting of the structure.

In order to take into account spatial changes in the soil movement and errors that can be made when determining the center of mass, Eurocode and the codes of the Republic of Kazakhstan suggest to move the design center of mass on each storey from its nominal position by the value of random eccentricity:

$$e_k = L_k h_{ek}, \quad h_{ek} = \pm 0{,}05 f_{ek}$$

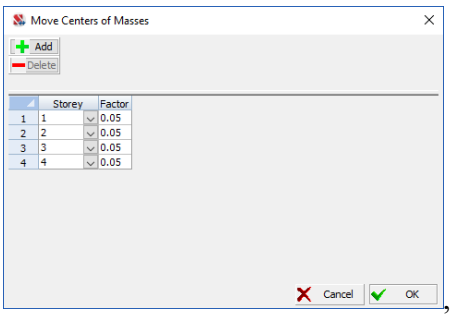
where L_k is the size of the floor perpendicular to the direction of the seismic load, f_{ek} – coefficient taking into account the building irregularity at the level of the k-th storey (it is specified in the codes of Kazakhstan).



To move the center of mass with respect to the vertical plane passing through the center of mass and parallel to the given direction of the seismic action ***S***, we use the technique of increasing the mass on one side and decreasing it on the other so that the total mass remains unchanged. Masses located at the level of the respective floor are considered for each storey, since it is here that the main loads with masses (dead weight and all live loads) are located.

If the direction of seismic action is specified, the size L_k is automatically determined as the distance between the most distant nodes with reduced masses. Therefore, the center of mass of a design model of a cantilever beam with a single mass on the storey L_k=0 will not be moved.

The respective button is used to specify the data for moving the centers of mass of the storeys (it is assumed that groups of elements corresponding to different storeys have previously been created). Clicking this button invokes the dialog box



, where using the **Add** and **Delete** buttons you can create a table with the data for moving the centers of mass. A group (storey) is selected from the list of available groups of elements in the first column of this table, and the coefficient h_{ek} described above has to be specified in the second column.

The centers of masses are moved only for the loadings with a specified direction of action.

Number of modes taken into account is specified in all dialog boxes. However, if the percents of modal masses which have to be reached for each direction are specified, and the Lanczos method is selected for calculating eigenvalues and natural modes of oscillations, *not less* than the specified number of modes will be determined.

Accounting for the vertical component of the seismic load is required for a number of cases of the seismic load. However, it can not be taken into account automatically, since there are no such concepts as a "horizontal or inclined cantilever structure", "bridge superstructure" or "... a span of more than 24 meters" in the description of the design model assumed in **SCAD** and many other programs which use the finite element method. These and other cases are calculated for the action of a vertical seismic load, if when inputting the initial data you specify its direction along the Z axis.

Design values of SSS, for example, in accordance with the formula (8) of SP 14.13330.2014, components of seismic response are calculated as the square root of the sum of squares of values corresponding to different modes of natural oscillations. In the result of this nonlinear transformation (Rosenblum estimate) displacements and internal forces do not correspond to each other and do not satisfy the conditions of equilibrium, since for different modes of natural oscillations they appear at different moments. This fact should be taken into account when analyzing the results of the calculation. The correspondence of these components is guaranteed only for each mode individually.

When calculating structures symmetrical in plan, displacements (and their corresponding forces) may appear not only in the direction of action of the seismic force, but also in the orthogonal direction, although based on the conditions of symmetry this should not happen. This occurs due to the implementation of codes, which can be illustrated by a simple example.

Example. Let a vertical cantilever with a mass at the top have oscillation modes (vectors are written in a line) $f_1(1)$={1; 0} and $f_2(1)$={0; 1}. Then at the direction cosines {1; 0} the first mode will be fully taken into account, and the second mode will not work, and the answer in some dimensionless units will be {1; 0}. However, if the same pair of multiple modes is determined as $f_1(2)$={0,707; 0,707} and $f_2(2)$ ={0,707; -0,707}, we will get the answer {1; 1}. Components with opposite signs will not annihilate each other because the squares (!) of components are summed and then the square root of the sum is determined.

It only remains to note that for multiple eigenvalues characteristic of symmetrical structures, when all directions are equal, the solution $f_1(1)$, $f_2(1)$ and the solution $f_1(2)$, $f_2(2)$ are completely equal. Which of them will be selected by the program (any, not just SCAD) is determined by such uncontrollable details as rounding errors of floating point numbers.

The problem disappears when an accelerogram based analysis is performed, because there is no "square root from the sum of squares" operation.

Allowing for the proximity of frequencies. For example, according to formula (9) of SP 14.13330.2014 components of seismic response have to be calculated taking into account the mutual correlation of close frequencies. You can select the method of correlation when performing the analysis according to some codes if several variants are allowed.

Calculation of residual modes. This checkbox prompts the software to add the "residual modes" to the set of the natural oscillation modes, which will supplement the sum of modal masses to 100% in the respective directions. These modes are "assigned" a frequency equal to that of the last mode determined on the basis of a standard modal analysis. It should be noted that even if not a single eigenpair is determined, the residual modes will still provide 100% of modal masses (the following requirement is introduced in the codes: a certain percent of modal masses has to be taken into account "honestly", and only the remainder can be covered by the residual modes). It is assumed that there is no mutual correlation of the residual modes.

Plots of the dynamic amplification factor — some codes allow to use special forms of the plot (spectrum) of the dynamic amplification factor. The plot can be specified as a piecewise linear function and saved in a file with the extension **.spd**. Besides a single plot you can also specify up to six plots — separately for each direction of the action.

A format of the file with the description of the plot of the dynamic amplification factor is given below. A file with a similar format can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 33.10). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

```
Dynamic amplification factor spectrum
#
0.030  1.030  0.060  1.45  0.100  2.500  0.147  3.100
0.202  2.820  0.261  2.64  0.359  2.560  0.464  2.030
0.681  1.400  1.000  1.06  1.565  0.650  0.966  0.390
```

A comment goes before the # symbol followed by the description of vertices of a polyline that approximates the plot where the period in seconds and the value of its corresponding coefficient are consequently specified for each point.

Accelerogram files (extension **.spc**) are used in the accelerogram based analyses. They must be in the same directory as the data of the problem. Several standard accelerograms come together with the complex. You can find them in the software directory. Here is an example of one of them, having the name 1m1-z.spc:

```
Design accelerogram in cm/(sec*sec) for MDE   on the site of a nuclear reactor.
     Component  -  Z.  Model - M1c. Amax =  42.2 cm/(sec*sec).
     Number of points N =  2047; Time step Dt =  0.05000 sec.
#
0.01 2047 0.05
  0.0    0.0   -0.0   -0.3   -0.6   -0.4    0.8    1.8    1.9    0.3
 -1.6   -2.5   -3.5   -2.7   -1.7   -1.3    2.0    3.6    0.1   -0.6 ...
```

Ordinates of accelerograms can be specified in any units of measurement. The first number after the # symbol is a conversion factor for the used units. The specified values are multiplied by it to obtain data in m/sec^2. In this case this factor is equal to 0,01. It is followed by the number of points (2047), the time step (0.05 sec) and consequent values of ordinates. A period is used as a separator between the integer and fractional parts. The tab enables to specify the scale factor for the ordinates of the accelerogram.

To facilitate the preparation of accelerograms **SCAD Office** includes a special program, **Accelerogram Editor**, described in Section 33.8. If a file with an accelerogram is selected, the button enables to invoke **Accelerogram Editor** to view the accelerogram and its parameters.

Seismic Analysis according to SNiP II-7-81*

Initial data are specified in the **Seismic Action (SNiP II-7-81*)** tab (Fig. 8.7-3) according to SNiP II-7-81* [9, 10]. Without describing each control of this tab in detail we will focus on the issues which require additional explanation. When preparing the initial data it is necessary to pay attention to the following factors:

Seismicity is specified without taking into account the soil category. The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor.

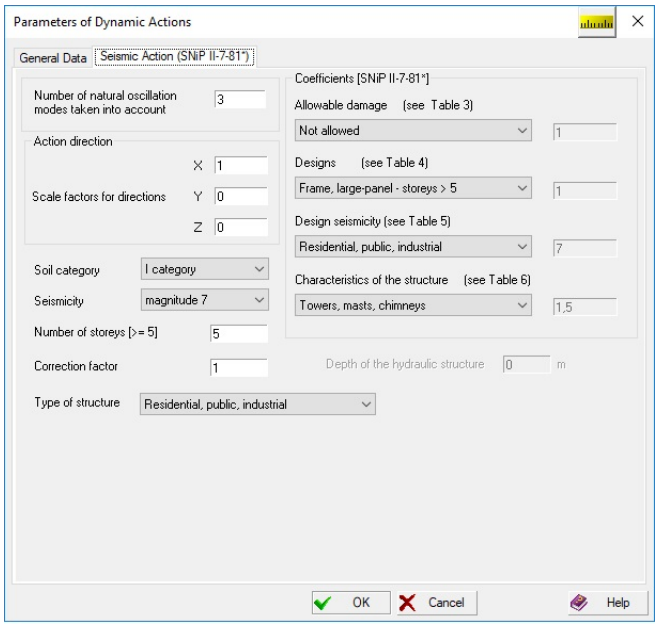


Figure 8.7-3. *The* **Seismic Action (SNiP II-7-81*)** *tab*

Correction factor is specified for the correction of the initial data if it is necessary to take into account the requirements of SNiP more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Action direction is specified by the scale factors with respect to the axes of the global coordinate system.

Coefficients allowing for the allowable damage, designs, design seismicity and characteristics of the structure are specified by selecting them from the lists in accordance with the sections of codes indicated beside the lists.

Seismic Analysis according to SNiP II-7-81* with Modifications from 27.12.99

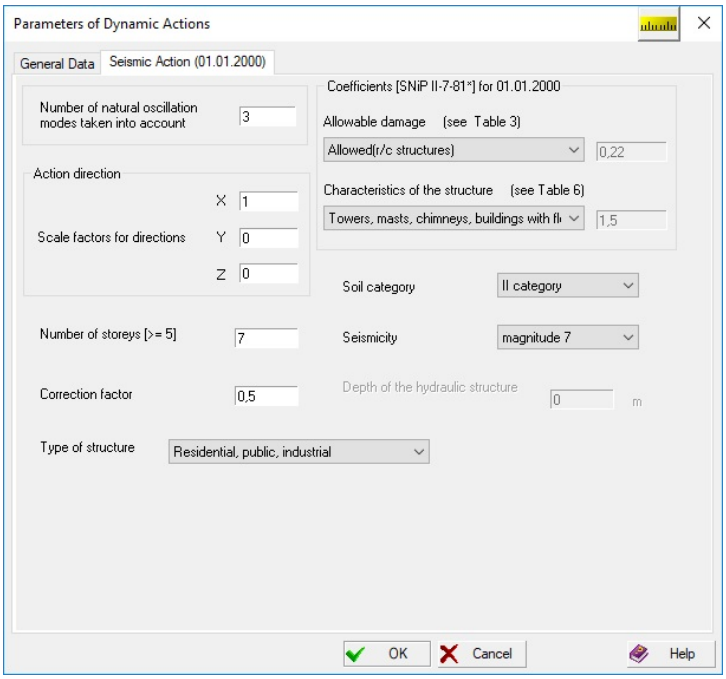


Figure 8.7-4. *The* **Seismic Action** *tab*
(edition of SNiP II-7-81* before 01.01.2000)

When preparing the initial data it is necessary to pay attention to the following factors:

Seismicity — is specified without taking into account the soil category. The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor.

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of SNiP more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Action direction is specified by the scale factors with respect to the axes of the global coordinate system.

Coefficients allowing for the allowable damage and characteristics of the structure are specified by selecting them from the lists in accordance with the sections of codes indicated beside the lists.

The software implements SNiP II-7-81* "Construction in Seismic Regions" taking into account all five changes made by the regulations of the USSR and Russian State Committee for Construction from 27.12.99. A module which enables to perform the analysis according to the edition of SNiP before 01.01.2000 is saved to support the projects the analysis of which had been performed before the changes were made.

It should be noted that the change No.5 was fundamental. It introduced a set of new seismic zoning maps of Russia, which implement different probabilities of design earthquake. Thus, different seismic zoning maps are used for the design of buildings and structures of different importance.

Therefore, the coefficient K_2 is excluded from the main design formula (1) of SNiP II-7-81* which dramatically reduced the number of design gradations of the types of buildings and structures (taking into account that this selection is implemented after the selection of a zoning map). Moreover, the plots of the dynamic amplification factor are changed significantly in the direction of increasing values in almost the entire real range of periods of natural oscillations. Therefore, a direct comparison of the results of the calculations performed earlier using only the initial data used in the analysis according to SNiP before applying the change No.5 is impossible. You have to specify the entire set of new initial data taking into account the possible change of the seismic region.

Seismic Analysis according to SP 31-114-2004 of the Russian Federation

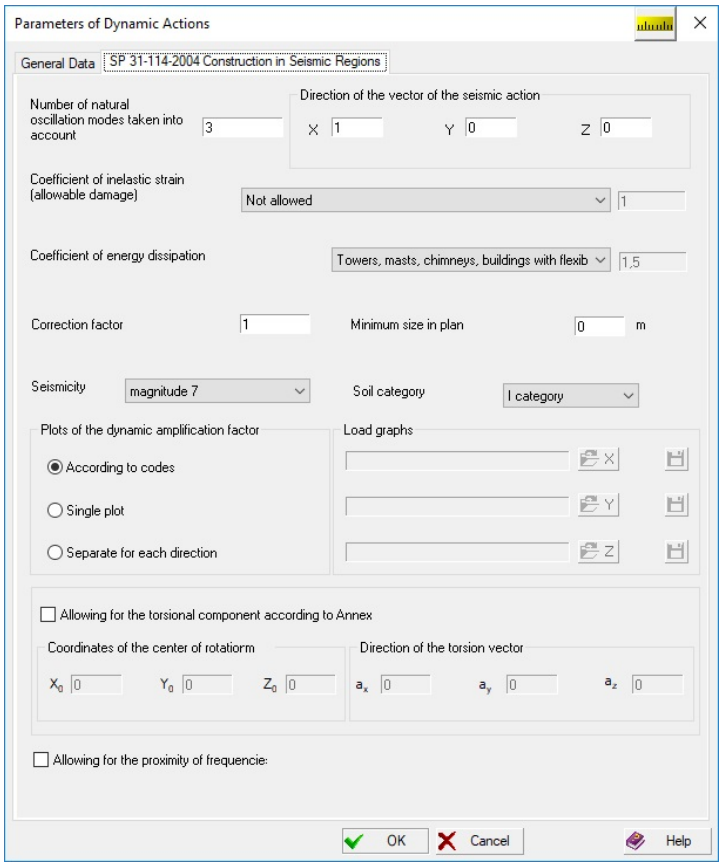


Figure 8.7-5. *The* **SP 31-114-2004 Construction in Seismic Regions** *tab*

When preparing the initial data it is necessary to pay attention to the following factors:

Seismicity is specified without taking into account the soil category. The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor.

Coefficient of inelastic strain accounts for the inelasticity of the structure and the allowable damage in it (is taken from Table 3 of SNiP II-7-81*).

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of SP more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Direction of the vector of the seismic action is defined by the vector specified in the global coordinate system.

Minimum size in plan is used to determine the coefficient taking into account the seismic wavelength spectrum (see Sec. B.2 of Annex B).

Allowing for the torsional component — if the torsional component is taken into account (the respective checkbox is checked), it is necessary to specify the coordinates of the center of rotation and the direction of the torsion vector.

Coordinates of the center of rotation — they should be taken as the coordinates of the center of gravity of the subgrade area of the building, considering the subgade uniform over the entire area.

Direction of the torsion vector is specified by the vector of rotational acceleration.

Allowing for the proximity of frequencies — if this checkbox is checked, the formula 6.4, which enables to take into account the mutual correlation of the generalized coordinates, is used for the neighbor natural oscillation modes which differ by less than 10%.

Plots of the dynamic amplification factor — the considered codes allow to use special forms of the plot (spectrum) of the dynamic amplification factor. The plot can be specified as a piecewise linear

function and saved in a file with the extension **.spd**. The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 33.10). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

Besides a single plot you can also specify three plots — separately for each direction of the action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single plot** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button).

Seismic Analysis according to SP 14.13330.2011 (Updated edition of SNiP II-7-81*) of the Russian Federation

When preparing the initial data it is necessary to pay attention to the following factors:

Number of natural oscillation modes taken into account — according to this code the minimum number of natural oscillation modes taken into account is specified in such a way so that "the sum of effective modal masses is not less than 90% of the total mass of the system excited in the direction of the seismic action for horizontal actions and not less than 75% — for the vertical one" (SP). The Lanczos method (only together with a multifrontal or PARFES processor) is used for the automatic determination of the necessary number of the natural oscillation modes.

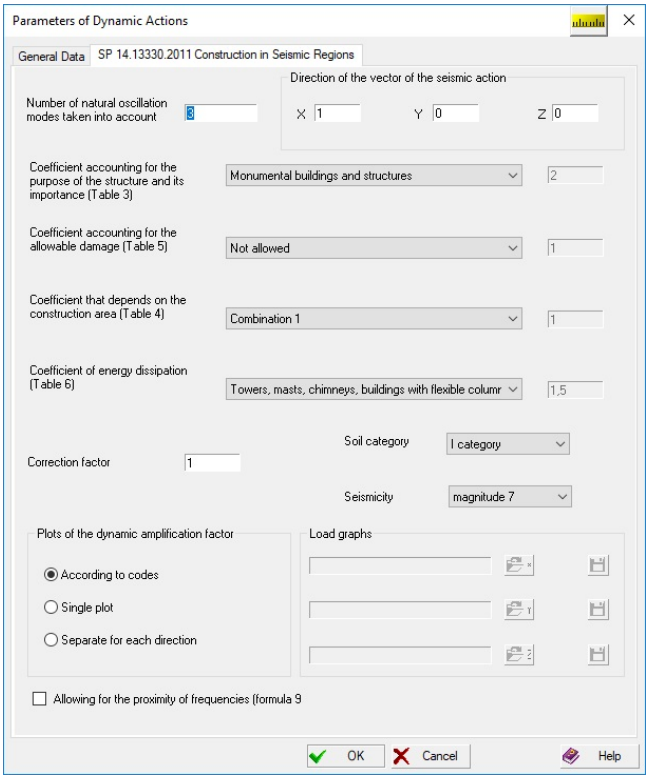


Figure 8.7-6. *The* **SP 14.13330.2011 Construction in Seismic Regions** *tab*

Seismicity is specified without taking into account the soil category. ***The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor***. The value of design seismicity for a certain terrain can be specified by selecting from the list or using the **Select a settlement** operation, which can be invoked by clicking the button on the right from the list. In the last case the dialog box appears with a list of settlements according to Annex A. After selecting the value of seismicity in the list and clicking the **OK** button the dialog box closes, and the selected value is displayed in the **Seismicity** list.

Coefficient of energy dissipation accounts for the inelasticity of the structure (is taken from Table 6).

Coefficient accounting for the purpose of the structure and its importance is selected according to Table 3.

Coefficient accounting for the allowable damage is selected according to Table 5.

Coefficient that depends on the construction area is selected according to Table 4.

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of SNiP more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Direction of the vector of the seismic action is defined by the vector specified in the global coordinate system.

Allowing for the proximity of frequencies — if this checkbox is checked, the formula (9), which enables to take into account the mutual correlation of the generalized coordinates, is used for the neighbor natural oscillation modes which differ by less than 10%.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single plot** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 33.10). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

Seismic Analysis according to MGSN 4.19-05

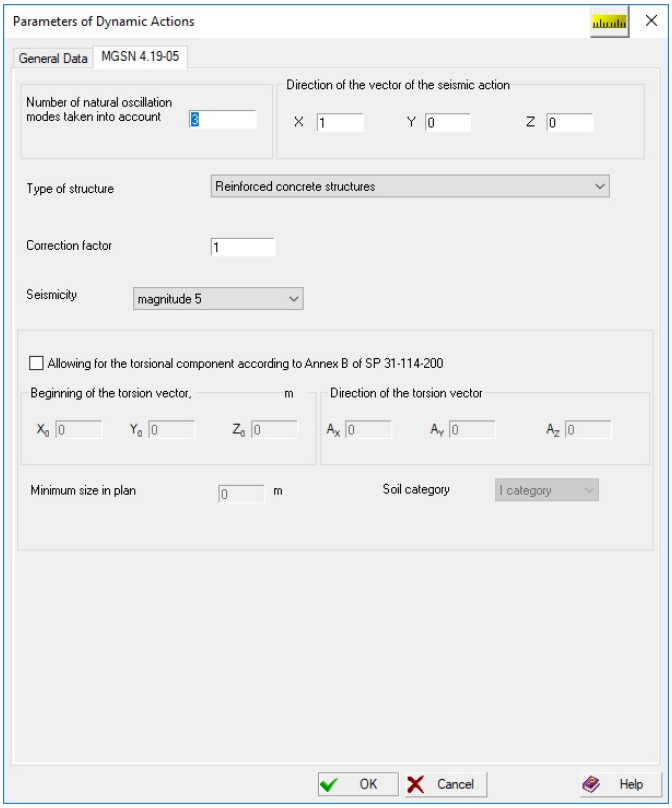


Figure 8.7-7. *The* **MGSN 4.19-05** *tab*

When preparing the initial data it is necessary to pay attention to the following factors:

Seismicity is specified without taking into account the soil category. ***The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor.***

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Type of structure — a steel or reinforced concrete structure is selected.

Direction of the vector of the seismic action is defined by the vector specified in the global coordinate system.

Minimum size in plan is used to determine the coefficient taking into account the seismic wavelength spectrum (see Sec. B.2 of Annex B).

Allowing for the torsional component — if the torsional component is taken into account (the respective checkbox is checked), it is necessary to specify the coordinates of the center of rotation and the direction of the torsion vector (Annex B of SP 31-114-2004).

Coordinates of the center of rotation — they should be taken as the coordinates of the center of gravity of the subgrade area of the building, considering the subgade uniform over the entire area.

Direction of the torsion vector — is specified by the vector of rotational acceleration.

Seismic Analysis according to SP 14.13330.2014 and SP 14.13330.2018 (Updated edition of SNiP II-7-81*) of the Russian Federation

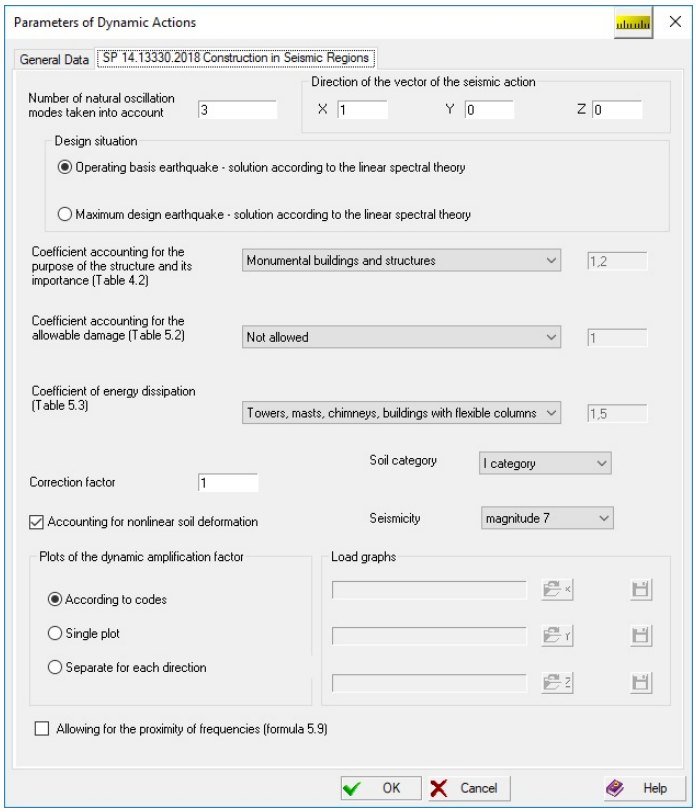


Figure 8.7-8. *The* **SP 14.13330.2014 Construction in Seismic Regions** *tab*

When preparing the initial data it is necessary to pay attention to the following factors:

Design situation — according to SP 14.13330.2014 the calculations can be performed for two design situations (operating basis/design earthquake or maximum design/control earthquake).

Number of natural oscillation modes taken into account — according to this code the minimum number of natural oscillation modes taken into account is specified in such a way so that "the sum of effective modal masses is not less than 90% of the total mass of the system excited in the direction of the seismic action for horizontal actions and not less than 75% — for the vertical one". The Lanczos method (only together with a multifrontal or PARFES processor) is used for the automatic determination of the necessary number of the natural oscillation modes.

Seismicity is specified without taking into account the soil category. ***The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor***.

Coefficient of energy dissipation accounts for the inelasticity of the structure (is taken from Table 5 of 14.13330.2014 and Table 5.3 of SP 14.13330.2018)

Coefficient accounting for the purpose of the structure and its importance is selected according to Table 3 of 14.13330.2014 and Table 4.2 of SP 14.13330.2018.

Coefficient accounting for the allowable damage is selected according to Table 4 of 14.13330.2014 and Table 5.2 of SP 14.13330.2018.

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Direction of the vector of the seismic action is defined by the vector specified in the global coordinate system.

Allowing for the proximity of frequencies — if this checkbox is checked, the formula (9) of SP 14.13330.2014 and formula (5.9) of SP 14.13330.2018, which enables to take into account the mutual correlation of the generalized coordinates, is used for the neighbor natural oscillation modes which differ by less than 10%.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single plot** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 33.10). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

Accounting for nonlinear soil deformation — if this checkbox is checked and the specified magnitude is 8 and greater increased only due the presence of Category III and IV soil, a reduction factor is introduced (see notes to the formula (2) of SP 14.13330.2014 and formula (5.2) of SP 14.13330.2018).

Seismic Analysis according to SP 14.13330.2018 with Chanhe No.1 (Updated Edition of SNiP II-7-81*) of the Russian Federation

When preparing the initial data it is necessary to pay attention to the following factors:

The most dangerous direction of action is determined automatically in this edition of SP 14.1333.2018.

Number of natural oscillation modes taken into account — according to this code “The minimum number of natural oscillation modes taken into account in the analysis should be specified in such a way so that the sum of effective modal masses is not less than 90% of the total mass of the system”. The Lanczos method (only together with a multifrontal or PARFES processor) is used for the automatic determination of the necessary number of the natural oscillation modes.

Design case — analyses can be performed for two design cases according to the **Change No.1** to **SP 14.13330.2018**:

a) design analysis using the response spectrum method (RSM) to obtain data for the structural design;

b) nonlinear dynamic verification analysis in the time domain.

Coefficient accounting for the purpose of the structure and its importance is selected according to Table 5.3 of the **Change No.1** to **SP 14.13330.2018**.

Coefficient accounting for the allowable damage is selected according to Table 5.4 of the **Change No.1** to **SP 14.13330.2018**.

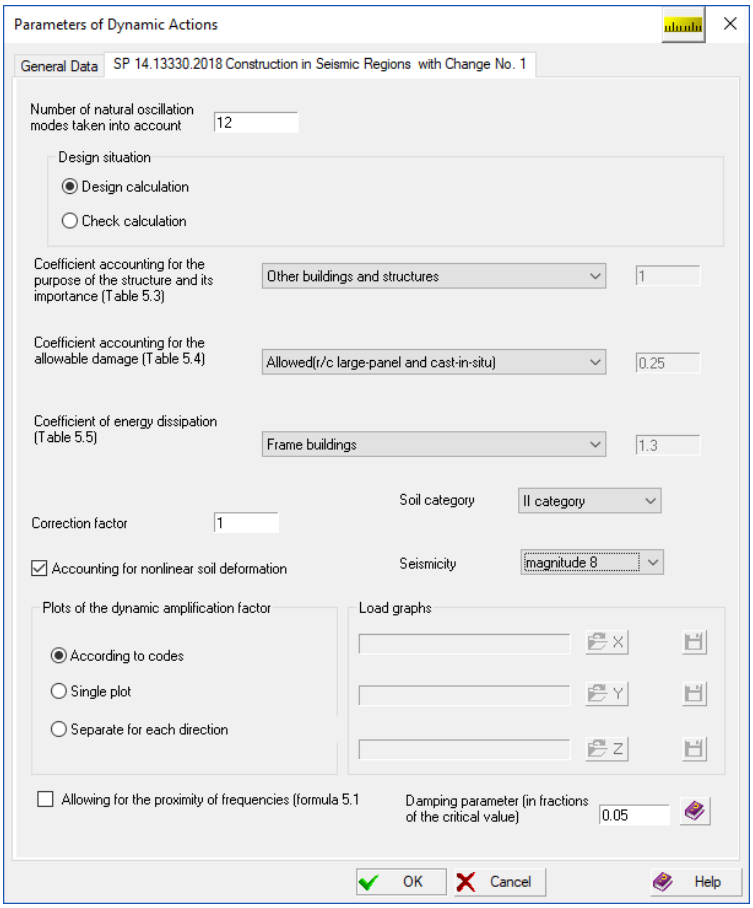


Figure 8.7-9. *The* **SP 14.13330.2019 with the Change No.1 Construction in Seismic Regions** *tab*

Coefficient of energy dissipation accounts for the inelasticity of the structure (is taken from Table 5.5 of the **Change No.1** to **SP 14.13330.2018**)

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of the codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Soil category is selected from the respective list.

Seismicity is specified without taking into account the soil category. ***The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor***.

Allowing for the proximity of frequencies — if this checkbox is checked, the formula (5.13) of the **Change No.1** to **SP 14.13330.2018**, which enables to take into account the mutual correlation of the generalized coordinates, is used for the neighbor natural oscillation modes which differ by less than 10%.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single plot** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 33.10). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

Allowing for nonlinear soil deformation — if this checkbox is checked, a reduction factor is introduced for magnitude 8 and greater increased only due to the III and IV soil categories (see the notes to Sec. 5.23).

Damping parameter (in fractions of the critical value) is specified depending on the material of the structure and its members. The reference values of the parameter for various structures are given in the **Reference Data on the Internal Friction** dialog box, which is invoked by clicking the button .

Calculation according to the Design Codes for Earthquake-Resistant Nuclear Plants NP-031-01

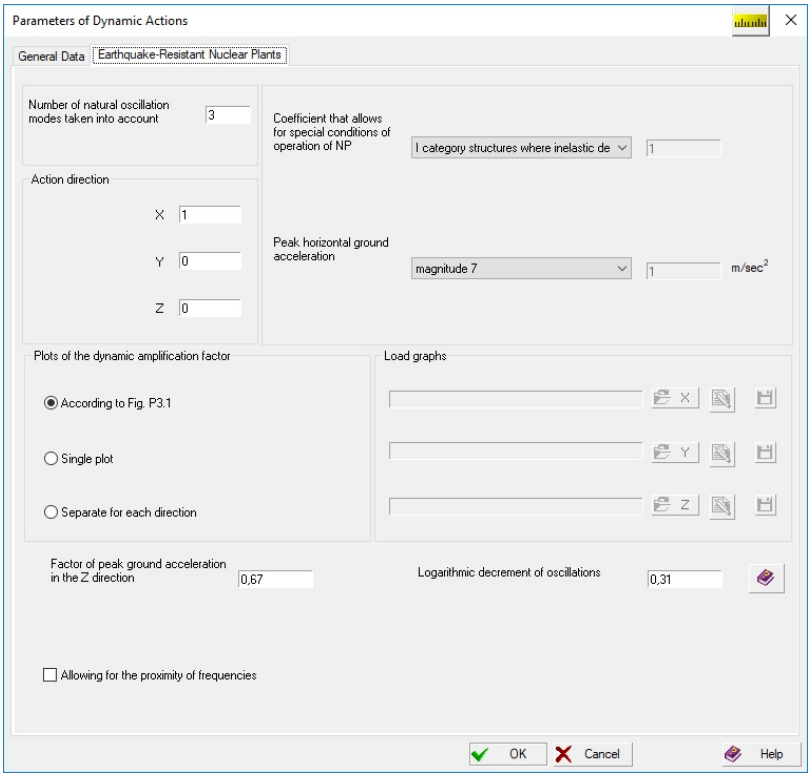


Figure 8.7-10. *The* **Earthquake-Resistant Nuclear Plants** *tab*

When preparing the initial data it is necessary to pay attention to the following factors:

Number of natural oscillation modes taken into account — this code does not provide any recommendations on the minimum necessary number of natural oscillation modes taken into account. However, based on the recommendations of similar codes and the requirements of IAEA we recommend to specify the number of modes in such a way so that "the sum of effective modal masses is not less than 90% of the total mass of the system excited in the direction of the seismic action for horizontal actions and not less than 75% — for the vertical one". The Lanczos method (only together with a multifrontal or PARFES processor) is used for the automatic determination of the necessary number of the natural oscillation modes.

Action direction is defined by the vector specified in the global coordinate system.

Coefficient that allows for special conditions of operation of NP — the value is selected automatically depending on the category and purpose of the structure (is selected from the drop-down list). If the **Other** item is selected in the list, it is necessary to specify the coefficient in the respective text field.

Peak horizontal ground acceleration is taken depending on the seismicity of the site (is selected from the drop-down list). If the **Other** item is selected in the list, it is necessary to specify the acceleration in the given units of measurement in the respective text field.

Coefficient allowing for the peak ground acceleration in the Z direction should be taken as 2/3 according to Sec. 5 of Annex 3.

Logarithmic decrement of oscillations — parameter defining the damping of oscillations of building structures and pipelines. The value of the coefficient is specified by the user or can be determined with help of the **Reference data on the internal friction** operation in the respective dialog box.

According to Sec.6 of Annex 3 this coefficient is required only when the plot of the dynamic amplification factor given in the code is used.

Plots of the dynamic amplification factor —in addition to the ranges of plots of the dynamic amplification factors given in the Annex 3 (P.3.1) it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to Fig. P3.1.** radio button), load a prepared file with a single plot (the **Single plot** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 33.10). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

Allowing for the proximity of frequencies — if this checkbox is checked, the formula which enables to take into account the mutual correlation of the generalized coordinates is used for the neighbor natural oscillation modes which differ by less than 10%.

Seismic Analysis according to Ukrainian Design Codes DBN B.1.1-12:2006

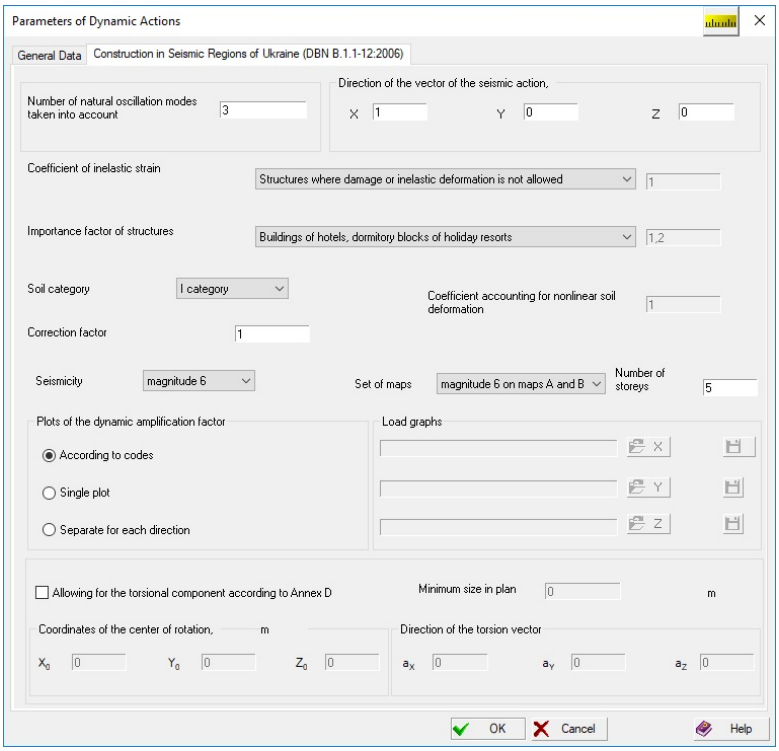


Figure 8.7-11. *The* **Construction in Seismic Regions of Ukraine** *tab*

When preparing the initial data it is necessary to pay attention to the following factors:

Number of natural oscillation modes taken into account — according to this code the minimum number of natural oscillation modes taken into account is specified in such a way so that the sum of modal masses is not less than 85% of the total mass of the system excited in the direction of the seismic action for horizontal actions and not less than 75% — for the vertical one. The Lanczos method (only together with a multifrontal or PARFES processor) is used for the automatic determination of the necessary number of the natural oscillation modes.

Seismicity is specified without taking into account the soil category. ***The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor.*** The value of design seismicity for a certain terrain can be specified by selecting from the list or using the **Select a settlement** operation, which can be invoked by clicking the button on the right

from the list. In the last case the dialog box appears with a list of settlements according to Annex A. After selecting the value of seismicity in the list and clicking the **OK** button the dialog box closes, and the selected value is displayed in the **Seismicity** list.

Coefficient of inelastic strain accounts for the inelasticity of the structure and the local damage in it (is taken from Table 2.3). The value of the coefficient depends on the seismicity of the construction site.

Importance factor of structures is selected from the list corresponding to Table 2.4. If the **Other** item is selected in the list, the value has to be specified in the text field on the right from the list.

Soil category is selected from the respective list. If the IV category is selected, it is necessary to specify the **coefficient accounting for nonlinear soil deformation** in the respective field.

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Direction of the vector of the seismic action is defined by the vector specified in the global coordinate system.

Allowing for the torsional component (Annex D) — if the torsional component is taken into account (the respective checkbox is checked), it is necessary to specify the coordinates of the center of rotation and the direction of the torsion vector.

Coordinates of the center of rotation — they should be taken as the coordinates of the center of gravity of the subgrade area of the building, considering the subgade uniform over the entire area.

Direction of the torsion vector is specified by the vector of rotational acceleration.

Minimum size in plan is used to determine the coefficient taking into account the sizes of the structure in plan (Sec. D.6).

Number of storeys — number of storeys.

Plots of the dynamic amplification factor —in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single diagram** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 33.10). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

Seismic Analysis according to Ukrainian Design Codes DBN B.1.1-12:2014

When preparing the initial data it is necessary to pay attention to the following factors:

Number of natural oscillation modes taken into account — according to this code the minimum number of natural oscillation modes taken into account is specified in such a way so that the sum of modal masses is not less than 85% of the total mass of the system excited in the direction of the seismic action for horizontal actions and not less than 75% — for the vertical one. The Lanczos method (only together with a multifrontal or PARFES processor) is used for the automatic determination of the necessary number of the natural oscillation modes.

Seismicity is specified without taking into account the soil category. ***The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor***. The value of design seismicity for a certain terrain can be specified by selecting from the list or using the **Select a settlement** operation, which can be invoked by clicking the button on the right from the list. In the last case the dialog box appears with a list of settlements according to Annex A. After selecting the value of seismicity in the list and clicking the **OK** button the dialog box closes, and the selected value is displayed in the **Seismicity** list.

Coefficient of inelastic strain accounts for the inelasticity of the structure and the local damage in it (is taken from Table 6.3). The value of the coefficient depends on the seismicity of the construction site.

Importance factor of structures is selected from the list corresponding to Table 2.4. If the **Other** item is selected in the list, the value has to be specified in the text field on the right from the list.

Soil category is selected from the respective list. If the IV category is selected, it is necessary to specify the **coefficient accounting for nonlinear soil deformation** in the respective field.

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

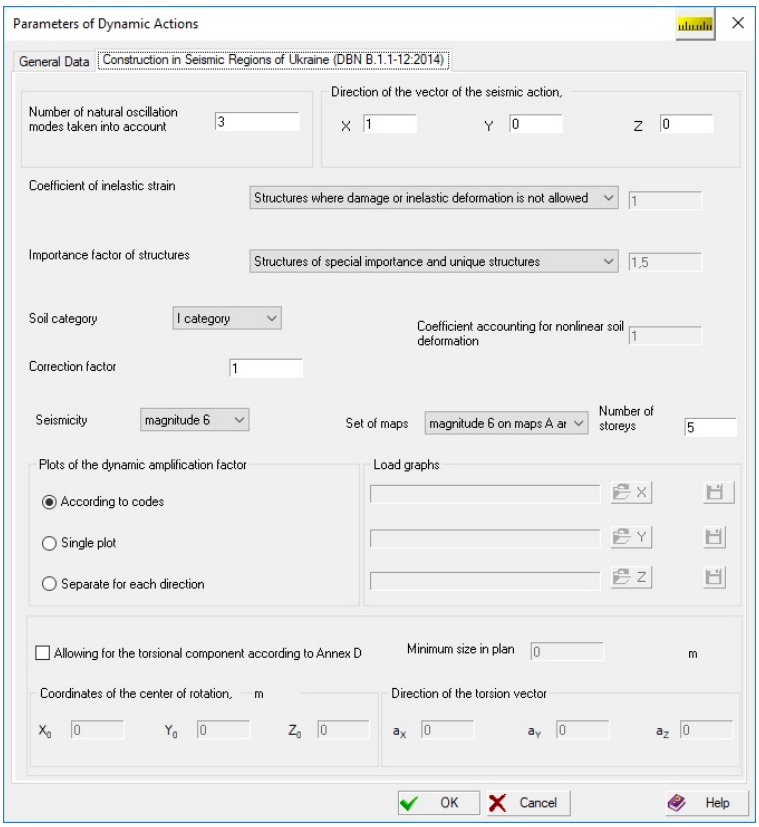


Figure 8.7-12. *The* **Construction in Seismic Regions of Ukraine** *tab*

Direction of the vector of the seismic action is defined by the vector specified in the global coordinate system.

Allowing for the torsional component (Annex I) — if the torsional component is taken into account (the respective checkbox is checked), it is necessary to specify the coordinates of the center of rotation and the direction of the torsion vector.

Coordinates of the center of rotation — they should be taken as the coordinates of the center of gravity of the subgrade area of the building, considering the subgade uniform over the entire area.

Direction of the torsion vector is specified by the vector of rotational acceleration.

Minimum size in plan is used to determine the coefficient taking into account the sizes of the structure in plan (Sec. I.6).

Number of storeys — number of storeys.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single diagram** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 33.10). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

Seismic Analysis according to SNiP B1.2-4-98 of the Republic of Kazakhstan

When preparing the initial data it is necessary to pay attention to the following factors:

Seismicity is specified without taking into account the soil category. ***The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor***.

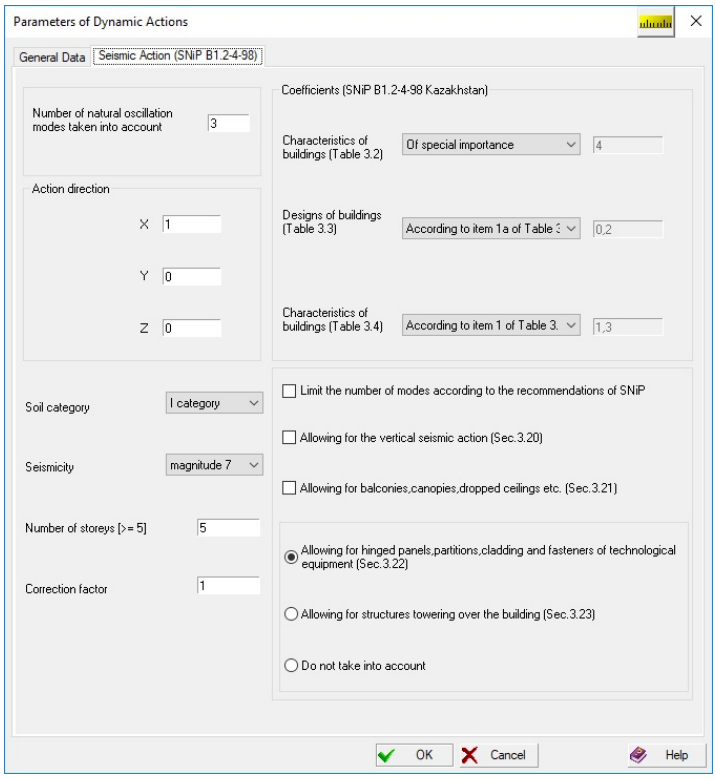


Figure 8.7-13. *The* **Seismic Action (SNiP B1.2-4-98)** *tab*

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Action direction is defined by the vector specified in the global coordinate system.

Values of coefficients K_1, K_2 and K_ψ are specified automatically by selecting the characteristics from the drop-down lists coinciding with Tables 3.2, 3.3 and 3.4 of SNiP respectively. The display of coefficients in the text fields on the right from the drop-down list is given for reference. The fields become available for input in the case if the **Other** item is selected in the list.

Options **Limit the number of modes …**, **Allowing for the vertical seismic action**, **Allowing for balconies, canopies, dropped ceilings …** can be selected in any combination.

Options **Allowing for hinged panels, partitions …** and **Allowing for structures towering over the building** are mutually exclusive.

Seismic Analysis according to SNiP RK 2.03-30-2006 of the Republic of Kazakhstan

When preparing the initial data it is necessary to pay attention to the following factors:

Seismicity is specified without taking into account the soil category. ***The change of seismicity depending on the soil category should be taken into account when specifying the value of the correction factor.***

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

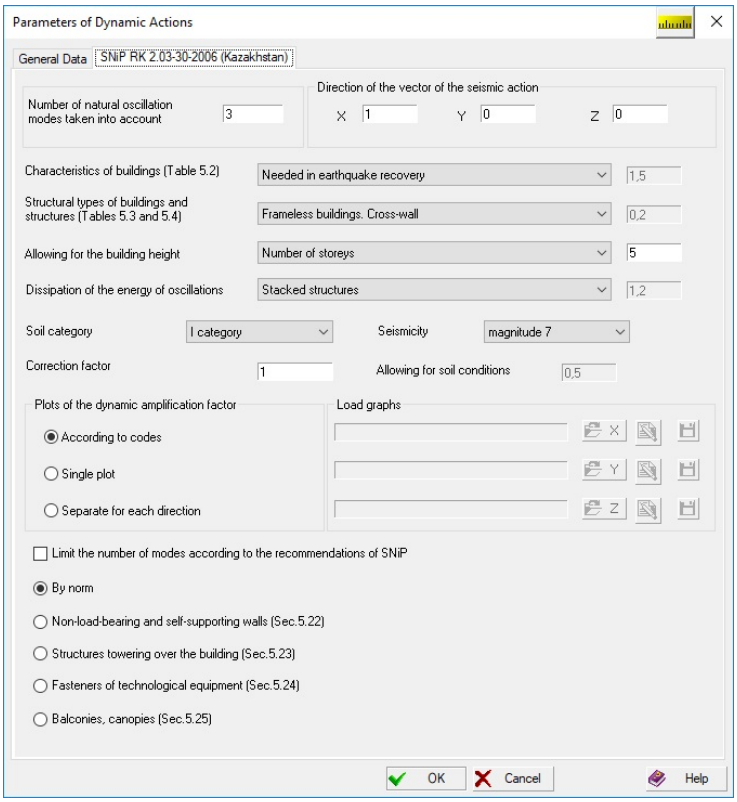


Figure 8.7-14. *The* **SNiP RK 2.03-30-2006 (Kazakhstan)** *tab*

Action direction is defined by the vector specified in the global coordinate system.

Values of coefficients K_1 (allowing for the importance of the building), K_2 (allowing for the designs of the building), K_3 (allowing for the height of the building) and K_ψ (allowing for the dissipation of the energy of oscillations) are specified automatically by selecting the characteristics from the drop-down lists coinciding with Tables 5.2, 5.3 and 5.4 of SNiP respectively. The display of coefficients in the text fields on the right from the drop-down list is given for reference. The fields become available for input in the case if the **Other** (K_1) or **According to the results of the research** (K_2) item is selected in the list.

Options **Non-load-bearing and self-supporting walls** and **Structures towering over the building**, **Fasteners of technological equipment**, **Balconies, canopies** are mutually exclusive.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single diagram** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 33.10). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

Seismic Analysis according to SP RK 2.03-30-2017, SP RK 2.03-30-2017*, and NTP RK 08-01.1-2017 of the Republic of Kazakhstan

When preparing the initial data it is necessary to pay attention to the following factors:

Acceleration – design horizontal acceleration at the construction site determined according to Sec. 7.5.5 of SP or according to its Annex F.

Action direction is defined by the vector specified in the global coordinate system.

Importance class by purpose is specified in accordance with Table 7.2.

Importance class by the number of storeys is specified in accordance with Table 7.3.

Building type defines the choice of the coefficients from Table 7.8 or 7.9. The coefficient is displayed in the text field on the right from the drop-down list. The text field will become available for input if a column 10a or 10b is selected in the list of the Table 7.8.

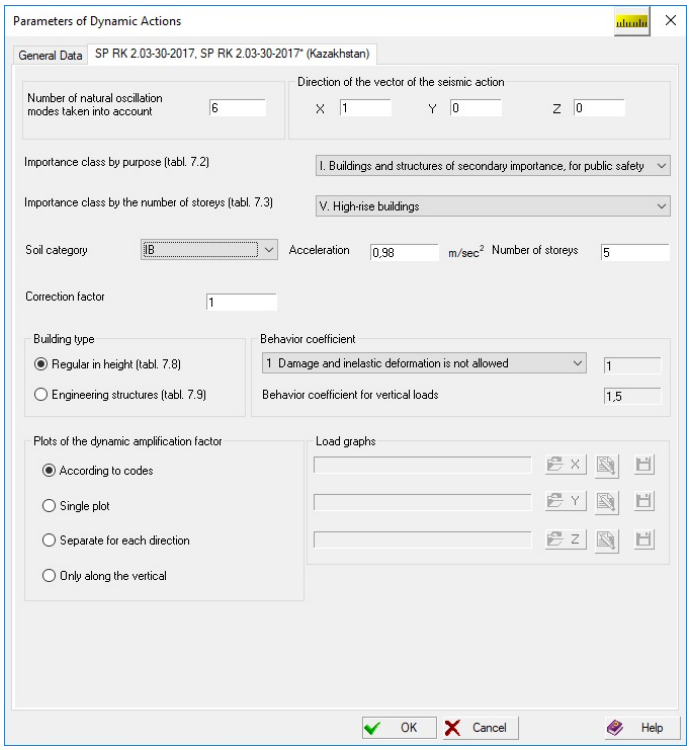


Figure 8.7-15. *The* **SP RK 2.03-30-2017 (Kazakhstan)** *tab*

The behavior factor for vertical loads defines the behavior factor q taken into account when determining the design vertical seismic loads. This factor takes the value of 1.5 in all cases except for the case when the selected building type is *Experimental studies* (the user can enter the value refined by the results of experimental and theoretical studies).

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single diagram** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). If **Along the vertical** option is selected, a plot for the implementation of the requirements of Sec. 7.5.4 is specified. The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package (see Sec. 30.8). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

The boost factor f_{vk} can be specified for each element with the help of the **Specification of the additional data** for elements operation. If this parameter is not specified for an element, its analysis is performed on the assumption that f_{vk}=1.

Seismic Analysis according to SNRA II-2.02.94 of the Republic of Armenia

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Action direction is defined by the vector specified in the global coordinate system.

The values of other coefficients are taken from the tables of the code.

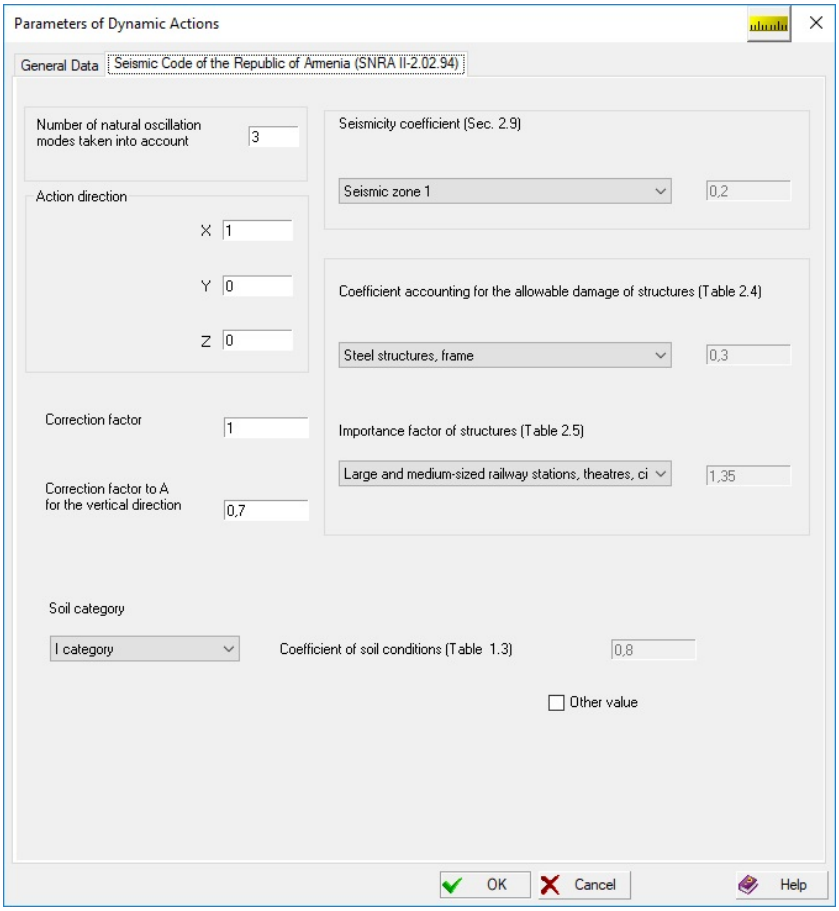


Figure 8.7-16. *The* **Seismic Code of the Republic of Armenia (SNRA II-2.02.94)** *tab*

Seismic Analysis according to SNRA II-6.02.2006 of the Republic of Armenia

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

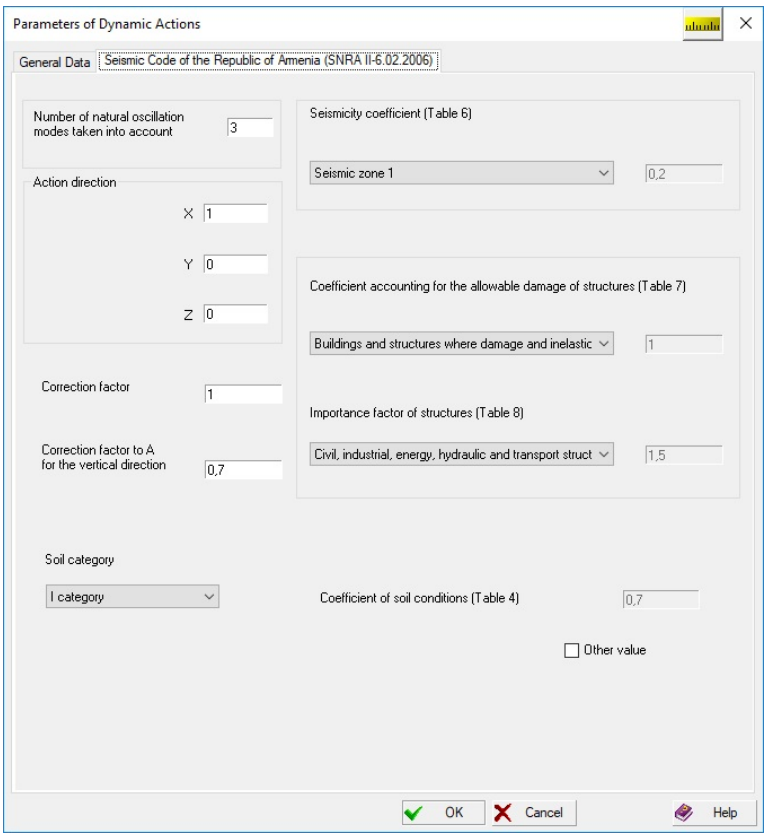


Figure 8.7-17. *The* **Seismic Code of the Republic of Armenia (SNRA II-6.02.2006)** *tab*

Action direction is defined by the vector specified in the global coordinate system.
The values of other coefficients are taken from the tables of the code.

Seismic Analysis according to SNRA AzDTN 2.3-1 of the Republic of Azerbaijan

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

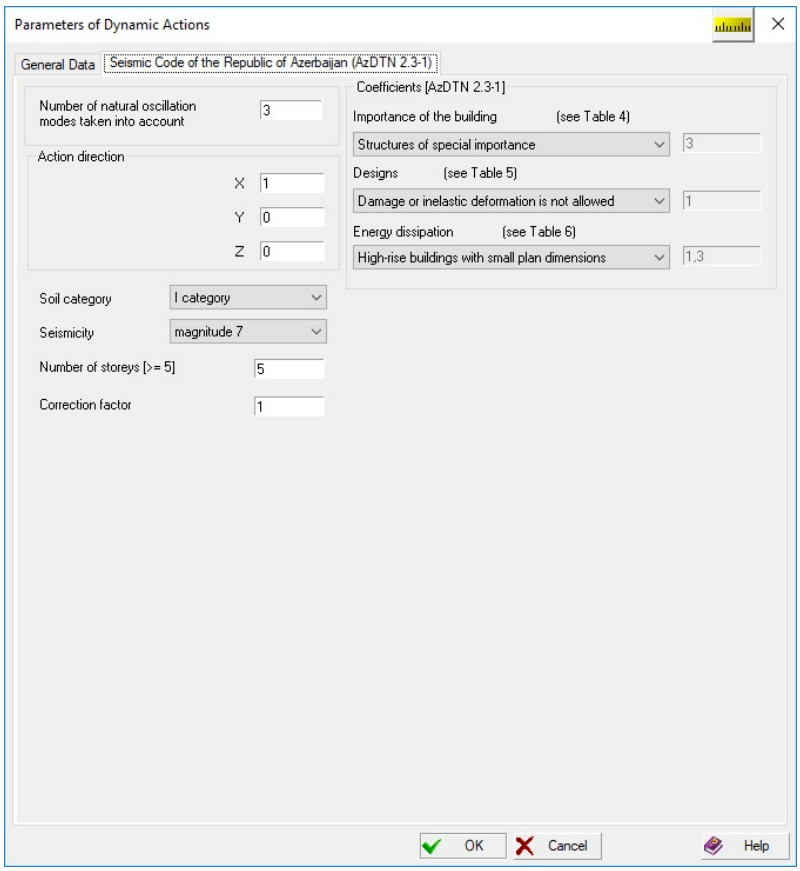


Figure 8.7-18. *The* **Seismic Code of the Republic of Azerbaijan (AzDTN 2.3-1)** *tab*

Action direction is defined by the vector specified in the global coordinate system.

The values of other coefficients are taken from the tables of the code.

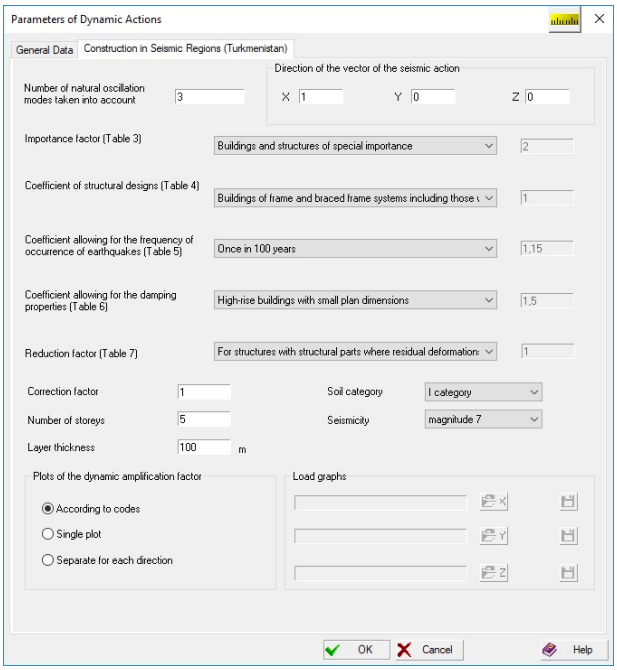
Seismic Analysis according to SNT 2.01.08-99 and SNT 2.01.08-99* of Turkmenistan

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Direction of the vector of the seismic action is defined by the vector specified in the global coordinate system.

The values of other coefficients are taken from the tables of the code.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single diagram** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). The values of other coefficients are taken from the code.



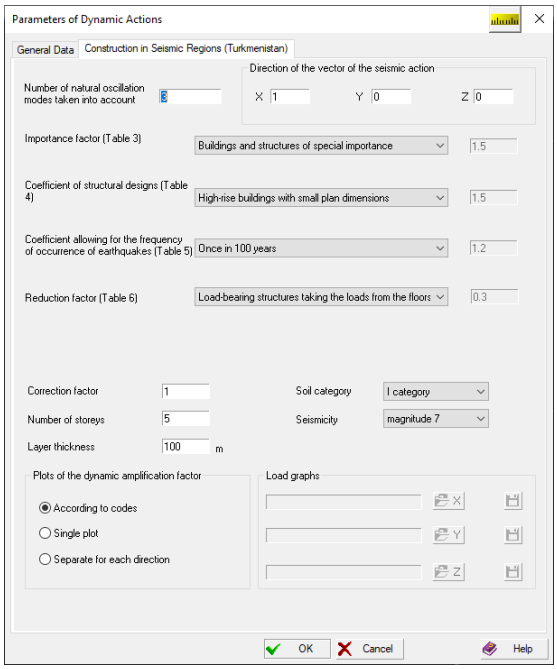


Figure 8.7-19. *The* **Construction in Seismic Regions (Turkmenistan)** *tab*
SNT 2.01.08-99 and SNT 2.01.08-99*

Seismic Analysis according to KMK 2.01.03-96, KMK 2.01.03-19 of the Republic of Uzbekistan

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

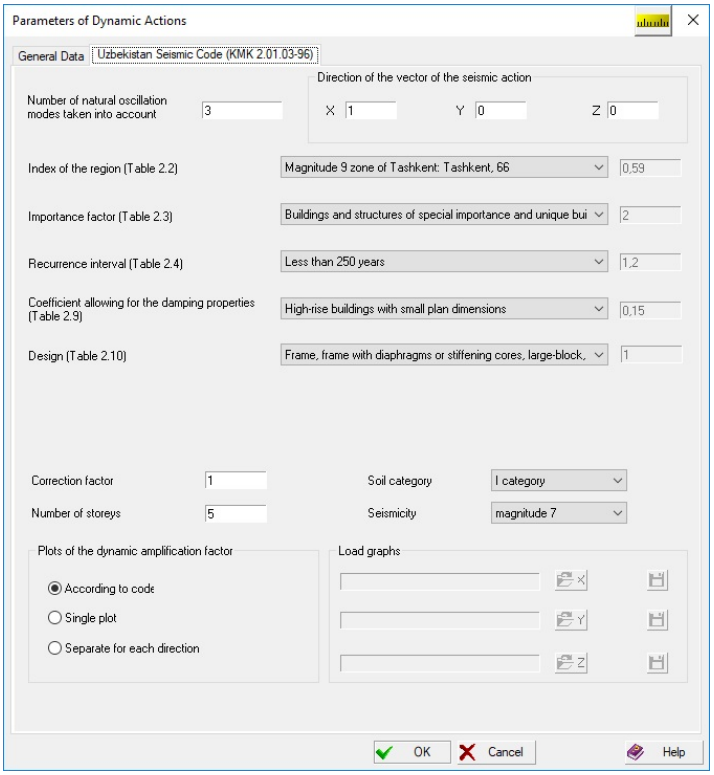


Figure 8.7-20. *The* **Construction in Seismic Regions of Uzbekistan** *tab*

Direction of the vector of the seismic action is specified by the values of cosines of angles with respect to the axes of the global coordinate system.

The values of other coefficients are taken from the tables of the code depending on the factor defining them selected from the respective list. These values are displayed in the information fields on the right from the list.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single diagram** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button).

The parameter μ (limit relative inelastic deformation) can be specified for each element with the help of the **Specification of the additional data for elements** operation (see Sec. 7.3). If this parameter is not specified for an element, its analysis is performed on the assumption of perfect elasticity (μ = 1) and the reduction factor *r* in formula (2.8) of KMK 2.01.03-96 is taken as 1.

Seismic Analysis according to SNiP KR 20-02:2009 of the Kyrgyz Republic

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

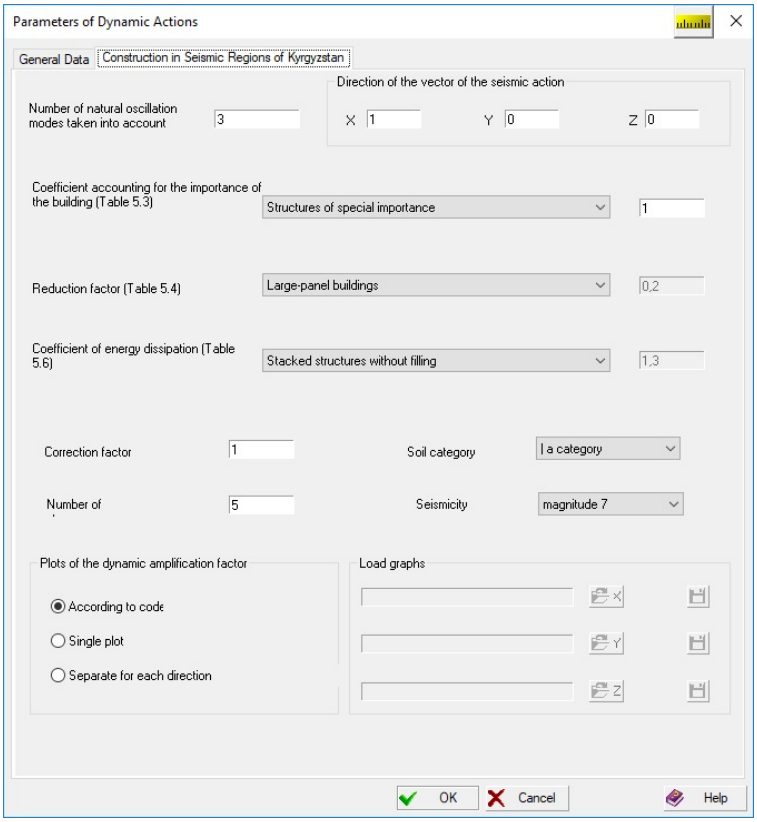


Figure 8.7-21. *The* **Earthquake Engineering of the Kyrgyz Republic** *tab*

Direction of the vector of the seismic action is specified by the values of cosines of angles with respect to the axes of the global coordinate system.

The values of other coefficients are taken from the tables of the code depending on the factor defining them selected from the respective list. These values are displayed in the information fields on the right from the list.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single diagram** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button).

Seismic Analysis according to GNiP RT 22-07-2015, GNiP RT 22-07-2018 Tajikistan Republic

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Direction of the vector of the seismic action is specified by the values of cosines of angles with respect to the axes of the global coordinate system.

The values of other coefficients are taken from the tables of the code depending on the factor defining them selected from the respective list. These values are displayed in the information fields on the right from the list.

Plots of the dynamic amplification factor — in addition to the ranges of plots of the dynamic amplification factors given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single diagram** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button).

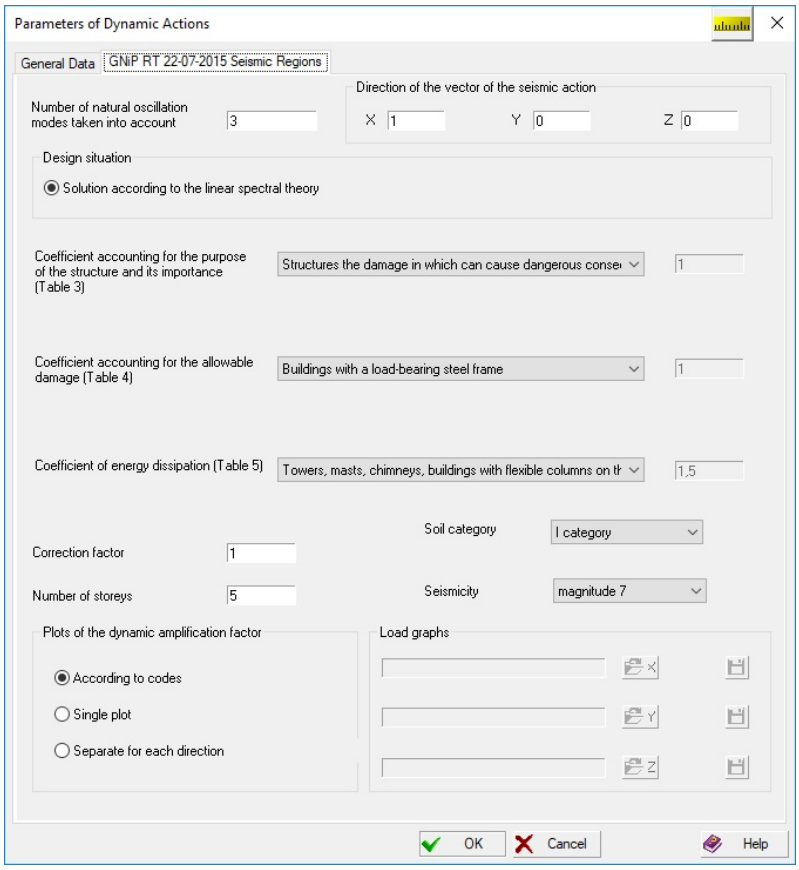


Figure 8.7-22. *The* **Earthquake Engineering of the Tajikistan Republic** *tab* **(GNiP RT 22-07-2015/2018)**

Seismic Analysis according to SP 358.1325800.2017

When preparing the initial data it is necessary to pay attention to the following factors:

Seismic soil category is selected from the respective list.

Coefficient of inelastic strain (Coefficient allowing for damping properties of the structure) accounts for the inelasticity of the structure and the allowable damage in it. The type of structure is selected from the drop-down list. The reference value of the coefficient is displayed in the information field next to the list.

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of SP more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Direction of the vector of the seismic action is specified by the values of cosines of angles with respect to the axes of the global coordinate system.

Height of the hydraulic structure is specified in the respective field.

Design acceleration depends on the value of the initial seismicity of the construction site, design seismicity and soil conditions.

Plots of the dynamic amplification factor — the considered codes allow to use special forms of the plot (spectrum) of the dynamic amplification factor. The plot can be specified as a piecewise linear function and saved in a file with the extension **.spd**. The plot can be obtained using **Editor of Dynamic Amplification Factor** included in the **SCAD Office** system package. If a file with a plot is selected, the button , enables to invoke **Editor of Dynamic Amplification Factor** for viewing the plot.

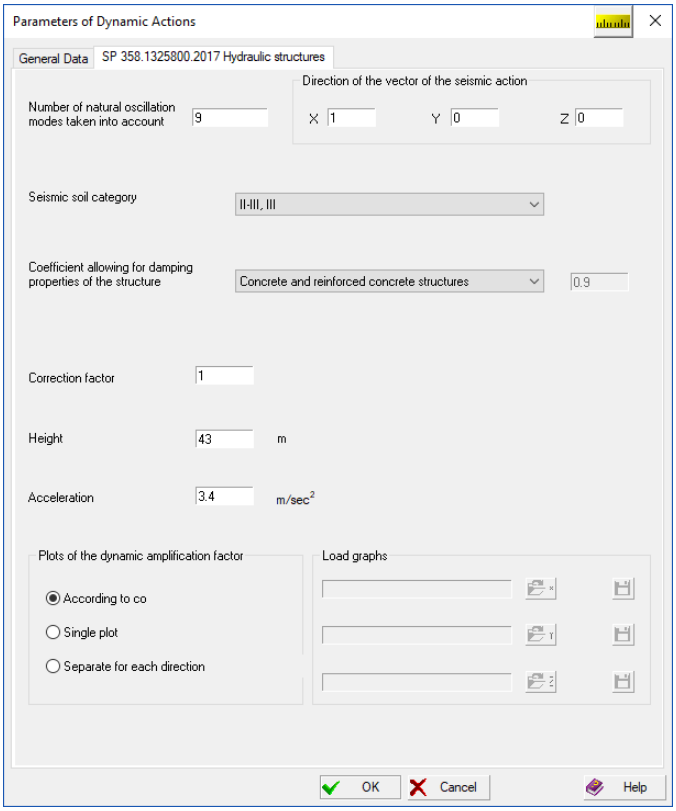


Figure 8.7-23. *The* **SP 358.1325800.2017 Hydraulic Structures** *tab*

Besides a single plot you can also specify three plots — separately for each direction of the action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single plot** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button).

Seismic Analysis according to EN 1998-1

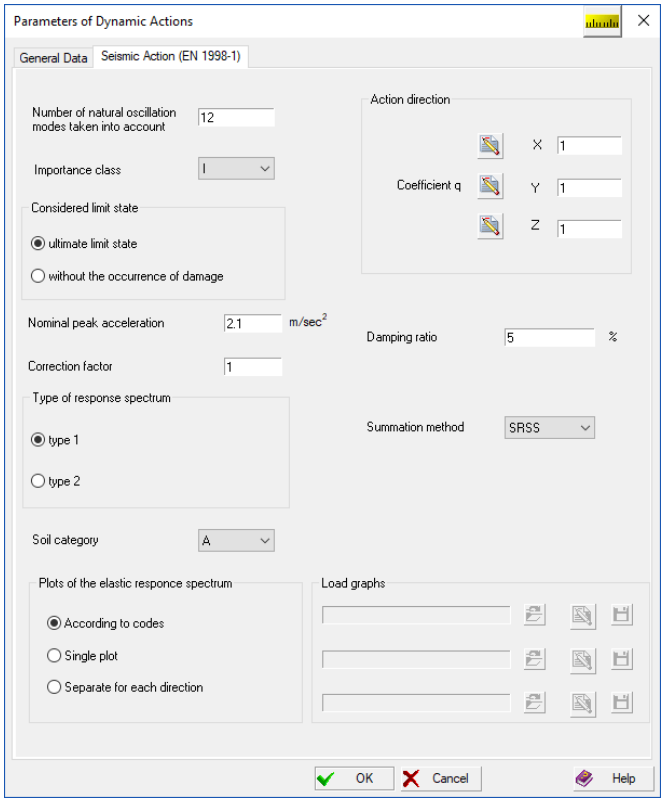


Figure 8.7-24. *The* **Seismic Action (EN 1998-1)** *tab of the* **Parameters of Dynamic Actions** *dialog box*

Data for performing the seismic analysis according to Eurocode are specified in the **Seismic Action (EN 1998-1)** tab of the **Parameters of Dynamic Actions** dialog box (Fig. 8.7-24) and include the following parameters:

Importance class — according to EN 1998-1 buildings are classified in 4 importance classes: I, II, III and IV. The importance classes are characterized by corresponding importance factors $\gamma_I = 0{,}8$; $\gamma_I = 1{,}0$; $\gamma_I = 1{,}2$; $\gamma_I = 1{,}4$.

Nominal peak acceleration — The design acceleration value is obtained by multiplying the nominal peak ground acceleration, which is determined from the seismic zoning map with the assumed probability of exceedance, by the factor γ_I.

Considered limit state — The structure shall be designed to withstand the design seismic action without local or global collapse (ultimate limit state). In this case we use a map with accelerations that have a 10% reference probability of exceedance and a reference return period of 475 years. Alternatively, the structure can be designed to withstand a seismic action without the occurrence of damage and the associated limitations of use, in which case we use a map with accelerations that have a 10% reference probability of exceedance and a reference return period of 95 years. These maps can be found in the National Annexes.

Action direction — Eurocode does not use the concept of "the direction of the seismic action". Three options are always considered (along the X axis, along the Y axis, and vertical — along the Z axis). The calculations for these options are performed taking into account the behaviour factor q (it is equal to 1,0 by default), specified for each direction of the seismic action.

The value of the behaviour factor q is selected in the respective dialog box (Fig. 8.7-25), which is invoked by clicking the button , corresponding to the necessary action direction.

Damping ratio — Eurocode considers 5% viscous damping. This value is set by default and can be changed if necessary.

Type of response spectrum — Unlike our national standards, Eurocode uses spectral curves $S_e(T)$, which depend on the direction of the seismic action, instead of the dynamic amplification factor curves. Two types of spectra can be used: Type 1 and Type 2. If the earthquakes that contribute most to the seismic hazard defined for the site for the purpose of probabilistic hazard assessment have a surface-wave magnitude not greater than 5,5, it is recommended that the Type 2 spectrum is adopted.

Soil category — Soil conditions of the construction site are characterized by one of the following categories: A, B, C, D or E (unlike our national standards which use only three soil categories).

Summation method — SCAD provides various methods for obtaining the resulting action, taking into account actions in different directions (see Table 8.6-2):

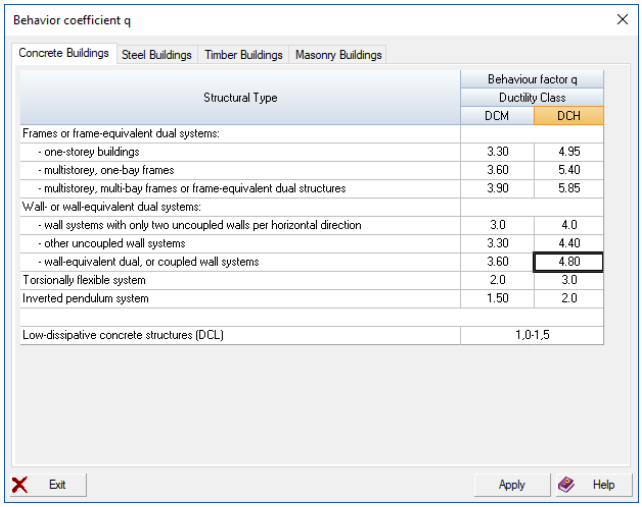
1. DSM+SRSS;
2. DSM+Newmark(0.3);
3. DSM+ Newmark(0.4);
4. SRSS+ Newmark(0.3);
5. SRSS+ Newmark(0.4);
6. CQC+ SRSS;
7. CQC+ Newmark(0.3);
8. CQC+ Newmark(0.4);
9. SRSS.

The first method is the modal summation method for all modes in the directions X, Y and Z. Then combinations are created from the three obtained values by the second method.

Correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it.

Number of natural oscillation modes taken into account — according to this code the minimum number of natural oscillation modes taken into account is specified in such a way so that the sum of modal masses is not less than 85% of the total mass of the system excited in the direction of the seismic action for horizontal actions and not less than 75% — for the vertical one. The Lanczos method (only together with a multifrontal or PARFES processor) is used for the automatic determination of the necessary number of the natural oscillation modes.

Elastic response spectra — in addition to the response spectra given in the codes it is allowed to use special plots specified as a file. Such single plot can be used for all three directions, or different plots can be used for each direction of action. Depending on the selected radio button the software can use a plot given in the codes (the **According to codes** radio button), load a prepared file with a single plot (the **Single plot** radio button) or three files with separate plots for each direction (the **Separate for each direction** radio button). If a file with a plot is selected, the button enables to invoke **Editor of Dynamic Amplification Factor** for viewing the elastic response spectrum.



Structural Type	Behaviour factor q, Ductility Class DCM	DCH
Frames or frame-equivalent dual systems:		
- one-storey buildings	3.30	4.95
- multistorey, one-bay frames	3.60	5.40
- multistorey, multi-bay frames or frame-equivalent dual structures	3.90	5.85
Wall- or wall-equivalent dual systems:		
- wall systems with only two uncoupled walls per horizontal direction	3.0	4.0
- other uncoupled wall systems	3.30	4.40
- wall-equivalent dual, or coupled wall systems	3.60	4.80
Torsionally flexible system	2.0	3.0
Inverted pendulum system	1.50	2.0
Low-dissipative concrete structures (DCL)	1,0-1,5	

Figure 8.7-25. The **Behaviour Factor q** dialog box

Seismic Analysis Based on an Accelerogram

Data for performing the accelerogram based analysis are specified in the **Seismic Analysis Based on One Accelerogram** tab (Fig. 8.7-24). In this analysis one accelerogram acts in the X, Y and Z directions. It is necessary to specify the following information in this mode:

- number of natural oscillation modes taken into account;
- action direction of the seismic load is defined by the vector specified in the global coordinate system.

In addition to these data you also have to select a name of the file (by clicking the button) containing ordinates of an accelerogram, and to specify the loss factor and the accelerogram scale factor.

The button invokes a dialog box which enables to obtain the reference information on the coefficients of inelastic resistance of different structures (Fig. 8.7-26). After selecting the necessary type of structure and clicking the **OK** button, you can transfer the respective value of the loss factor to the data of the loading. Moreover, if the **Other types of structures** item is selected in the **Type of structure** list, having specified one of the possible representations of information on the inelastic resistance (for example, logarithmic decrement of oscillations), you can obtain the necessary loss factor.

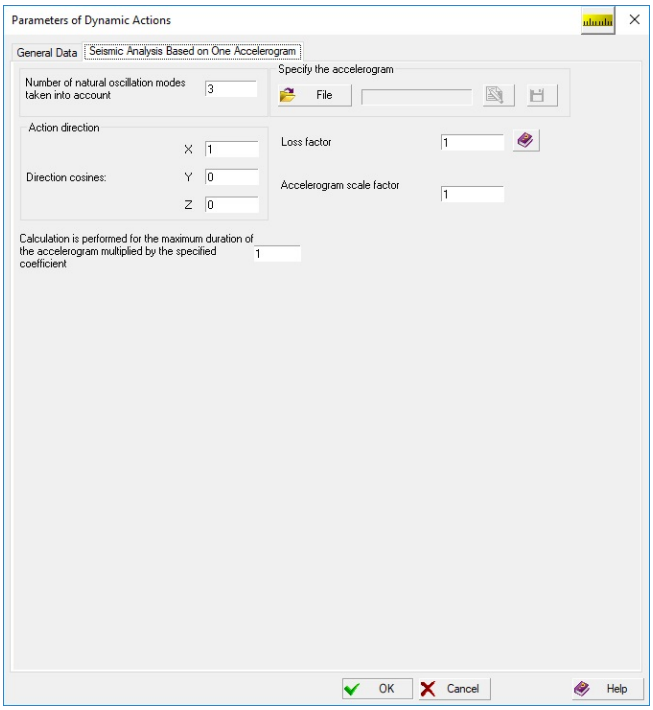


Figure 8.7-26. *The* **Seismic Analysis Based on One Accelerogram** *tab*

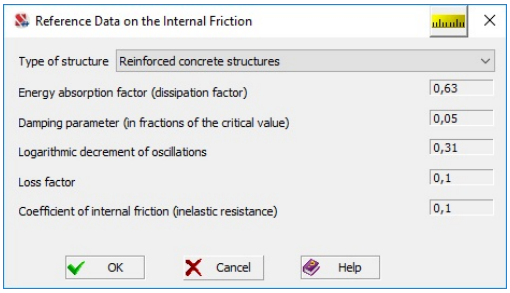


Figure 8.7-27. *The* **Reference Data on the Internal Friction** *dialog box*

Files of accelerograms (having an extension **.spc**) can be prepared with the help of **Accelerogram Editor**, the description of which is given in Section 33.8.

The program enables to perform **the analysis for the maximum duration of the accelerogram multiplied by a coefficient specified by the user**. The meaning of this coefficient is that the calculation will be performed not only in the period of the accelerogram action but after the earthquake as well, when the impact is over, but the movement is still there.

Seismic Analysis Based on the Specified Accelerograms (3 Components)

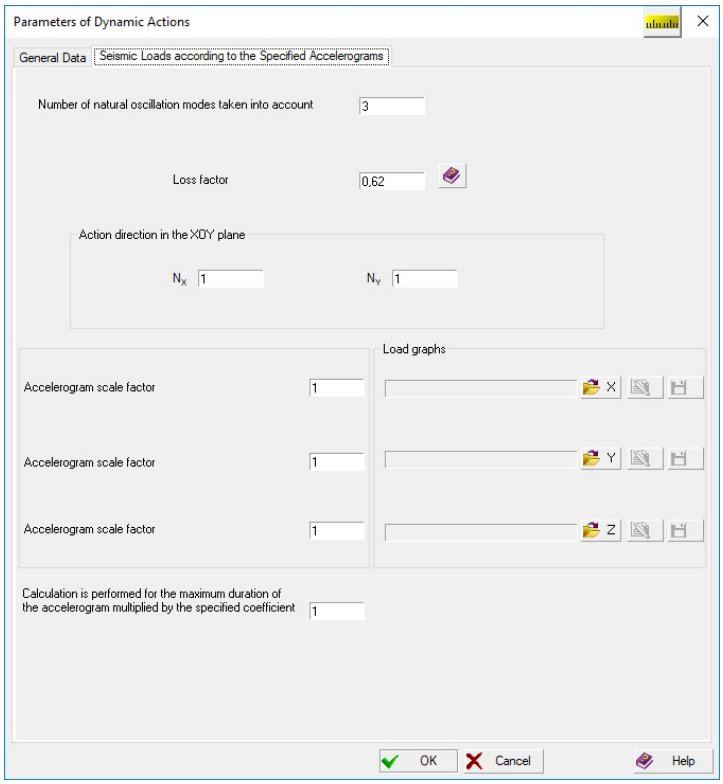


Figure 8.7-28. *The* **Seismic Loads According to the Specified Accelerograms** *tab*

Data for performing the seismic analysis based on the accelerograms taking into account the simultaneous actions in three directions are specified according to the same rules as for the analysis for one direction. The difference is that the list of accelerograms includes the names of three files with ordinates of accelerograms in the directions X, Y and Z, respectively (Fig. 8.7-28). **Action direction in the XOY plane** defines the local X_1 axis corresponding to the action direction of the first accelerogram. The second accelerogram will act normal to this direction.

Files of accelerograms (having an extension **.spc**) can be prepared with the help of **Accelerogram Editor**, the description of which is given in Section 33.8.

If the file with an accelerogram is selected, the button enables to invoke the **Accelerogram Editor** to view the accelerogram and its parameters.

A scale factor can be specified for each accelerogram. Each accelerogram is loaded using the respective button .

The direction of action of the seismic load (in the XOY plane) is defined by the vector specified in the global coordinate system.

The button invokes a dialog box which enables to obtain the reference information on the coefficients of inelastic resistance of different structures (Fig. 8.7-25). After selecting the necessary type of structure and clicking the **OK** button, you can transfer the respective value of the loss factor to the data of the loading. Moreover, if the **Other types of structures** item is selected in the **Type of structure** list, having specified one of the possible representations of information on the inelastic resistance (for example, logarithmic decrement of oscillations), you can obtain the necessary loss factor.

The program enables to perform **the analysis for the maximum duration of the accelerogram multiplied by a coefficient specified by the user**. The meaning of this coefficient is that the calculation will be performed not only in the period of the accelerogram action but after the earthquake as well, when the impact is over, but the movement is still there.

Seismic Analysis Based on the on the Six-Component Action

The operation enables to perform a dynamic analysis for the case when a plot of the dynamic amplification factor is specified for each of the six directions. When specifying the initial data it is necessary to specify at least one plot. If a plot is not specified for a certain direction, it is considered that there is no action in the given direction.

Plots of the dynamic amplification factor can be loaded by clicking the respective button and selecting the necessary spd-file. If the file with a plot of the dynamic amplification factor is selected, the button enables to invoke the **Editor of Dynamic Amplification Factor** to view it (Fig. 8.7-29).

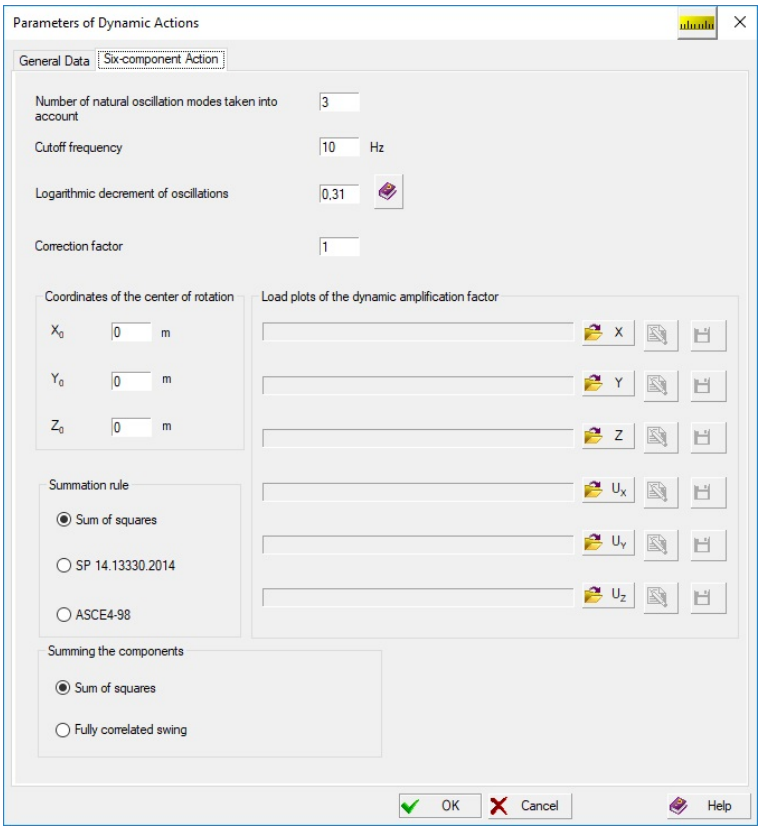


Figure 8.7-29. *The* **Six-component Action** *tab*

Moreover, it is necessary to specify the following data:

- number of natural oscillation modes taken into account;
- cutoff frequency — the upper limit of frequencies which are adequately represented in the model for the calculation of the dynamic interaction of the structure with the subgrade. It can be selected at the level of the doubled dominant frequency of the “structure-subgrade” system in the selected direction, but not less than 10 Hz;
- logarithmic decrement of oscillations;
- correction factor is specified for the correction of results if it is necessary to take into account the requirements of codes more fully. This factor can take any positive value, and the results of the analysis of inertial forces from the seismic action are multiplied by it;
- allowing for the proximity of frequencies — you can select a rule for taking into account the neighbor natural oscillation modes which differ by less than 10% (according to the "sum of squares" rule, according to the recommendations of SP 14.13330.2011 (formula (9)) or ASCE4-98(formula (3.2.19)));
- summing the components – you can select a standard rule for summing the reactions (according to the “sum of squares” rule) in the directions of a six-component seismic action or use the summing with a fully correlated oscillation which is performed according to the following formula $\pm\sqrt{\left(\left|R_x\right|+\left|R_{ux}\right|\right)^2+\left(\left|R_y\right|+\left|R_{uy}\right|\right)^2+R_z^2+R_{uz}^2}$.

The program enables to perform **the analysis for the maximum duration of the accelerogram multiplied by a coefficient specified by the user**. The meaning of this coefficient is that the calculation will be performed not only in the period of the accelerogram action but after the earthquake as well, when the impact is over, but the movement is still there.

The button invokes a dialog box which enables to obtain the reference information on the coefficients of inelastic resistance of different structures. After selecting the necessary type of structure and clicking the **OK** button, you can transfer the respective value of the coefficient of inelastic resistance to the

data of the loading. Moreover, if the **Other types of structures** item is selected in the **Type of structure** list, having specified one of the possible representations of information on the inelastic resistance (for example, logarithmic decrement of oscillations), you can obtain the necessary coefficient of inelastic resistance.

Wind Pulsation

Pulsation Component of the Wind Load

Pulsation component of the wind load determines the motion of the structure with respect to the displaced equilibrium state the structure takes under the action of the average (static) component of the wind flow (see Sec. 2.4). The intensity of wind pulsations is proportional to the force of the static wind, therefore the calculation for wind pulsations requires references to the data on the static wind.

The method of analysis for the dynamic action of wind pulsations is determined by current SNiP on the basis of the hypothesis of the static independence (absence of correlation) at the motion of the structure for different modes of natural oscillations. This hypothesis does not always hold, and therefore it has been adjusted in Moscow City Building Codes MGSN [3,7] to take into account the statistical interaction between individual oscillation modes. It should be noted that it becomes meaningless to consider the results of the calculation for a certain mode of natural oscillations.

For asymmetric design models it is sometimes necessary to provide for the action of the wind first from one side and then from the other. It is often impossible to solve the problem considering the wind load as alternating, since when the wind changes its direction to the opposite one, the values of aerodynamic coefficients change as well (see for example model 9 from Annex E of SP 20.13330.2011). It is also necessary to consider two different dynamic loadings.

Average (Static) Component of the Wind Load

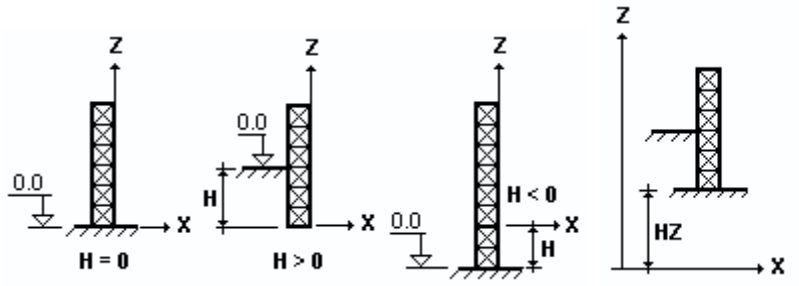


Figure 8.7-30.

Static (or average) component of the wind load is described by a separate static loading taking into account the position of the design model (see Fig. 8.7-30). Not only concentrated nodal forces but any local loads provided in **SCAD** are used when specifying it. Using the distributed loads enables to describe the wind action in accordance with the requirements of codes, according to the arrangements of wind loads with their aerodynamic coefficients (see Annex 4 of SNiP 2.01.07-85* and Annex E of SP 20.13330.2011).

Wind Pulsations according to SNiP 2.01.07-85*

Characteristics of loadings for the analysis for wind actions according to SNiP 2.01.07-85* are specified in the **Pulsation Component of the Wind Load (SNiP 2.01.07-85*)** tab (Fig. 8.7-31).

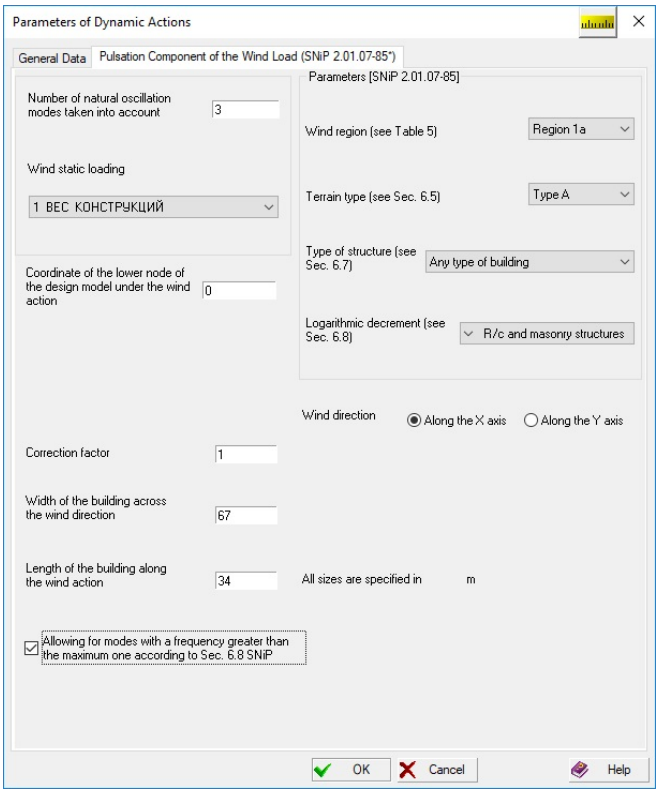


Figure 8.7-31. *The* **Pulsation Component of the Wind Load (SNiP 2.01.07-85*)** *tab*

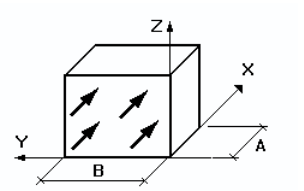


Figure 8.7-32.

When specifying the data it is necessary to pay attention to the following:

- the direction of the wind action has to be specified;
- in accordance with the specified direction it is necessary to specify the value of the width of the building across the wind direction and its length along the wind action (Fig. 8.7-32);
- it is necessary to select the number of the wind static loading from the list (it has to be selected even if this loading is the first in the list), having made sure that the selected wind static loading corresponds to the specified direction of the wind;
- correction factor (it is taken as 1 by default) must be greater than 0.

The number of modes of natural oscillations of a structure that have to be taken into account in the analysis is usually specified as 3 — for plane structures and 6 — for spatial ones. However, there are also such structures for which a much greater number of modes have to be taken into account.

For example, if the first modes of oscillations are orthogonal to the action of the wind. The number of modes taken into account in the result can turn out to be less, since if the natural frequency is greater than the limit frequency of the considered structure, then according to SNiP it is not taken into account.

All other characteristics are specified by selecting the necessary line in the lists.

Wind Pulsations according to Moscow City Building Codes (MGSN)

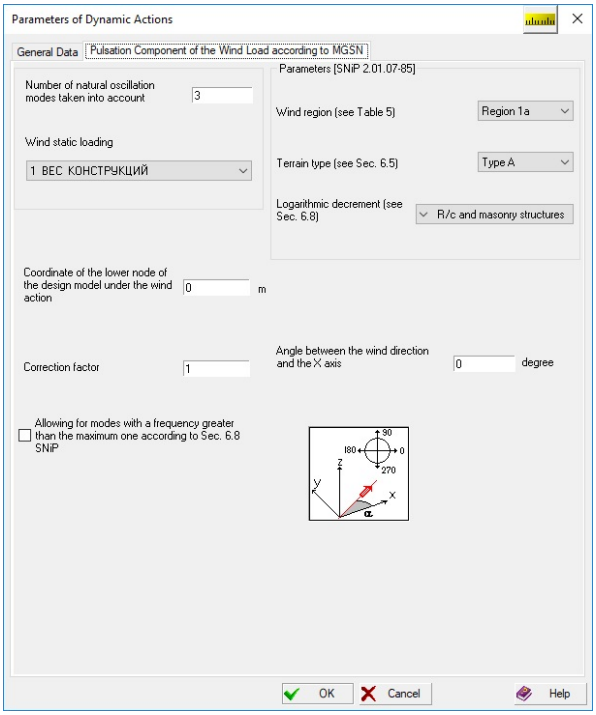


Figure 8.7-33. *The* **Pulsation Component of the Wind Load According to MGSN** *tab*

Characteristics of loadings for the analysis for wind actions according to MGSN [3] are specified in the **Pulsation Component of the Wind Load According to MGSN** tab (Fig. 8.7-33). The analysis is performed according to the method developed by V.A. Kucherenko Central Research Institute of Building Constructions. When specifying the data it is necessary to pay attention to the following:

- it is necessary to specify the direction of the wind action in accordance with which the correlation coefficients will be determined (see drawing 3 in SNiP 2.01.07-85*);
- it is necessary to select the number of the wind static loading from the list (it has to be selected even if this loading is the first in the list), having made sure that the selected wind static loading corresponds to the specified direction of the wind;
- correction factor (it is taken as 1 by default) must be greater than 0.

Wind Pulsations according to SP 20.13330 (Updated Edition of SNiP 2.01.07-85*)

Characteristics of loadings for the analysis for wind actions according to SP 20.13330.2011, SP 20.13330.2016 are specified in the **Pulsation Component of the Wind Load (SP 20.13330.2011, SP 20.13330.2016)** tab (Fig. 8.7-34).

When specifying the data it is necessary to pay attention to the following:

- the direction of the wind action has to be specified (along the X axis, along the Y axis or in the XoY plane; in the latter case the **In the X direction** and **In the Y direction** values define the wind direction vector (not necessarily normalized) in the horizontal plane);
- in accordance with the specified direction it is necessary to specify the value of the width of the building across the wind direction and its length along the wind action (Fig. 8.7-29);
- it is necessary to select the number of the wind static loading from the list (it has to be selected even if this loading is the first in the list);
- correction factor (it is taken as 1 by default) must be greater than 0.

When specifying the static component of the wind load you should keep in mind that a certain direction of the wind speed has already been specified. The program does not control the correspondence between the “static direction” and the direction specified in this dialog box. It is left to the user.

The number of modes of natural oscillations of a structure that have to be taken into account in the analysis is usually specified as 3 — for plane structures and 6 — for spatial ones. However, there are also such structures for which a much greater number of modes have to be taken into account.

For example, if the first modes of oscillations are orthogonal to the action of the wind. The number of modes taken into account in the result can turn out to be less, since if the natural frequency is greater than the limit frequency of the considered structure, then according to SP it is not taken into account.

All other characteristics are specified by selecting the necessary line in the lists.

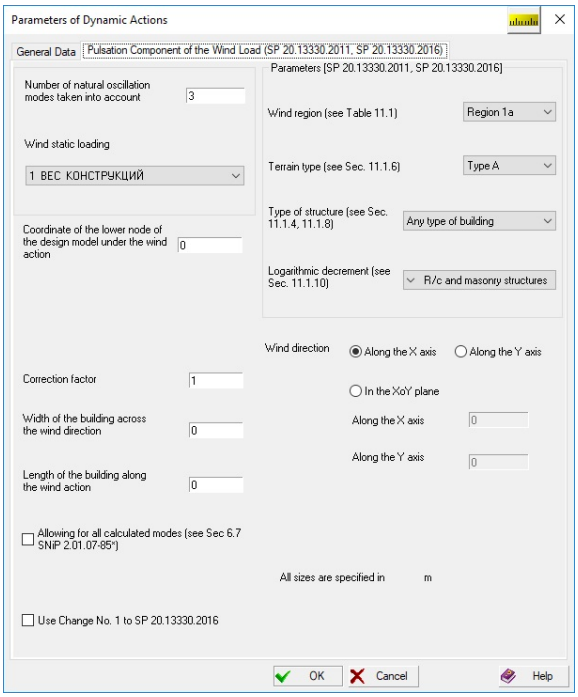


Figure 8.7-34. *The* **Pulsation Component of the Wind Load (SP 20.13330.2011, SP 20.13330.2016)** *tab*

s natural oscillation modes have to be taken into account in the dynamic analysis of buildings and structures with the second natural frequency less than the limit value according to Sec. 11.1.8 of SP 20.13330.2016. *s* is determined from the following condition $f_s < f_{lim} < f_{s+1}$. But the formulas for calculating the respective components of the dynamic load are not given. Therefore **the Calculation according to Sec. 6.7 of SNiP 2.01.07-85* to take into account all the calculated modes** checkbox is provided, which enables to use the recommendations of SNiP 2.01.07-85* for this case.

Pulse or Impact Action

The following types of calculations for impact and pulse actions are implemented in the program:

- impact;
- impact with history tracking;
- impact accounting for the effect of the mass of the impact body;
- pulse;
- pulse with history tracking.

The initial data for all above types of action are of the same type and are specified in the tab of the dialog box (see Fig. 8.7-35). It is necessary to specify here the number of oscillation modes taken into account and the coefficient of inelastic resistance. The button invokes the dialog box which enables to obtain the reference information on the coefficients of inelastic resistance of different structures (see Fig. 8.7-25).

Dynamic loads are specified with the help of the respective dialog boxes which are invoked by clicking the button in the **Loadings** section (Fig. 8.7-40).

Three modules for the calculation of impact actions are implemented in the program.

If the mass of the impact body is small, the impact action can be considered as a pulse the value of which is calculated as follows:

$$S = m\, v_0 (1 + k_0),$$

where m — mass of the impact body;

v_0 — velocity at the beginning of impact;

k_0 — coefficient of restitution, which depends on the properties of colliding bodies, and when there is no more accurate data it is taken from Table 5.1 of the Handbook on the Dynamics of Structures [11].

Therefore the calculation for impact is performed as for a single bell-shaped pulse with the value S. There are two possible variants of calculation:

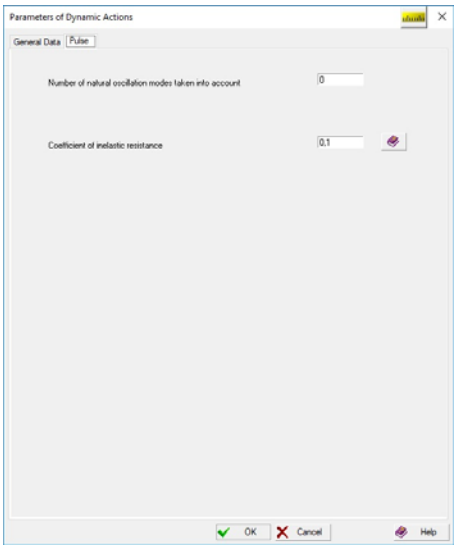


Figure 8.7-35. *The* **Pulse or Impact Action** *tab*

- the evaluation of the maximum is performed in the **Impact** module according to the formulas given in [11], when the pulse is specified by the mass of the impact body, its velocity, time of interaction and the coefficient of restitution (these formulas give an approximate estimation of the maximum reaction of the structure);
- tracking of the history of behavior of the system (integration of the equations of motion by the method of expanding by natural oscillation modes) and calculation of the maximum is performed in the **Impact with history tracking** module. You should understand that in this case the process takes much more time, but enables to obtain more accurate estimations of maximum deformations and forces. Pulse is specified by the mass of the impact body, its velocity, time of interaction and the coefficient of restitution.

Pulse and **Pulse with history tracking** modules, where an approximate estimation or integration of the equations of motion is performed, have similar differences.

If the mass of the impact body is large, the **Impact accounting for the effect of the mass of the impact body** module is used, and the evaluation of the maximum is performed without expanding by oscillation modes. Mass of the impact body (it is assumed that there is only one impact body), which is reduced to an equivalent kinetic energy concentrated at the impact point by the amount of motion, is included in the moving system. Moreover, an indirect impact can be considered. Then an angle between the direction of the impact and the normal to the area it is acting on has to be additionally specified.

Modal Analysis

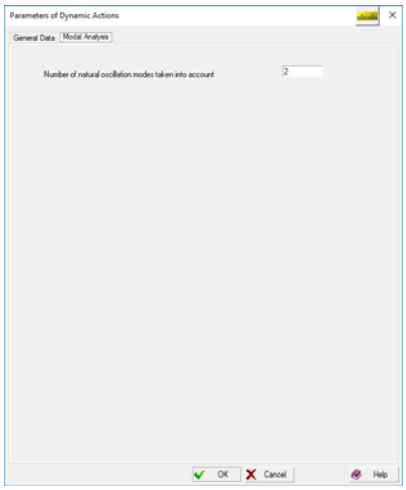


Figure 8.7-36. *The* **Modal Analysis** *tab*

The only characteristic necessary for performing the modal analysis, the number of natural oscillation modes taken into account, is specified in the **Modal Analysis** tab (Fig. 8.7-36).

If the frequency search range has to be specified instead of the number of modes, you should use the Lanczos method. Then the **Lanczos method** radio-button has to be selected and the **Analysis in the specified frequency range** checkbox has to be checked in the **General Data** tab of the **Parameters of Dynamic Actions** dialog box, and the values of frequencies of the beginning and end of the search interval have to be specified in the **From** and **To** text fields. The number of calculated frequencies will be not less than that specified in the **Modal Analysis** tab.

Harmonic Oscillations

One of the following problems can be solved when performing the analysis of a structure for harmonic excitation:

- analysis for the specified frequency;
- analysis taking into account the starting resonance;
- analysis for the specified frequency taking into account the possible error in the determination of natural frequencies.

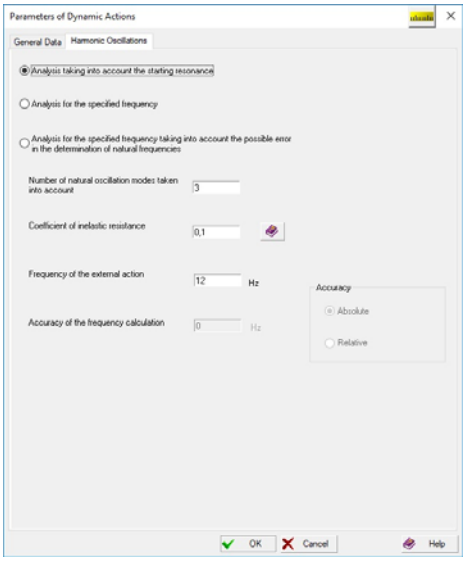


Figure 8.7-37. *The* **Harmonic Oscillations** *tab*

In the first case the frequency of excitation is assumed to be precisely specified and stable, the possibility of its interaction with all calculated natural frequencies of the structure is taken into account. The corresponding amplitude values of the real and imaginary components of the inertial forces under the action of harmonic force are determined by the formulas 2.4.13 given in Sec. 2.4.

In the second case it is assumed that the specified frequency is achieved by its gradual increase from the zero value to the specified upper value, and the possibility of resonant oscillations at natural frequencies less than the upper value of the excitation frequency is taken into account.

Finally, the third case assumes that due to the approximation of the design model (inaccuracy of the initial data, possible changes in the location of masses, etc.) actual natural frequencies can differ from the calculated ones by the value $\pm$ **ε**.

Then the following variants of the position of the specified frequency **ω** with respect to the calculated frequencies are considered:

- ω coincides with one of the calculated frequencies with the specified accuracy ω_k — the calculations are performed for a series of frequencies $\omega\frac{\omega_i}{\omega_k}$;
- $\omega \geq \omega_k$ and $\omega < \omega_{k+1}$ — in this case two series of frequencies are assumed $\omega_i - \varepsilon$ and $\omega_i + \varepsilon$;
- $\omega < \omega_1 - \varepsilon$ — assumed series of frequencies $\omega_i - \varepsilon$;
- $\omega > \omega_1 + \varepsilon$ — assumed series of frequencies $\omega_i + \varepsilon$.

Depending on whether the **Absolute** or **Relative** radio button is selected ε can be specified in absolute values (for example, in Hz) or in relative ones (in percent).

Amplitude values of the real and imaginary components of the inertial forces under the action of harmonic force, which are determined by the formulas 2.4.13 given in Sec. 2.4, are calculated for each of these series of "refined" natural frequencies.

The initial data are specified in the **Harmonic Oscillations** tab (Fig. 8.7-37).

The button invokes a dialog box which enables to obtain the reference information on the coefficients of inelastic resistance of different structures (Fig. 8.7-25). After selecting the necessary type of structure and clicking the **OK** button, you can transfer the respective value of the coefficient of inelastic resistance to the data of the loading. Moreover, if the **Other types of structures** item is selected in the **Type of structure** list, having specified one of the possible representations of information on the inelastic resistance (for example, logarithmic decrement of oscillations), you can obtain the necessary coefficient of inelastic resistance.

Direct Integration of the Equations of Motion

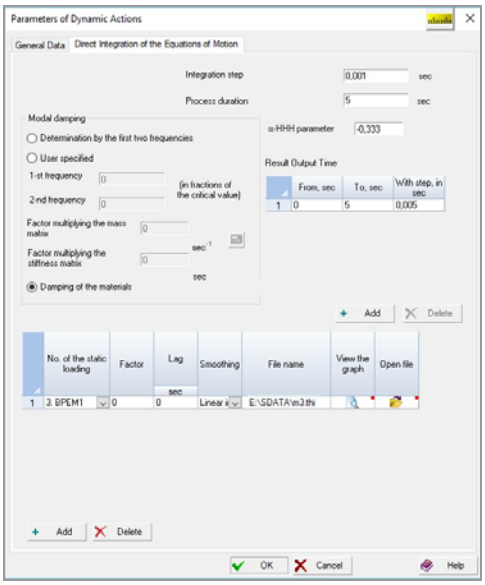


Figure 8.7-38. *The* **Direct Integration of the Equations of Motion** *tab*

It is necessary to perform the following actions to perform the calculation using the direct integration of the equations of motion:

- select the **Direct integration** radio button in the **Type of action** group of radio buttons in the **General Data** tab of the **Parameters of Dynamic Actions** dialog box;
- specify the name of the loading and parameters of conversion of static loads into masses;
- prepare the description of action in the **Direct Integration of the Equations of Motion** tab (Fig. 8.7-41);
- close the dialog box by clicking the **OK** button;
- invoke the **Parameters of dynamic loads** operation and specify the dynamic loads in the **Nodal Load with Time** dialog box.

The work of this dynamic module is specified in more detail in Chapter 15.

The nonlinear dynamic analysis problem is implemented in two formulations. A static load is first applied in both formulations, neglecting the forces of inertia and dissipation, and the stress-strain state of the system is determined. Then, small linear oscillations with respect to the static equilibrium position are considered in the first formulation of the problem

$$\begin{cases} \mathbf{M}\ddot{\mathbf{u}}_d + \mathbf{C}\dot{\mathbf{u}}_d + \mathbf{K}_t\mathbf{u}_d = \mathbf{p}_d(t) \\ \mathbf{u}_d(0) = \dot{\mathbf{u}}_d(0) = 0 \end{cases},$$

$\mathbf{u}_d$ is the nodal displacement vector due to the dynamic load,
$\mathbf{M}$ is the mass matrix,
$\mathbf{C}$ is the damping matrix,
$\mathbf{K}_t$ is the tangent stiffness matrix which is determined by the static analysis and remains constant during the entire dynamic analysis,
$\mathbf{p}_d(t)$ is the external dynamic force vector,
$\bar{\mathbf{u}}_d(t)$ are forced time-dependent displacements,
$\dot{\bar{\mathbf{u}}}_d(t)$ are time-dependent velocities defined as time derivatives of the forced displacements,
$\mathbf{K}^t_{21}$ is the tangent stiffness submatrix $\mathbf{K}_t$, covering the degrees of freedom with the specified forced displacements,
$\mathbf{C}_{21}$ is the dissipation submatrix $\mathbf{C}$, covering the degrees of freedom with the specified forced displacements.

The justification of the above equation for a linear problem is given in 2.

After solving a dynamic problem, the final solution is determined by the superposition of static and dynamic solutions:

$$\mathbf{u}(t) = \mathbf{u}_{stat} + \mathbf{u}_d(t), \quad f(t) = f_{stat} + f_d(t),$$

where $\mathbf{u}$ and f are the total displacement vector and the total force factor respectively (internal force, stress, strain at any point), $\mathbf{u}_{stat}$ and f_{stat} are the displacement vector and the force factor due to the static load respectively, f_d is the force factor due to the dynamic load, t is the arbitrary time point.

Nonlinear oscillations of a system with finite amplitudes are considered in the second formulation of the problem. Since the superposition principle is not applicable here, we cannot consider the static and

dynamic states separately, and then add them. Therefore, the nonlinear oscillation problem is formulated as follows:

$$\begin{cases} \mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + N(\mathbf{u}) + \mathbf{\Gamma}\mathbf{u} = \mathbf{p}_{stat} + \mathbf{p}_d(t) + \mathbf{\Gamma}\bar{\mathbf{u}}(t), \\ \mathbf{u}(0) = \mathbf{u}_0, \quad \dot{\mathbf{u}}(0) = 0, \end{cases}$$

$$\mathbf{p}_d(t) = \sum_{p=1}^{N_p} \mu_p \mathbf{k}_p^d f_p(t - t_0^p), \quad \bar{\mathbf{u}}(t) = \sum_{s=1}^{M_s} \mu_s \mathbf{k}_s^d f_s(t - t_0^s).$$

Here $\mathbf{u} = \mathbf{u}(t)$ is the total displacement vector due to both static $\mathbf{p}_{stat}$, and dynamic $\mathbf{p}_d(t)$ loads, $\mathbf{u}_0$ are displacements obtained by solving the nonlinear static problem $N(\mathbf{u}_0) = \mathbf{p}_{stat}$, $N(\mathbf{u})$ is the nonlinear operator which returns the vector of internal forces of the system. For physically nonlinear problems, the type of this operator is defined by the accepted theory of plasticity – deformation theory of plasticity or the yield theory. In the case of the deformation theory of plasticity, the operator $N(\mathbf{u})$ is determined by the σ – ε diagram for a homogeneous material, and for reinforced concrete – by the σ – ε diagrams for concrete and reinforcement. In the case of the yield theory, the operator $N(\mathbf{u})$ is determined by the von Mises yield surface for the homogeneous material and reinforcement, and by the Drucker-Prager or Geniev yield surface for concrete. The σ – ε diagrams for a homogeneous material, concrete and reinforcement determine the expansion (isotropic hardening) or contraction (softening – typical of concrete during cracking) of the yield surface and its translation in the principal stress space during the development of plastic deformations.

The static load $\mathbf{p}_{stat}$ and the heterogeneous initial conditions $\mathbf{u}_0$ are automatically transferred to the dynamic part of the problem, so **the user does not have to specify the static load in the initial data for the dynamic problem**.

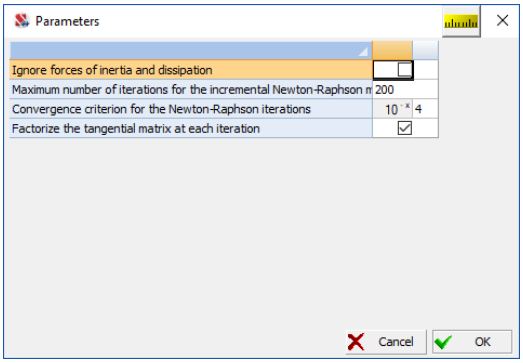
The dynamic load $\mathbf{p}_d(t)$ is represented as an expansion in time-independent vectors of the spatial configuration of the load $\mathbf{k}^d_p$, multiplied by the time functions $f_p(t - t_0^p)$, where t_0^p is the lag time. Here N_p is the number of reference loadings, and p is the current number of the static loading at the calculation for force actions, which sets the spatial configuration of the load $\mathbf{k}^d_p$. Coefficients μ_p act as scale factors.

In the case of forced displacements, the penalty parameter method is used and the diagonal matrix $\mathbf{\Gamma}$ is derived. If there are no displacements $\mathbf{\Gamma} = 0$. Similarly to the dynamic load the forced displacements are represented as an expansion in time-independent vectors of the spatial configuration of the load $\mathbf{k}^d_s$, multiplied by the time functions $f_s(t - t_0^s)$, where t_0^s is the lag time, M_s is the number of reference loadings, and s is the current number of the static loading at the calculation for the forced displacements, which sets the spatial configuration of the load $\mathbf{k}^d_s$.

Checking the **Nonlinear analysis** checkbox invokes the mode of solving nonlinear equations of motion.

The preceding static load is specified in the **Modeling of Nonlinear Loads** dialog box even if it was linear.

Clicking the button invokes the dialog box where you can set the parameters of the nonlinear analysis:



The ***Ignore forces of inertia and dissipation*** option enables to solve static problems based on the algorithm developed for nonlinear dynamics

$$\begin{cases} N(\mathbf{u}) + \mathbf{\Gamma u} = \mathbf{p}_{stat} + \mathbf{p}(t) + \mathbf{\Gamma}\overline{\mathbf{u}}(t) \\ \mathbf{u}(0) = \mathbf{u}_0 \end{cases}.$$

When $\mathbf{p}_{stat} = 0$, you should specify only one step with a load factor of zero to initialize physically nonlinear finite elements in the **Control the Stepwise Process** dialog box which can be invoked from the **Design Model** section of the project tree (the **Modeling of Nonlinear Loads** operation). In this case $\mathbf{u}_0 = 0$. This approach gives more flexibility in specifying the increment of the static load. For example, the following loading can be implemented: $\mathbf{p}(t) = \mathbf{p}_0 \cdot \sin(\pi t/4)$, where $t \in [0, 1]$. Then in the case of the uniform step Δt of the quasi-time parameter t the load increment $\Delta\mathbf{p} \to 0$ when $t \to 1$ (see Fig. 8.7-39), which can be useful when approaching the limit point of the equilibrium state curve.

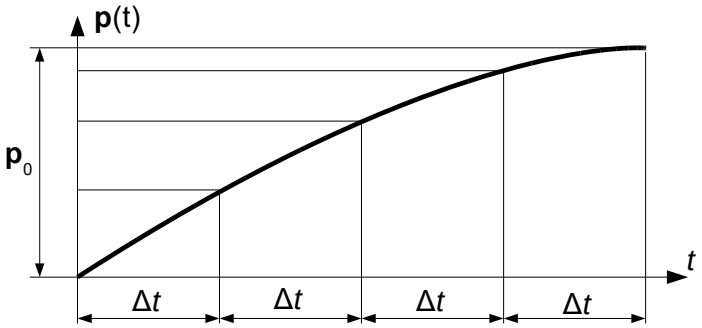


Figure 8.7-39

This option also enables to simulate the cyclic loading.

The maximum number of iterations of the Newton-Raphson method limits the number of iterations performed by the mentioned method in order to avoid looping at this time step in the absence of convergence of the iterative process. If the convergence with a given accuracy determined by the ***Convergence criterion of the Newton-Raphson iterations*** parameter is achieved, then iterations are terminated and the algorithm proceeds to the next integration step. The algorithm for solving the nonlinear dynamic problem is presented in 1.

The ***Arc-length strategy*** option can be enabled only for static problems (the ***Ignore forces of inertia and dissipation*** checkbox has to be checked) with one-parameter loading, i.e. $N_p = 1$ or $M_s = 1$, and leads to the fact that the lead parameter is not the load parameter t, but the arc-length in the state space, formed by the extension of the initial N – dimensional space of the finite element model (N is the number of degrees of freedom) per unit by adding the variable t. This approach enables to elegantly go through the limit points of equilibrium state curves for geometrically nonlinear problems and, in most cases, significantly simplify the approach to a limit point when solving elastic-plastic problems. The ***Maximum displacement value for the arc-length strategy*** parameter serves as a criterion for the algorithm termination. Once the translational displacement of any node of the finite element model in any direction exceeds this parameter, the calculation will stop. The details are given in 1.

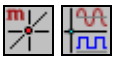
The **Linear search** checkbox starts the linear search algorithm at each iteration starting from the load step defined by the ***Time step number to start the linear search*** parameter and ending with the load step defined by the ***Time step number to stop the linear search*** parameter. In some cases, when the Newton-Raphson method does not converge, it is recommended to enable this option. This primarily refers to the statics problems (***Ignore forces of inertia and dissipation***). If there is no convergence in the dynamics problems, it is usually more effective to use the α – HHT method and reduce the integration step. The ***Accuracy of iterations of the linear search algorithm*** together with the ***Maximum number of linear search iterations*** parameter determines when to stop the iterations of the linear search algorithm. The linear search algorithm is terminated once the accuracy of its iterations is achieved or the specified number of iterations is exhausted, depending on what happens first, and the program will proceed to the next step of the Newton-Raphson iterations. The details are given in 1.

It is recommended to uncheck the **Factorize the tangent matrix at each iteration** checkbox for elastic-plastic deformation problems, since otherwise there is little chance of ensuring the convergence of the Newton-Raphson method. If this checkbox is unchecked, the tangent stiffness matrix is calculated only once at the beginning of each time or load step.

References:

1. S. Fialko, *Application of finite element method to the analysis of strength and bearing capacity of thin-walled concrete structures*, taking into account the physical nonlinearity, SCAD Soft, ASV, Moscow, 2018.
2. S. Fialko, V. Karpilovskyi, *Time history analysis formulation in SCAD FEA software*, J. of Measurements in Engineering, **6** (4), 2018, 173 – 180.

Specifying Dynamic Loads and Masses



Masses and dynamic loads can be specified only in an already created dynamic loading.

The **Inertial properties** operation is used to specify masses. When this operation is invoked the **Masses** dialog box appears (Fig. 8.7-40), where you have to select a method for specifying masses (in nodes or on elements), specify the weight of mass and its snap (the snap is specified if the mass is concentrated on the element), and snap the masses to nodes and elements according to the same rules as for loads after closing the dialog box.

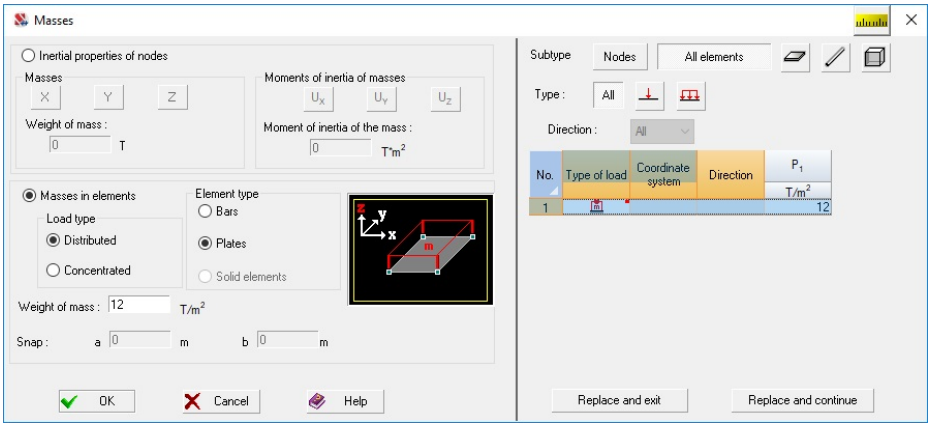


Figure 8.7-40. *The* **Masses** *dialog box*

The value of mass is characterized by its weight and is specified in tons (or other force measurement units selected for a particular problem), i.e. as a product of mass and the gravitational acceleration.

When specifying masses in nodes it is necessary to specify the directions of the corresponding dynamic displacements and the value of the mass (or the mass moment of inertia). Before specifying the characteristics of masses applied to plate elements it is necessary to select the **Plates** radio button.

Depending on the type of dynamic loading (pulse, impact, …) clicking the **Parameters of dynamic loads** button invokes one of the dialog boxes shown in Figures 8.7-41, *a–d*.

When specifying loads for different types of calculation for dynamic actions you have to:

- specify the data in the respective dialog box;
- click the **OK** button in the dialog box;
- select nodes or elements in the model the specified actions are applied to;
- click the **OK** button in the **Loadings** section.

After specifying masses or dynamic loads it is necessary to save the loading under a number previously specified for the dynamic loading.

The specified parameters of dynamic loads will be input into the tables of the respective dialog box. The procedure for correcting the data specified earlier does not differ from the similar operations when specifying other loads:

- select a line with the parameters you want to correct in the table, and the contents of the line will be input into the text fields;
- correct the data;
- use the **Replace and exit** or **Replace and continue** buttons depending on whether it is necessary to change other data.

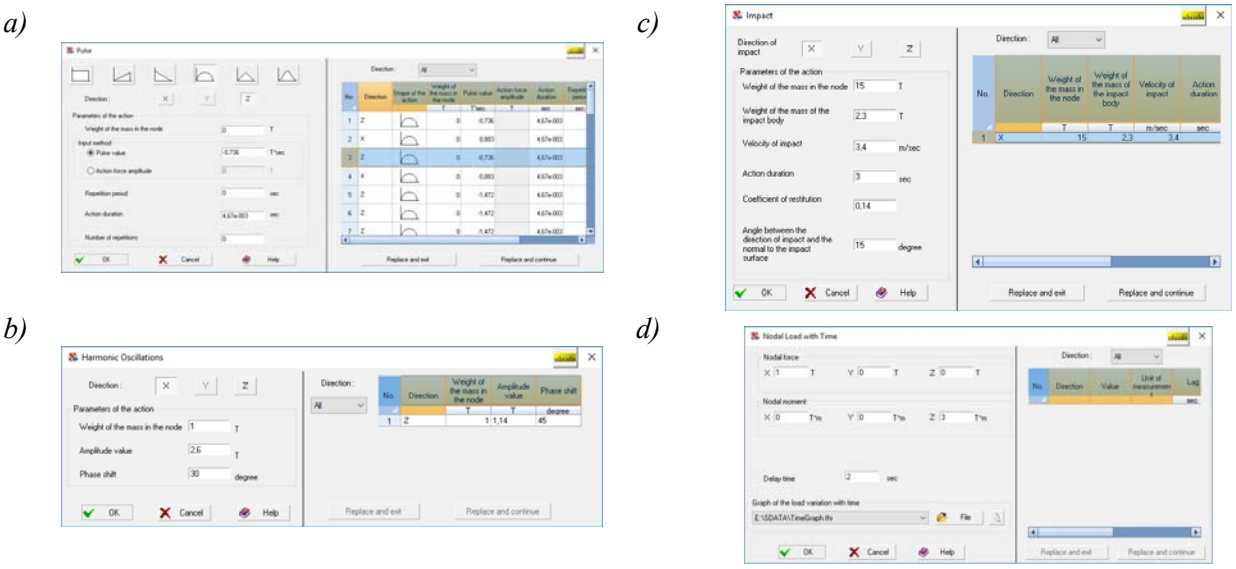
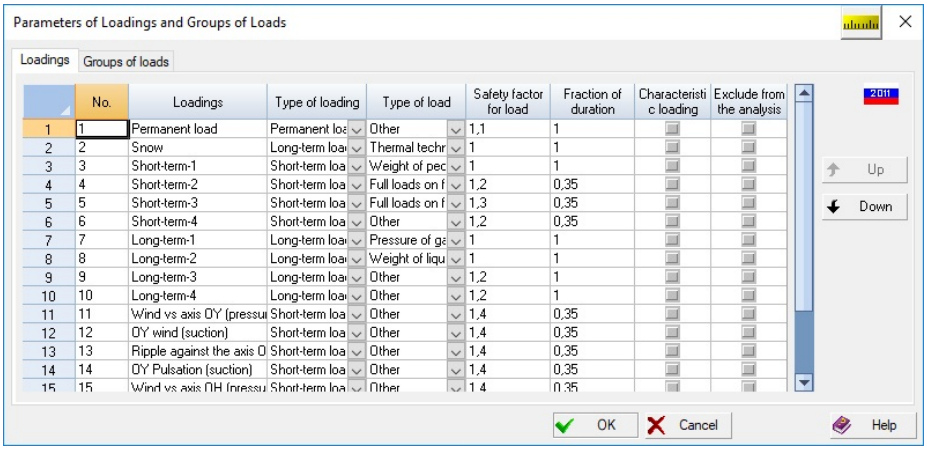


Figure 8.7-41. *Dialog boxes for specifying the characteristics of dynamic loads a) Pulse;b) Harmonic Oscillations; c) Impact; d) Nodal Load with Time*

8.8. Correcting the Parameters of Loadings/Groups of Loads

This operation enables to change the types of loadings, types of loads, safety factors and fractions of duration of one or several loadings and/or groups of loads. The operations are performed in a two-tab dialog box.



The first tab contains a table with a list of all loadings and their parameters, each of which can be changed, and the second one provides information on the groups of loads.

The numbering of loadings and groups of loads can be changed. In order to do this you have select the necessary loadings/groups of loads and move them in the list by clicking the **Up** or **Down** buttons.

If you check the checkbox in the **Exclude from the analysis** column, the respective loading will be excluded from the calculation.

References to Chapter 8

1. Dynamic Analysis of Structures for Special Actions. Designer's handbook / B.G. Korenev and I.M. Rabinovich— M.: Stroyizdat, 1981.
2. V.K. Egupov, T.A. Kamandrina, Seismic Analysis of Buildings. — Kyiv: Budivelnyk, 1969.
3. MGSN. Multifunctional Buildings and Complexes / Government of Moscow.— M.: 1994.
4. International Building Code of CIS. Construction in Seismic Regions. (Project), 2002. — Earthquake Engineering. Safety of Structures. 2002, No. 3, p. 27-54
5. S.V. Polyakov, Earthquake-resistant Building Structures.— M.: High school, 1983.
6. Codes for Earthquake-resistant Nuclear Power Plants. NP-031-01. Gosatomnadzor of Russia. — Bulletin of Gosatomnadzor of Russia. 2001, No.6, p.7-31.
7. Manual for the Analysis of Buildings and Structures for the Wind Action / Kucherenko V.A. Central Research Institute of Building Constructions.— M.: Stroyizdat, 1978.
8. SNiP 2.01.07-85*. Loads and Actions. Building rules and regulations / State Comm. for Construct. of Russia. – Moscow, 2001. — 44 p.
9. SNiP II-7-81*. Construction in Seismic Regions/Ministry of Construction of Russia, 1996. — 52 p.
10. SNiP II-7-81*. Construction in Seismic Regions /State Comm. for Construct. of Russia. – Moscow, 2000. — 44 p. + Annex 2 G 10 maps.
11. Handbook on the Dynamics of Structures. — M.: Stroyizdat, 1972. — 511 p.
12. Code of the Republic of Armenia. Earthquake Engineering. Design standards. SNRA II-2.02-94. –Yerevan. 1994.
13. Code of the Republic of Kazakhstan. SNiP b.1.2-4-98. Construction in Seismic Regions. Almaty. 1998, 41 p.
14. Construction in Seismic Regions of Ukraine. DBN B.1.1-12:2006. – K.: Ministry of Construction, Architecture and Housing and Communal Services of Ukraine, 2006. – 84 p.
15. Rules for Designing Residential and Public Buildings for Construction in Seismic Regions. SP 31-114-2004. – M.:, 2005. – 41 p.
16. SNiP RK 2.03-30-2006. Construction in Seismic Regions. Building Codes and Regulations of the Republic of Kazakhstan. – p. 41-74.
17. SP 14.13330.2011. Construction in Seismic Regions (Updated edition of SNiP II-7-81*) M.:, 2010. — 91 p.
18. SP 14.13330.2014. Construction in Seismic Regions (Updated edition of SNiP II-7-81*) M.:, 2014. — 126 p.
19. Code of the Republic of Armenia. Earthquake Engineering. Design standards. SNRA II-6.02-2006. –Yerevan. 2006.
20. SP 358.1325800.2017. Hydraulic structures. Rules of design and construction in seismic-prone regions. M.: 2018. — 36 p.
21. Birbraer A.N. Seismic Analysis of Structures. – St. Petersburg: Nauka, 1988 – 254 p.
22. State Building Code of Ukraine. Construction in Seismic Regions of Ukraine. DBN B.1.1-12:2014. – K.: Ministry of Construction, Architecture and Housing and Communal Services of Ukraine, 2014. – 110 p.
23. Building Codes and Regulations of the Republic of Kazakhstan. SNiP RK 2.03-30-2017* Construction in Seismic Regions. – Astana: JSC “KazNIISA”, 2018. – 74 p.
24. State Building Code of the Republic of Azerbaijan .Az. DTN 2.3-1 Construction in Seismic Regions. – Baku: 2010. – 66 p.
25. Building Codes of Turkmenistan. SNT 2.01.08-99 Construction in Seismic Regions. – Ashgabat, 2000
26. Building Codes and Regulations of the Republic of Tajikistan. SNiP RT 22-07-2018 Earthquake Engineering. – Dushanbe: Committee for Architecture and Construction, 2019. – 48 p.
27. Building Codes and Regulations of the Republic of Uzbekistan. KMK 2.01.03-96 Construction in Seismic Regions. – Tashkent: Goskomarchitektstroy of the Republic of Uzbekistan, 1976. – 103 p.
28. Building Codes and Regulations of the Republic of Uzbekistan. KMK 2.01.03-19 Construction in Seismic Regions. Tashkent. – Ministry of Construction of the Republic of Uzbekistan, 2019. – 111 p.
29. EN 1990 (2002) (English): Eurocode – Basis of structural. design [Authority: The European Union Per Regulation. 305/2011, Directive 98/34/EC, Directive 2004/18/EC]. – 116 p.
30. Codes of the Kyrgyz Republic SN KR 20-02:2018 Earthquake Engineering. Seismic Design Code. – Bishkek, 2009. – 104 p.
31. SP 14.13330.2018. Construction in Seismic Regions. Updated edition of SNiP II-7-81*. M. – Standartinform, 2018. – 116 p.
32. Modification No. 1 to SP 14.13330.2018 SNiP II-7-81* Construction in Seismic Regions. – URL: https://docs.cntd.ru/document/564509554
33. Modification No. 2 to SP 14.13330.2018 SNiP II-7-81* Construction in Seismic Regions. – URL: https://docs.cntd.ru/document/728350349
34. NTP RK 08-01.7-2017. Design of Structures for Earthquake Resistance. Part. General Rules. Seismic Actions. – Astana: 2017. – 95 p.

35. NTP RK 08-01.7-2021 (to SP RK EN 1998-1:2004/2012) Design of Structures for Earthquake Resistance. Part: Design of Civil Buildings. General Requirements. – 98 p.
36. SP 20.13330.2011. Loads and Actions. Updated edition of SNiP 2.01.07-85*. M. –2011. – 81 p.
37. SP 20.13330.2016. Loads and Actions. Updated edition of SNiP 2.01.07-85* (with modifications No. 1, 2, 3). URL: https://docs.cntd.ru/document/456044318.

9. FORUM Preprocessor

9.1. General Principles for Creating a Model

Unlike finite element models where a design model is made up from separate finite elements, the model in **FORUM** preprocessor is based on *enlarged elements* (*objects*) which are as close as possible by their purpose and name to the functional components of a real structure. They include such commonly used objects as *columns*, *beams*, *walls*, *floors* and *roofs*. Groups of objects can be combined into *modules.* They are usually combined based on their position when elements which model one storey of a building or one section of a multisectional building are included in one module, though other conditions of grouping are possible as well (for example, all columns of a building may be combined into one module).

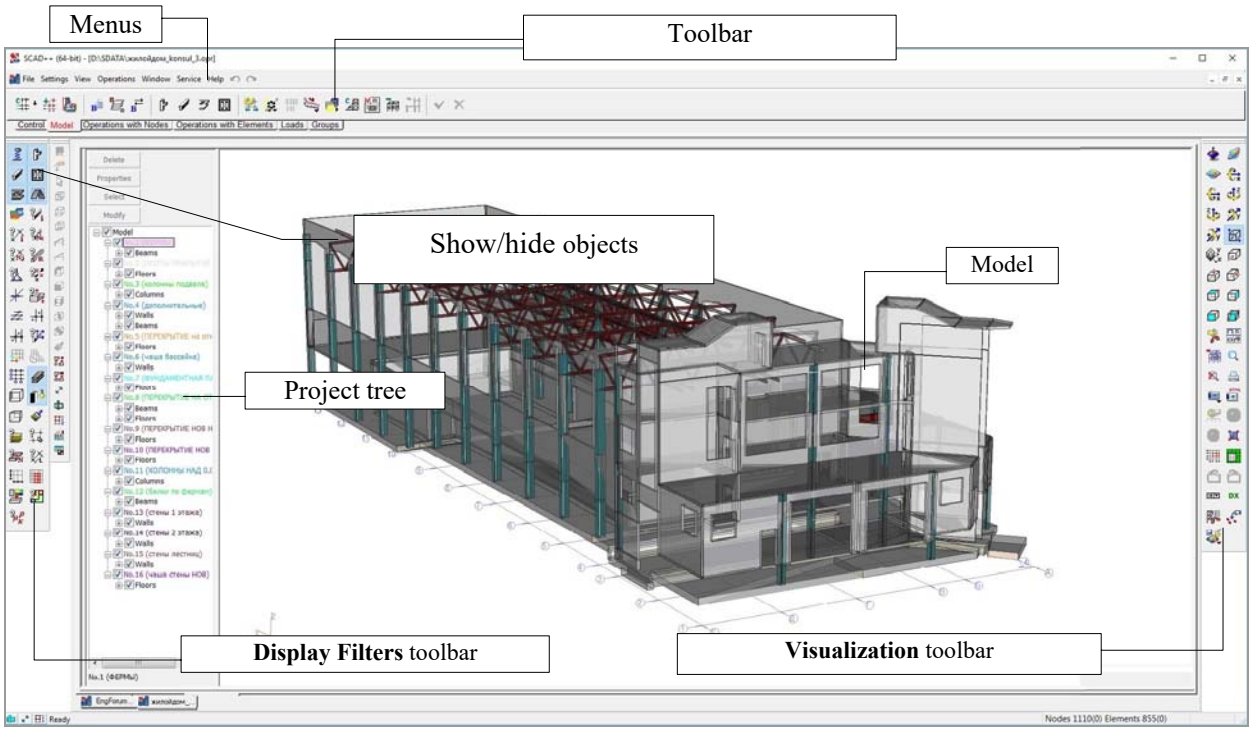


Figure 9.1-1. *General view of the* **FORUM** *preprocessor window*

This approach to modeling is often used in systems of computer-aided design of the architectural part of the project, which creates prerequisites for the transition from the architectural model to the design one and, to some extent, facilitates the generation of geometry of the design model, for which the architectural model can serve as a sub-base.

An obvious advantage of integrated models is their relative simplicity compared with the corresponding finite element models, which greatly facilitates the control of large models.

FORUM preprocessor included in **SCAD++** is used to both generate integrated models and finish models imported from architectural systems.

The presence of an integrated model consisting of objects enables to automate the process of combining finite elements into structural members (by belonging to one object) and into groups of elements. The transition from an integrated model to a finite element design model of **SCAD** complex is performed by an automatic or user-managed transformation of objects into sets of finite elements.

The control of the preprocessor is performed with the help of menus, toolbars, filters and dialog boxes. To make it easier for the users of **SCAD** complex to master the preprocessor the rules for working with the program and many operations adopted in **SCAD** are saved where it is reasonable. Therefore, the description of such "common" operations (such as the removal and transfer of nodes, the transfer of the origin of the global coordinate system, etc.) is not given in this section.

Toolbar

The toolbar contains six sections: **Control**, **Model**, **Operations with Nodes**, **Operations with Elements**, **Loads**, and **Groups**. Switching between sections is performed by clicking on "tabs" with their names.

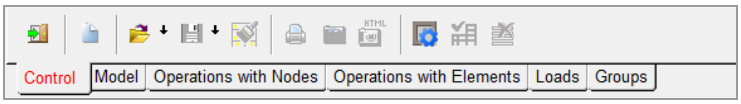


Figure 9.1-2. *The* **Control** *section of the toolbar*

The **Control** section (Fig. 9.1-2) includes the following operations:

— **Exit** — finish the work with the program;

— **New project** — create a new project;

— **Open project** — load an existing project;

— **Save** — save the project under its current name;

— **Pack data** — the deleted nodes and elements, nodes not belonging to elements, "empty" modules (modules not containing any elements) are removed from the project;

— **Print** — print the current image of the model (or its fragment);

— **Save the screen image** — the current image of the model is saved as a Windows metafile (in a file with extension **.emf** or **.wmf**) or raster image (**.bmp**, **.png**, ...) and can be used, for example, when preparing a report;

— **Save as HTML** — the current image of the model is saved as an HTML-file and can be used for example to put it on the Internet;

— **Save the fragment of the model** — save the fragment of the model as an individual project;

— **Settings** — specify the units of measurement and the type of representation of numerical information, select fonts, ...;

— **Express control of the model** — express-control of some parameters of the model;

— **Correct elements** — invalid elements (in terms of shape) are deleted or replaced;

— clear the list of messages.

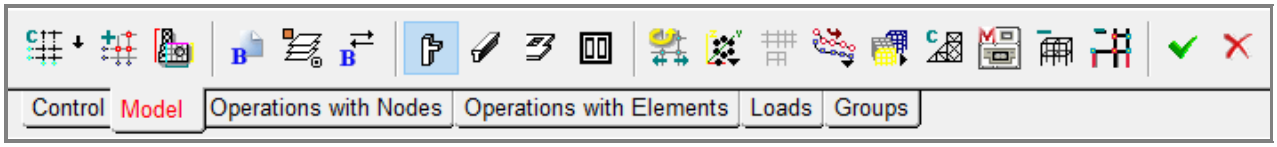


Figure 9.1-3. *The* **Model** *section of the toolbar*

The following operations can be invoked from the **Model** section (Fig. 9.1-3):

— **Specify grid lines** —specify the parameters of grid lines;

— **Generate nodes on the grid** — specify the input area for nodes at the intersection points of grid lines;

— **Create standard structures** — such as for example a staircase, or a truss;

— **Create module** —specify parameters and create a new module;

— **Control the modules** — a table with a list of modules similar to the tables of control of the groups of nodes and elements;

— create columns, beams, floor slabs and walls, respectively;

— **Geometric transformations**;

— **Copy** — copy modules and elements;

— **Copy the fragment of the model**;

— Creation and modifications of **trajectories**;

— Assemble the model from several models (**Assembly mode**);

— **Generate the resulting project** — automatic generation of a mesh of finite elements;

— **Export data to Monolit**;

— **Delete data on the model**;

— **Delete selected objects;**

— **Operations on modules** — transfer elements from one module to another;

— **Save the fragment of the model** — save the fragment of the model as an individual project;

— **Confirmation (OK)** — command for performing the selected operation;

— **Rejection** — cancel the selected operation.

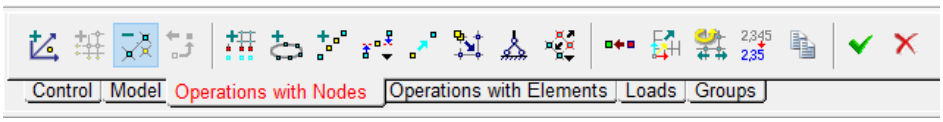


Figure 9.1-4. *The* **Operations with Nodes** *section of the toolbar*

The **Operations with Nodes** section of the toolbar (Fig. 9.1-4) includes the following operations:

— **Create nodes** — create nodes by specifying their coordinates;

— **Generate nodes on the grid** — specify the input area for nodes at the intersection points of grid lines;

— **Delete nodes**;

— **Restore deleted nodes**;

— **Create nodes at the specified distance from the selected ones** — create new nodes at the specified distance from the selected ones;

— **Generate nodes along an arc** — create nodes along a circular arc lying in the given plane;

— **Create additional nodes between nodes** — create nodes which divide the segment between the existing nodes according to the specified rule;

— **Move nodes to the given plane**;

— **Move nodes** — move the selected nodes by the specified distance;

— **Merge duplicate nodes** — delete nodes the coordinates of which coincide with the coordinates of the nodes created earlier;

— **Install constraints in nodes** — the operation is similar to that described in Section 7.4;

— **Merged displacements** in nodes — the operation is similar to that described in Section 7.4;

— **Replace a node in elements**;

— **Move the origin**;

— **Geometric transformations**;

— **Round the node coordinates**;

— **Copy node properties**;

— **Confirmation (OK)** — command for performing the selected operation;

— **Rejection** — cancel the selected operation.

Figure 9.1-5. *The* **Operations with Elements** *section of the toolbar*

The **Operations with Elements** section of the toolbar (Fig. 9.1-5) includes the following operations:

— **Delete elements**;

— **Restore deleted elements**;

— **Delete duplicate elements** — delete the elements of one type, all the nodes of which coincide with the nodes of the elements created earlier;

— **Move elements** — move the selected elements by the specified distance;

— **Specify subsoil parameters** (see Section 7.3);

— **Specify the orientation of the local coordinate axes of the elements** — rotate the section of the element by the specified angle;

— **Specify the directions of the stress output for plates** (see Section 7.3);

— **Change the stiffness properties** — change the stiffness properties of the selected elements;

— renumber rigidities;

— specify the reinforcement layout of plate elements (see Chapter 19);

— specify the reinforcement layout of bars (see Chapter 19) ;

— **Install hinges and delete hinges** — specify and delete hinges and sliders;

— **Move the plate midplane** — change the position of the slab or wall by the parallel translation of the midplane by the specified distance;

— **Bar division** — divide beams and columns into several elements;

— **Division of bars taking into account the intermediate nodes** — divide beams and columns into several elements taking into account all the nodes they intersect;

— **Divide bars at the intersection point** — divide intersecting elements;

— **Connect walls and floors** — extend walls to their intersection line (including floors);

— **Correct the description of openings** — combine intersecting openings;

— **Delete openings** — delete openings in the elements of slabs and walls;

— **Extend the bar to the plate** — extend the bar to the specified plate element;

— **Chamfer bars** — extend the bars to their intersection point;

— **Divide bars by a polygon**;

— **Expansion of the plate by an offset from the faces** — increase the overall dimensions of the plate by an offset from all its faces by the specified distance;

— **Division of floors** and walls by beams and columns lying in their plane;

— **Divide plates along the intersection line** — divide the intersecting plate elements along their intersection lines;

— **Operations with a pair of planar objects** — combine, intersect, ... plate elements lying in one plane;

— **Create surfaces of revolution** — create surfaces of the cylinder, cone and sphere type from plane elements;

— **Create elements along an arc** — create beams along a circular arc;

— **Change the types of elements** — the ability to change the type of the element (wall, partition, column, floor, roof);

— **Create a 0-element**;

— **Create constraints of finite rigidity** — create and specify parameters for constraints of finite rigidity;

— **Create an elastic constraint** — create and specify parameters for elastic constraints;

— **Create single-node peripheral elements of the plate** — create and specify parameters for single-node peripheral elements of the plate on the elastic subgrade;

— **Create two-node peripheral elements of the plate** — create and specify parameters for two-node peripheral elements of the plate on the elastic subgrade;

— **Create cable-stayed elements** — create and specify parameters of cable-stayed elements;

— **Rigid bodies** — create, correct and specify parameters of rigid bodies;

— **Create unilateral constraints** — create and specify parameters of unilateral constraints;

— **Copy element properties**;

— **Confirmation (OK)** — command for performing the selected operation;

— **Rejection** — cancel the selected operation.

Loads

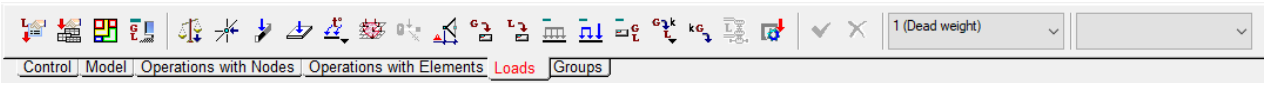


Figure 9.1-6. *The* **Loads** *section of the toolbar*

The **Loads** section of the toolbar (Fig. 9.1-6) includes the following operations:

— **Parameters of the load specification mode**;

— **Tables of loads**;

— **Specify the types of rooms**;

— **Delete loads**;

— **Confirmation (OK)** — command for performing the selected operation;

— **Rejection** — cancel the selected operation.

Groups

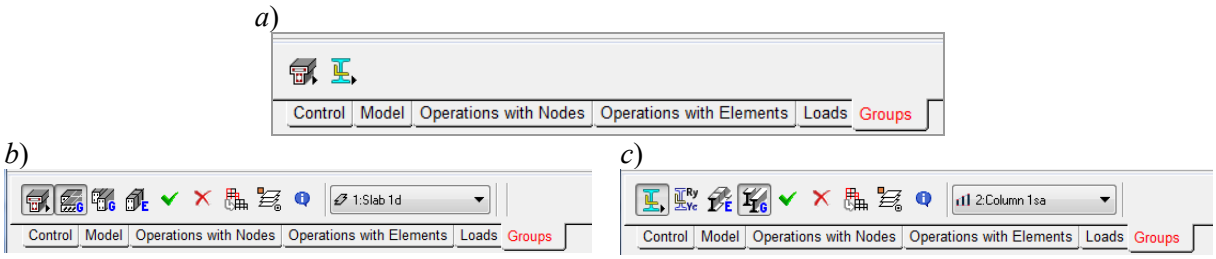


Figure 9.1-7. *The* **Groups** *section of the toolbar* (*a*),
toolbars for specifying structural groups of
elements of r/c (*b*) and steel (*c*) structures

The **Groups** section of the toolbar (Fig. 9.1-7) includes two operations — for specifying structural groups of elements of r/c and steel structures. The description of rules for creating and working with the respective structural groups is given in Sections 19 and 20.

9.2. Structure of the Model

The model in its general form is represented as a four-level tree structure — "model" (Fig. 9.2-1, *b*), "module" (Fig. 9.2-1, *c*), "group of the elements of the same type", "element" (Fig. 9.2-1, *d*). The modules and elements can have unique names (identifiers). Moreover, the elements have serial numbers in the model. Description of peculiarities of the data representation at each level and of the possible operations with the objects of these levels is given below.

Structuring enables to create a model as a collection of structural members of a particular type, and the elements (including the finite elements) will belong to a particular structural member at all stages of the transformation – generation of a design model, calculation, analysis of the results and design.

The structure of the model is displayed in the form of a project tree (Fig. 9.2-1, *a*), which is created automatically during the generation of the model.

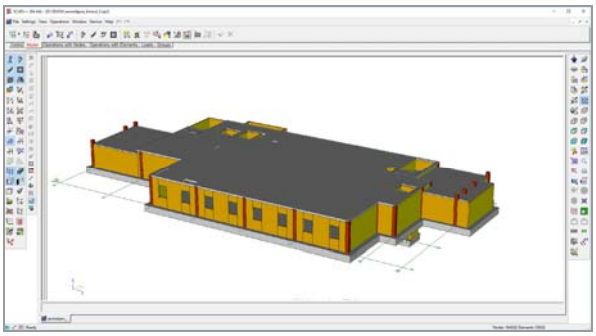
Nodes

Positioning elements in the model is performed by snapping them to the *nodes*. Nodes can be created in the model as intersection points of *grid lines*, or with the help of the respective operations.

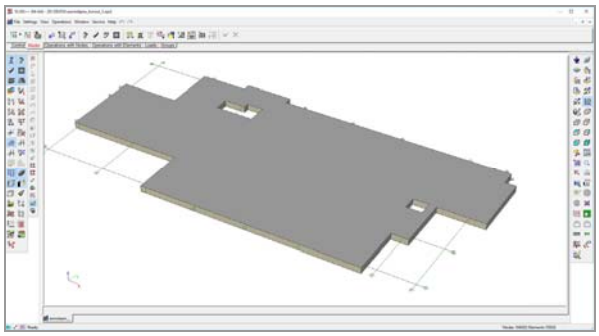
Modules

The modules can have different physical meaning, for example storeys, separate parts of the structure, parts of the building divided by an expansion joint etc. There must be at least one module in the model.

Before introducing elements into the model a module that includes these elements must be created. In some cases modules are created automatically. This occurs when you create a new model, perform the operation of copying a module, and also when importing data from architectural systems (for example, ArchiCAD®).



c) module (storey)



d) element (floor)

Figure 9.2-1. *Structure of the model*

Elements

Depending on the purpose of the model objects (elements) in the structure can be classified into the following five types: ***column***, ***beam***, ***wall***, ***floor*** and ***roof***. The first two types are two-node objects, and are modeled by bars in the finite element design model. The last three ones are planar and can have any number of nodes, include inner contours modeling openings and holes, and in the process of transformation into a design model are divided into three- and four-node shell elements (by default), plates or deep beams.

Objects of the ***roof*** type can get into the model when importing data from architectural design systems, or when creating floors (**Floor/roof** button). Unlike floors you cannot specify loads on roofs in **FORUM**. If it is necessary to specify such loads, it should be done in a finite element model.

The position of traditionally vertical (column, wall) or horizontal (floor, beam) elements in the model is not controlled in **FORUM** preprocessor allowing you to create a horizontal wall or column, or a vertical floor.

When creating the elements you have to specify their geometric parameters and characteristics of the material they are made of. The program enables to copy the properties of the elements created earlier when creating the new ones and correct the geometry of the elements.

9.3. Creating a New Model

The following procedure is recommended for creating a new model:

- click the **New project** button — in the **Control** section of the toolbar;
- specify the identification data on the project and the units of measurement of the main values in the **New Project** dialog box (hereinafter it is assumed that the dialog box is closed after specifying the data by clicking the **OK** button);
- specify the name of the file in which the project will be saved in the **Save As…** dialog box;
- click the **Specify grid lines** button — in the **Model** section of the toolbar;
- specify the parameters of grid lines in the **Specify Grid Lines** dialog box;
- click the **Generate nodes on the grid** button — in the **Model** section of the toolbar;
- specify the boundaries of the area within which the nodes will be created on the intersections of grid lines in the **Generate Nodes at the Intersection of Grid Lines** dialog box;
- click the **Create module** button — in the **Model** section of the toolbar;
- specify the name of the module and select the color which will be used to display the module in the model in the **Parameters of the Module** dialog box. After performing this operation the project tree will contain a section (empty so far) with the elements of a new module ;
- click the button with an image of the element you are creating (for example, column) — in the **Model** section of the toolbar;
- specify the parameters of the element (material, shape and sizes of the section, rules for specifying) in the **Rigidity of Bar Elements** dialog box;
- create the element.

If the model consists of several modules, then after creating the elements included in the current module, you have to create a new module and repeat the operations of creating elements.

New Project

Creating a new project in **FORUM** preprocessor is similar to creating a new finite element project. You just have to specify the **FORUM** type in the **New Project** dialog box (Fig. 9.3-1).

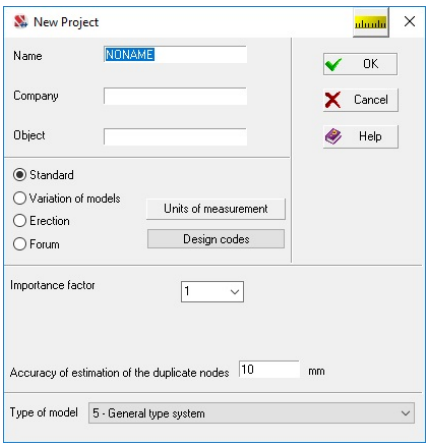


Figure 9.3-1. *The* **New Project** *dialog box*

Creating a New Module

Creating a model in the **FORUM** preprocessor begins with creating a module (button in the **Model** section). The **Parameters of the Module** dialog box will appear (Fig. 9.3-2) where you have to specify the name and display color.

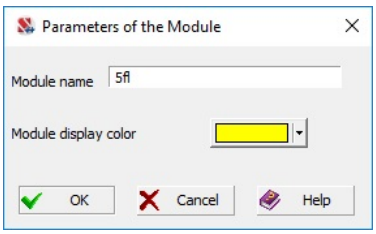


Figure 9.3-2. *The* **Parameters of the Module** *dialog box*

A module can be created at any step of creating a model. The newly created module automatically becomes **active**, i.e. it will contain all the elements created at once after creating the module.

It should be noted that when the elements are created, they always get only into the active module, the name of which is highlighted in the project tree. In order to make the module active, you just have to click on its name in the project tree.

The parameters (name and color) of the active module can be edited after clicking the **Properties** button in the upper part of the **Project tree** panel (Sec. 9.7).

Create Elements

As noted above the software enables to create five types of elements: *columns*, *beams*, *walls*, *floors* and *roofs*. It is necessary to specify the physical and mechanical characteristics of the material and geometric parameters for each type of element. Moreover, each element can have an identifier (name). The serial numbers of elements in the model are automatically assigned when these elements are created.

Columns

After invoking the operation of creating columns the **Rigidity of Bar Elements** dialog box appears (Fig. 9.3-3). The **General Data** tab of this dialog box enables to specify the grade of the column (or of the group of columns), type of the section (parametric, rolled steel, …), to select the input method (with the fixed height or by two nodes). The specification of the parameters (shape of the section and physical properties) is almost the same as the operation of specifying the rigidities of bar elements in a finite element model (see Section 7.1).

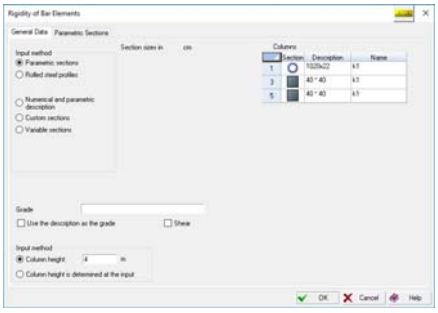


Figure 9.3-3. *The* **Rigidity of Bar Elements** *dialog box*

There are two methods for creating columns in the model. If the **Column height** radio button is selected, the column is created by clicking on one snap node of the column, and the column height is defined by the value specified in the text field (if the value is negative – the column will be directed downwards from the snap point). If the **Immediate execution** option is selected in the **Select Cursor** menu, the column will be created and displayed on the screen immediately after clicking on the snap node. Naturally, columns created in such a way are always vertical.

If the **Column height is determined at the input** radio button is selected, the column will be created by snapping to two nodes (a "rubber band" is stretched from the first node to the second one). If this method is used, the control of the position of the column (vertical or inclined) is not performed, which enables to implement non-standard situations. When this method of creating columns is used, you can use the **Create a series** checkbox to add a series of columns. They will be created until you click the button in the toolbar (or press the **Enter** key on the keyboard). The **Taking into account the intermediate nodes** checkbox enables not only to create new columns, but also to automatically divide each of them into several elements taking into account all the intermediate nodes lying along the line of the column.

Beams

Beams are created in the same way as columns, but always by stretching the “rubber band” between two nodes. There are no restrictions on the position of the beam in the model.

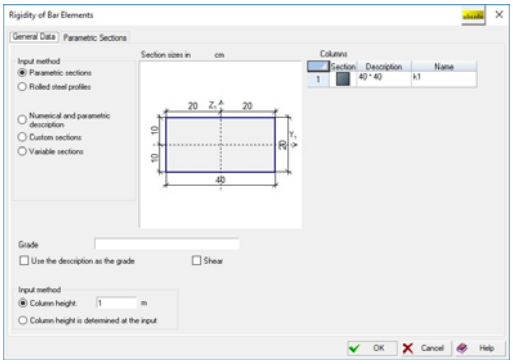


Figure 9.3-4. *The* **General Data** *tab*

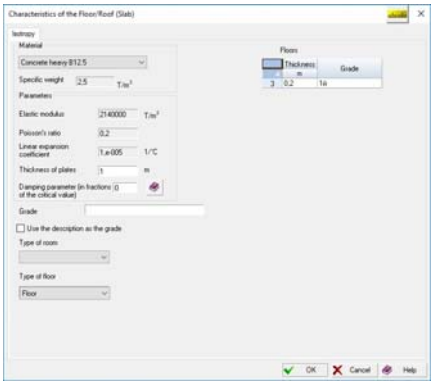


Figure 9.3-5. *The* **Characteristics of the Floor/Roof (Slab)** *dialog box*

Floors

A floor is created as a *flat* multi-node element of an arbitrary shape. It is done with the help of a "rubber band" which is stretched from node to node and ends by clicking on the first node of the floor for the second time. The position of the floor in the model is not controlled – it can be inclined or even vertical. This enables to use of a floor element to create elements of roofs and even walls. Holes and openings can be made inside the contour of the floor, the shape and the number of which is not limited.

After invoking the operation of creating a floor the **Characteristics of the Floor/Roof (Slab)** dialog box appears (Fig. 9.3-5). This dialog box enables to specify the grade of the floor (or the group of floors), its thickness and material. The dialog box contains a table with the parameters of the floors created earlier.

In those cases when an automatic specification of loads is assumed (see Section 9.9), it is necessary to specify the **Type of room** the created floor belongs to. The type is selected from the respective drop-down list.

A floor can be created by selecting bars forming its contour. In this case before performing the considered operation you have to select the respective bars and check the *Create from the selected bars* checkbox. The floor will be created after closing the dialog box by clicking the **OK** button.

The selected bar elements must meet the following requirements:

- lie in one plane;
- have a non-zero length;
- be continuous, i.e. form a closed line without self-intersections.

Walls

A wall is created as a *flat* multi-node element of an arbitrary shape. There are three methods for creating a wall in the model. In the *first* case it is done with the help of a "rubber band" by clicking on all nodes of the wall and ends by clicking on the first node of the floor for the second time. In this case the position of the wall in the model is not controlled.

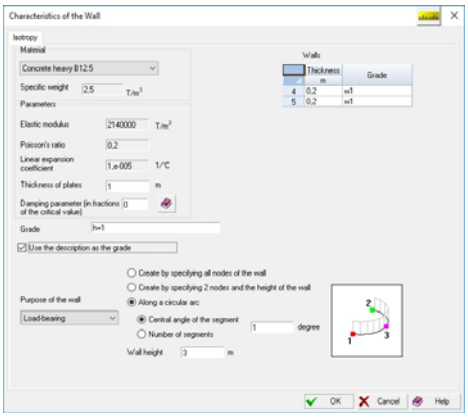


Figure 9.3-6. *The* **Characteristics of the Wall** *dialog box*

In the *second* case a wall with a fixed height is considered. In order to create it you just have to click on two nodes (a "rubber band" is stretched between them).

The third method of creating walls — along a circular arc. In this case a wall is considered as a set of rectangular segments the number of which depends on the specified parameters. To create a wall you have to click on three nodes (in the beginning, end, and on the arc) and click the **OK** button in the toolbar. After invoking the operation of creating a wall the **Characteristics of the Wall** dialog box appears (Fig. 9.3-6) where you have to specify the parameters of the wall and select the input method. You have to specify the grade of the wall (or the group of walls), its thickness and material in this dialog box.

Holes and openings can be made inside the contour of the wall, the shape and the number of which is not limited.

In those cases when an automatic specification of the load is assumed (see Section 9.9), you have to select whether a wall is **Load-bearing** or it is a **Partition** in the **Purpose of the wall** drop-down list. This will enable to generate a model taking into account the respective properties of the wall. In the case of a partition only the load will be included in the design model.

The dialog box contains a table with the parameters of the walls created earlier.

Copying Models and Modules

This operation enables to copy a model or its fragment taking into account the established rules and settings for copying modules. The controls in the **Copy the Model** dialog box (Fig. 9.3-7) are generally similar to those used for copying finite element models (see Sec. 5.14).

When copying a model there are four options for working with modules:

- **Each module – into a new module** — each module of the original model generates **n** new modules, where **n** is the number of copies;
- **Each module – into an original module** — all copies of the elements of each original module are written to the corresponding positions (by type) in the "user" original module;
- **All modules – into a new module** — all copies of the elements of the original modules are written to the corresponding positions (by type) in one automatically generated new module;
- **All modules – into the specified module** — all copies of the elements of each original module are written to the corresponding positions (by type) in the existing module selected from the list.

When copying nodes, columns and beams the type of the selected objects can be converted according to the following rules:

- if nodes have been selected in the model-prototype, a bar element connecting a node with its respective node in the copy is generated from each of these nodes;
- if column and/or beam elements have been selected in the model-prototype, each selected bar will generate a floor slab element, the size of which in the direction of copying will be equal to the specified step of copying .

It should be noted that after copying the selected bars the element type does not depend on the type of the prototype, and it can be changed when necessary using the respective operation in the **Operations with Elements** toolbar — .

The stiffness properties of elements obtained after converting the element type are specified traditionally using the operations for changing the stiffness — and of plates and bars respectively.

If planar elements are selected in the model, their type is not converted, and the elements are copied in accordance with the specified settings.

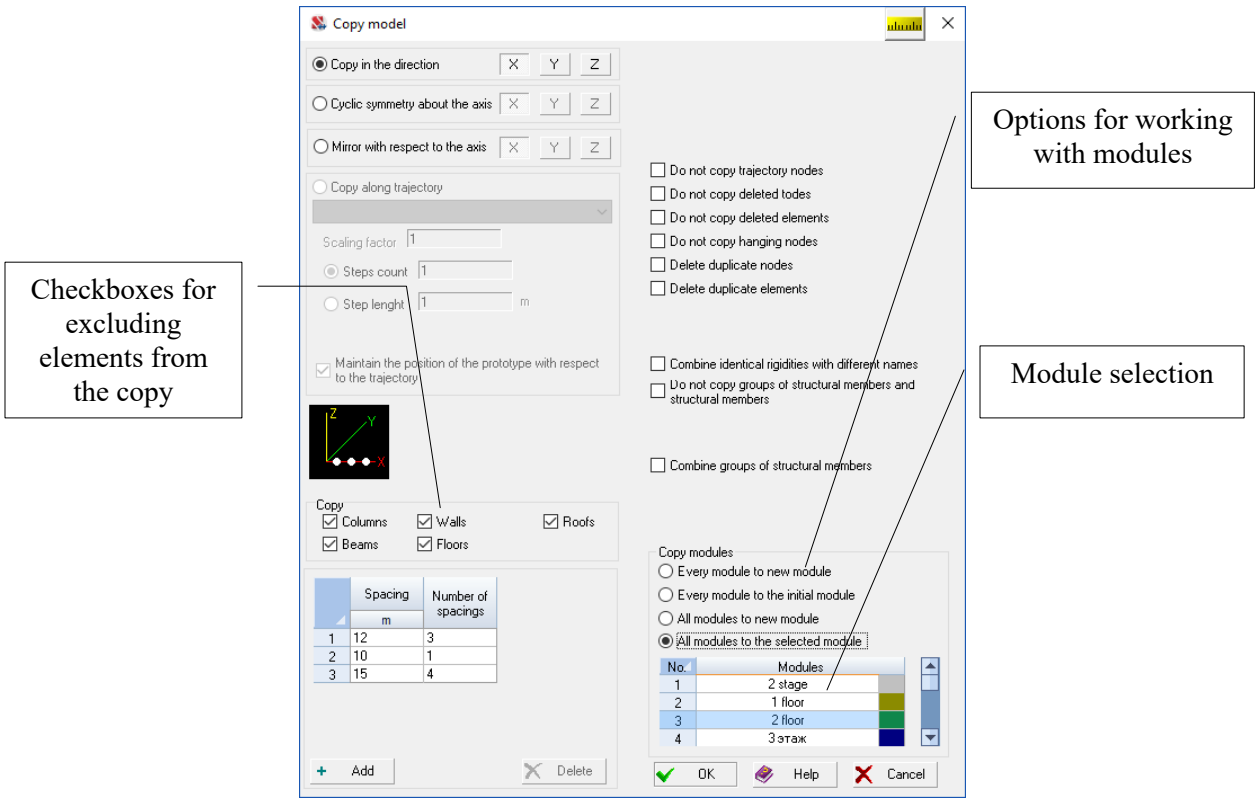


Figure 9.3-7. *The* **Copy the Model** *dialog box*

Some types of elements can be excluded from the copy by unchecking the respective checkboxes (all types of elements are copies by default).

In those cases when a fragment of the design model is displayed in the window or display filters are used to hide elements of some types and this operation is invoked, a drop-down menu appears where you can select whether to copy the whole model or its visible fragment. In the first case, i.e. when the whole model is copied, the operation will be applied to all nodes and elements of the design model according to the above rules regardless of whether they are included in the visible fragment or not. When the visible fragment is copied, the above actions will be performed only for the nodes and elements of the fragment with a possibility of converting their types.

Serial numbers are automatically assigned to the modules but not names. If the modules were named (the **Properties** button in the **Project tree**), these names are saved for copies as well, and only the numbers of the new modules change. The names and colors for the modules are assigned by the user.

The copying is performed after closing the dialog box by clicking the **OK** button.

Copying the Fragment of the Model

Besides copying a full model it is possible to copy its fragment as well. Any part of the model consisting of the selected elements can be considered as a fragment. When you copy a fragment, the type of the selected objects is not converted. Otherwise, the copying rules are the same as for copying a full model.

Transferring Elements from One Module to the Other

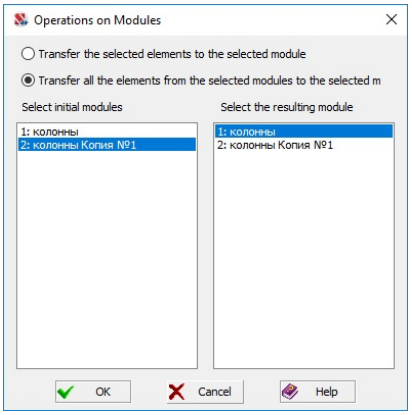


Figure 9.3-8. *The* **Operations on Modules** *dialog box*

This operation enables to reorganize the content of the module — "transfer" in this case means that the element included in the *Initial* module will be included in the *Resulting* module after performing the operation .

After invoking this operation the **Operations on Modules** dialog box appears which includes two radio buttons for specifying the type of the objects you want to transfer and lists for selecting the initial modules (from which the modules are taken) and resulting ones (where they should be transferred) (Fig. 9.3-8).

The following procedure is used to transfer the selected elements into the resulting module:

- select the radio button of the respective option in the **Operations on Modules** dialog box;
- select the module the elements are transferred into in the **Select the resulting module** list;
- click the **OK** button (the dialog box closes);
- select the elements you want to transfer in the model;
- click the **OK** button in the **Model** section.

If the option of transferring all elements from one module to the other is selected, you have to select the initial module in the left list, and the resulting module – in the right list, and click the **OK** button. The dialog box will close, and the elements will be transferred.

Control the Blocks

The operation is similar to the control og groups. It is used to control the attributes of blocks and enables to perform the following actions: rename blocks, sort blocks by a certain criterion, delete blocks and change the place of the selected block in the list of blocks.

9.4. Creating Standard Structures

The software enables to generate the design models of standard structures (substructures) commonly used in design practice. To do this, select the type of structure from the drop-down list in the dialog box (see Fig. 9.4-1), specify the geometric parameters and stiffness properties.

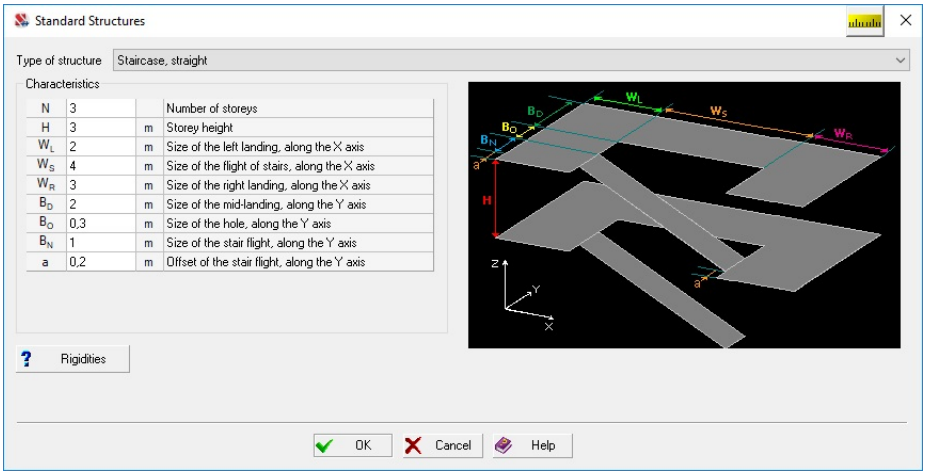
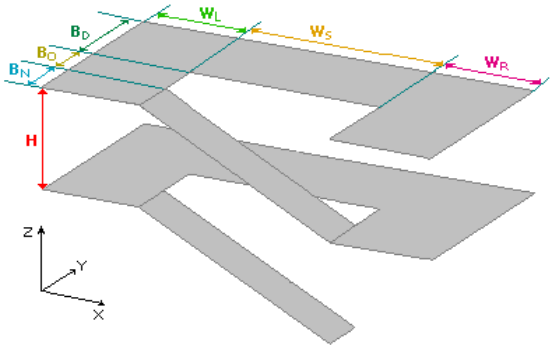
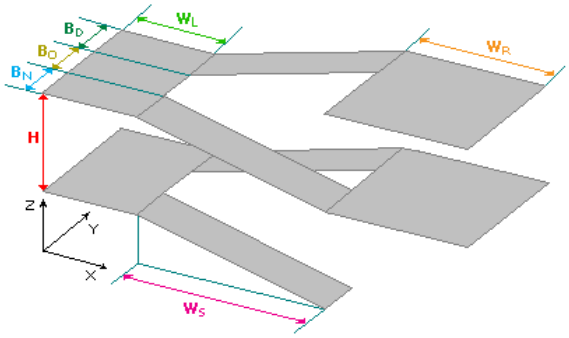


Figure 9.4-1. *The* **Standard Structures** *dialog box*

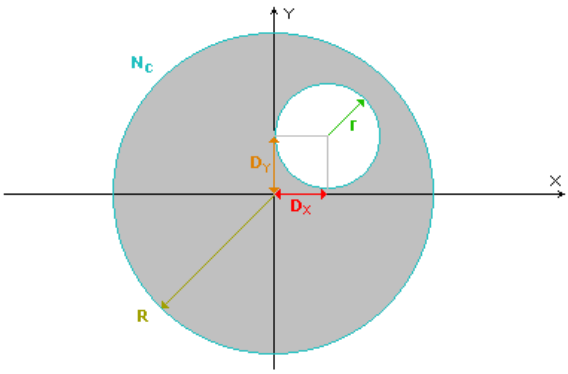
The following types of structures are implemented in the program:



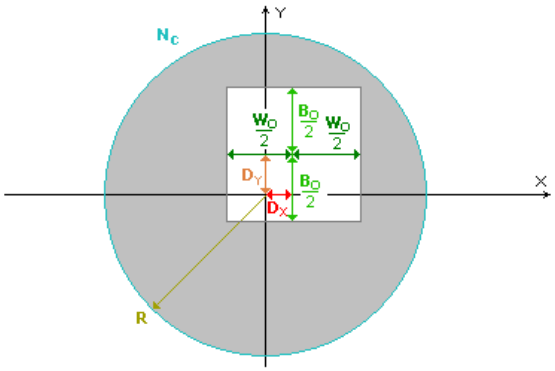
Staircase, straight



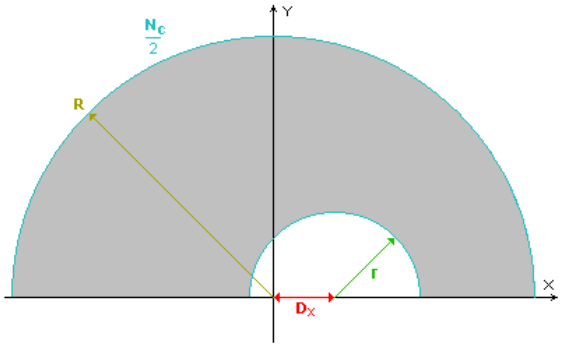
Staircase, dogleg



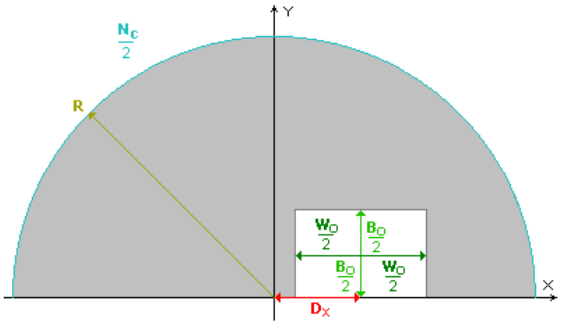
Round slab with a round hole



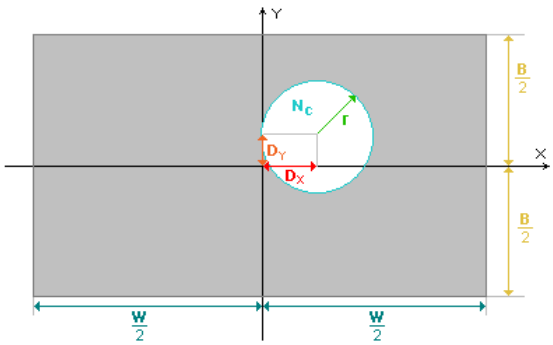
Round slab with a rectangular hole



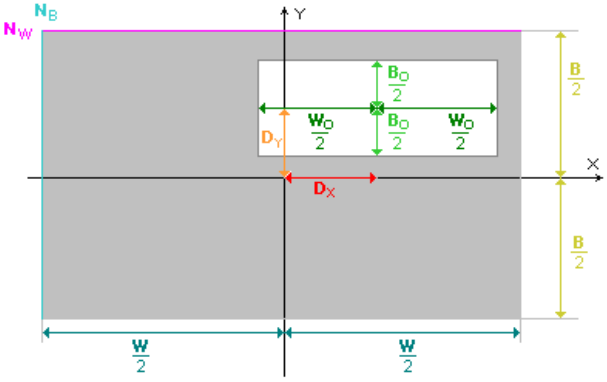
Semicircular slab with a semicircular notch



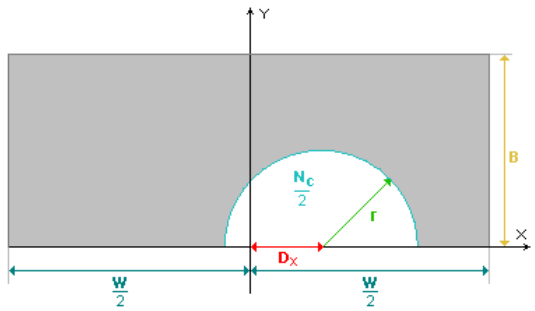
Semicircular slab with a rectangular notch



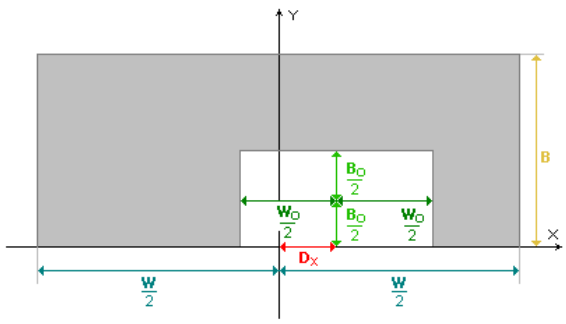
Rectangular slab with a round hole



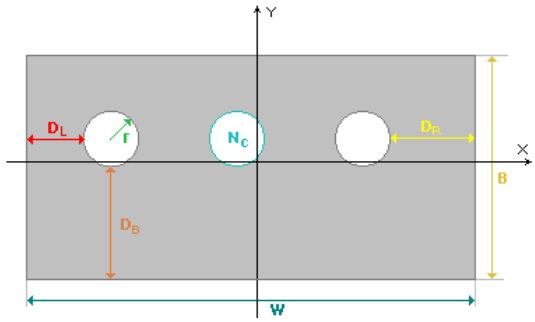
Rectangular slab with a rectangular hole



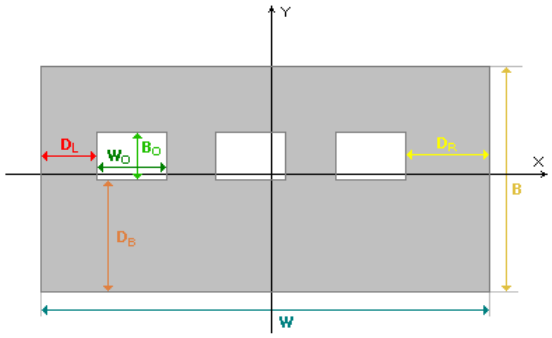
Rectangular slab with a semicircular notch



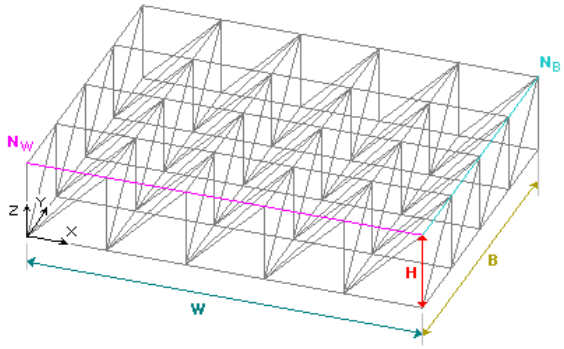
Rectangular slab with a rectangular notch



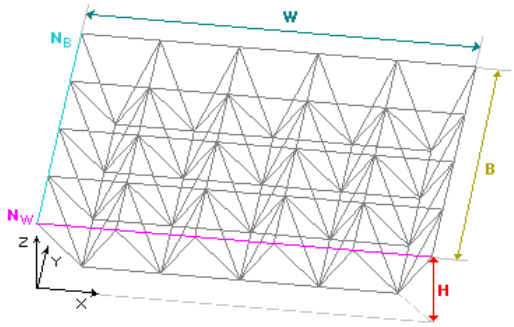
Rectangular slab with several round holes



Rectangular slab with several rectangular holes



Truss-roof



Truss-roof shifted by half step

9.5. Operations with Nodes

All operations with nodes of models of the **FORUM** type are invoked from the toolbar (Fig. 9.5-1).

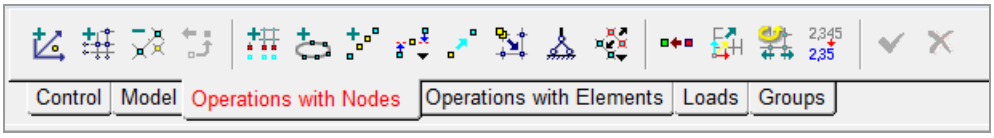


Figure 9.5-1. *The* **Operations with Nodes** *toolbar*

These operations are similar to the respective operations with finite element models described in Section 6.2.

9.6. Operations with Elements

Functions of the **Operations with Elements** section of the toolbar are described below. Most operations are similar to the respective operations with finite element models (see Section 6.3), therefore only the operations specific to the integrated models are described here.

Change the Stiffness Properties

These operations enable to change the stiffness properties of the selected bar or plate elements. They are similar to the respective operations for finite element models (see Section 7.1). The only difference is that the concept of *grade* is used here instead of the type of rigidity.

Connect Walls and Floors

As a rule, the problem of inconsistencies between walls or floors arises after importing the model from architectural systems. The considered operation is used to eliminate inaccuracies of the model by connecting such objects. When it is invoked a dialog box appears (Fig. 9.6-1). Its first tab enables to select a set of actions to adjust the model by checking the respective checkboxes.

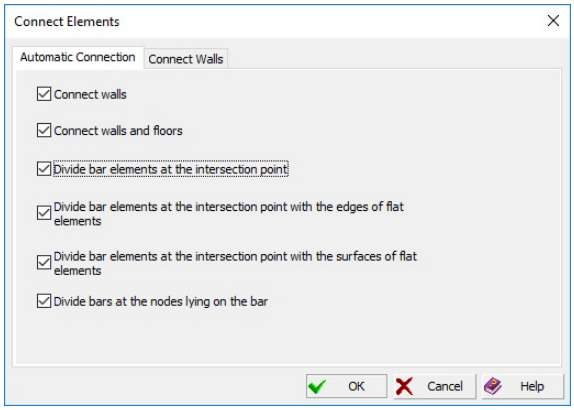


Figure 9.6-1. *The* **Automatic Connection** *tab*

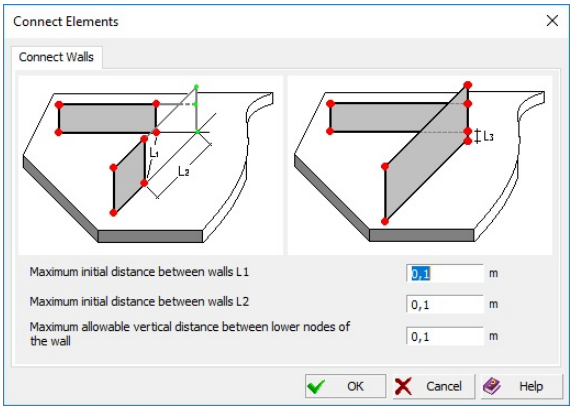


Figure 9.6-2. *The* **Connect Walls** *tab*

The following set of actions is provided here:

- connect walls;
- connect walls and floors;
- divide bar elements at the intersection point with the edges of flat elements;
- divide bar elements at the intersection point with the surfaces of flat elements;
- connect the selected elements;
- divide bars at the nodes lying on the bar.

If the **Connect walls** checkbox is checked, the dialog box will have two tabs. The second tab (Fig. 9.6-2) enables to specify the maximum distances (between walls and between lower nodes along the vertical), at which the software should consider two walls as such that have to be connected.

The procedure for performing this operation is as follows: invoke the operation, specify the parameters, select the elements, confirm (by clicking the **OK** button).

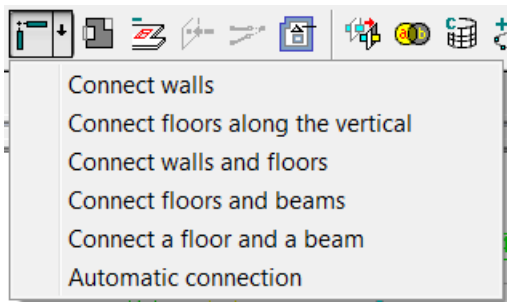


Figure 9.6-3. *Menu of operations*

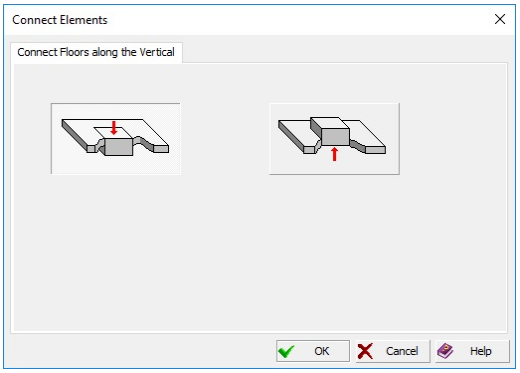


Figure 9.6-4. *The* **Connect Elements** *dialog box*

Clicking the arrow drops down the menu (Fig. 9.6-3), in which you can select only one of the above operations. Moreover, the menu contains the **Connect floors along the vertical** item, which enables to invoke the dialog box (Fig. 9.6-4), where you can select whether the created floors should be aligned along the upper or lower faces.

It should be noted that when the walls and floors are connected, the floor does not change its position – the operation is performed by extending the wall.

Correcting Openings

The problem of incorrect placement of openings with respect to the wall or floor usually arises when importing data from architectural systems. Sometimes this can be done on purpose in **FORUM** to model doorways by moving the window to the contour of the wall (by moving nodes). The considered operation is used to eliminate this kind of inaccuracies of the model. The intersecting openings (Fig. 9.6-5, *a*) will be combined into one opening in the result, and the contour of the opening crossing the contour of the element will form a new contour (Fig. 9.6-5, *b*). In order to perform the operation invoke it, select the elements with an incorrect arrangement of openings and click the **OK** button in the toolbar.

a) *b)*

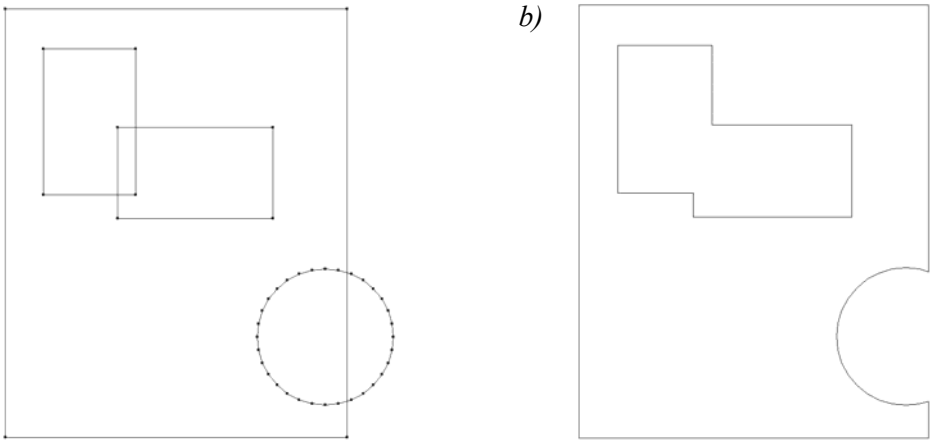


Figure 9.6-5. *Openings before (a) and after (b) performing the operation of correcting*

Delete Openings

This operation enables to delete openings in the elements of walls, floors and roofs.

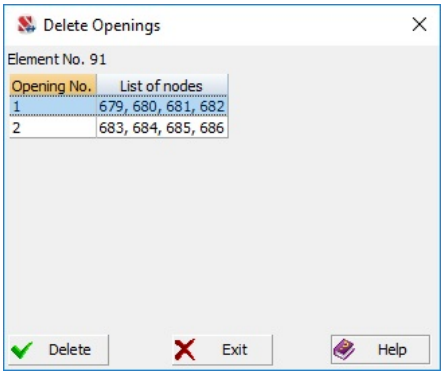


Figure 9.6-6. *The* **Delete Openings** *dialog box*

The following procedure is used to perform it:

- invoke the operation and the **Delete Openings** dialog box will appear (Fig. 9.6-6);

- select an element in the model or in the project tree which contains an opening or openings you want to delete;
- the **Delete Openings** dialog box displays the list of openings of the selected element indicating the numbers of nodes of each opening;
- select the opening you want to delete in the list (the opening will be highlighted in the model) and click the **Delete** button in the dialog box;
- repeat the previous action for other openings of this element you want to delete;
- after this operation without closing the dialog box select the next element in the model or in the project tree which contains openings you want to delete and repeat the above actions;
- close the dialog box after deleting the openings.

If when selecting one element, there are several elements in the cursor target at once, you can select it in the **Elements in the Cursor Target** dialog box.

If one or several elements were selected before invoking this operation, the element with the minimum number will be selected automatically.

Extend the Bar to the Plate

This operation enables to extend a bar (column of beam) to the selected planar object (floor or wall). In order to perform the operation invoke it, select the objects you want to apply this operation to and click the **OK** button in the toolbar.

Expansion of the Plate by an Offset from the Faces

The operation is used to change the dimensions of the plate by constructing an equidistant line, i.e. a line equidistant from all faces of the plate by the specified distance. Information necessary for performing the operation is specified in the respective dialog box (Fig. 9.6-7) and includes the value of the offset (distance from the faces). The **Change the external contour** and **Change contours of holes** checkboxes define the rules for performing the operation and enable to construct an equidistant for only one type of contour or specify different values of the offset for the outer contour and holes.

If the contour is non-convex and the value of offset is such that after changing the dimensions the faces intersect, an envelope is constructed without taking such segments into account (i.e. the contour on such a segment will become convex).

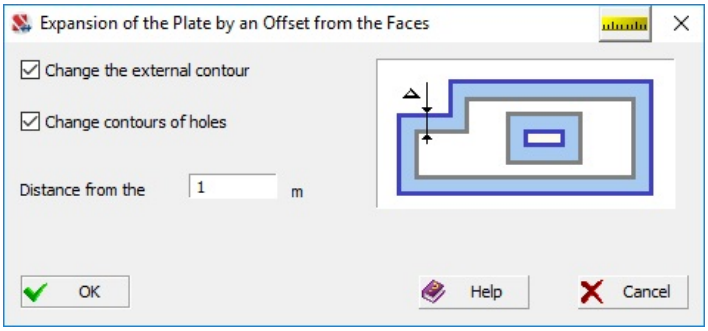


Figure 9.6-7. *The* **Expansion of the Plate by an Offset from the Faces** *dialog box*

Divide Plates along the Intersection Line

The mutual intersection of planar objects is taken into account in the process of an automatic generation of a finite element mesh at the transition from the **FORUM** preprocessor to the **SCAD** preprocessor. The mesh is generated “on both sides” of the intersection line (Fig. 9.6-8).

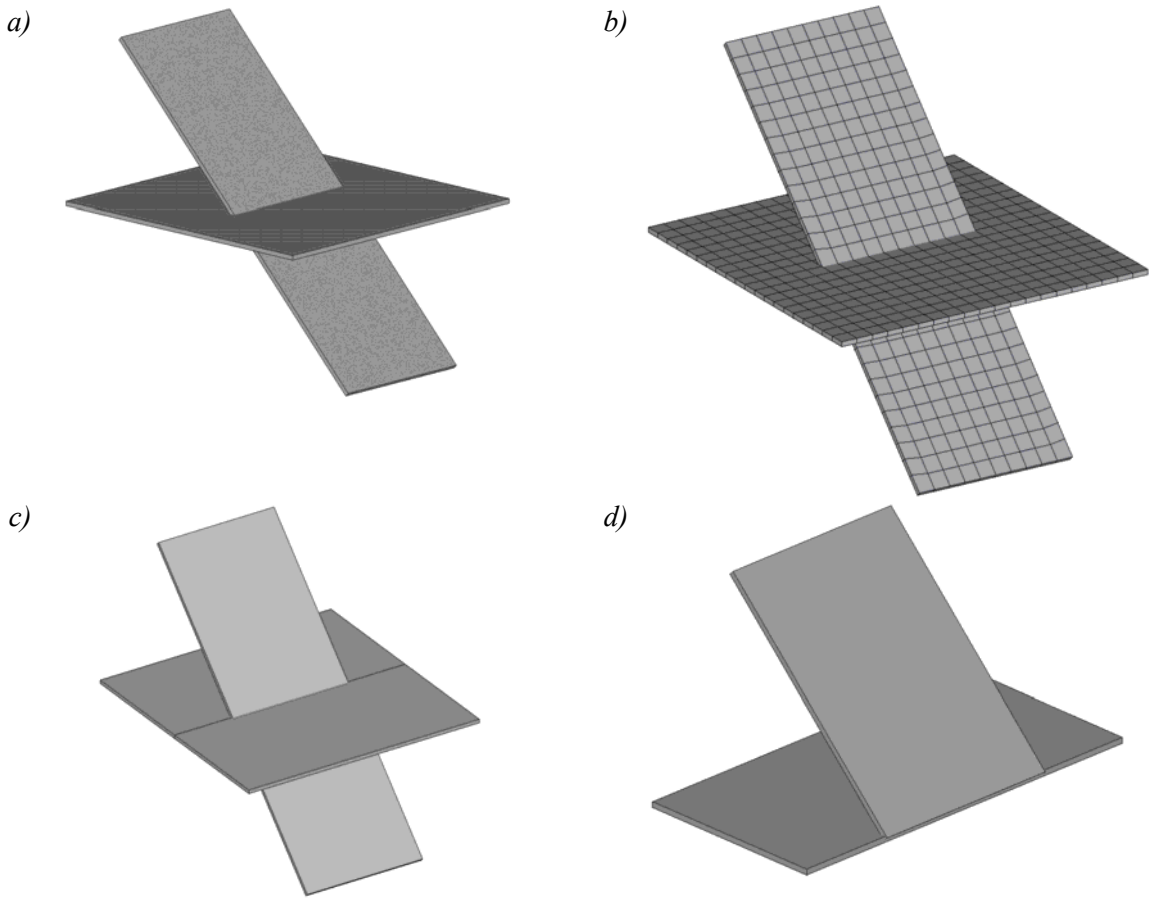


Figure 9.6-8. *a) Intersection of two planar objects in* **FORUM**
b) the result of an automatic mesh generation in **SCAD** *preprocessor*
c) the same objects after performing the **Divide plates along the intersection line** *operation*
d) after deleting “redundant” objects

In those cases when a part of the element lying on one side of the intersection line should not get into the design model, you can use the **Divide plates along the intersection line** operation, and each planar element will be divided into several parts along the intersection lines with other planar elements. In this case "redundant" elements can be deleted with the help of the **FORUM** preprocessor.

In order to perform the operation you have to invoke it, select the intersecting objects in the model and click the **OK** button in the toolbar.

Before performing the generation of a finite element mesh the check of planar objects is performed. In those cases when non-planar elements are detected in the model, a message is generated. If you continue the generation (ignore the message), the finite element mesh in the area of intersection of the non-planar elements might be generated incorrectly.

Operations with a Pair of Planar Objects

If two planar objects lying in the same plane have a non-empty intersection, this operation enables to perform their union, intersection, ... The type of operation is selected in the dialog box (Fig. 9.6-9), which appears after clicking the button .

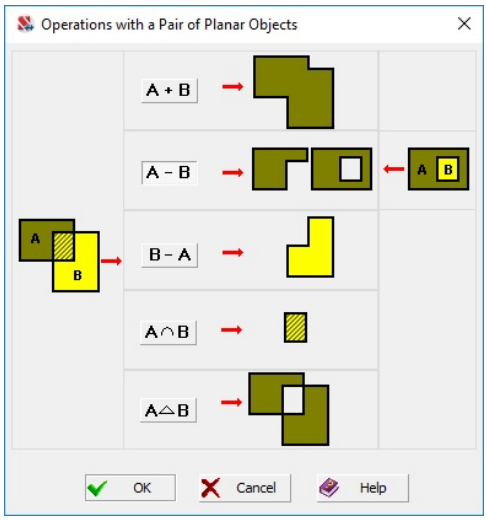


Figure 9.6-9. *The* **Operations with a Pair of Planar Objects** *dialog box*

You have to select the necessary operation in this dialog box, select two planar objects lying in the same plane in the model, and click the **OK** button in the toolbar. An example of using the operation of union is shown in Figure 9.6-10.

a) *b)*

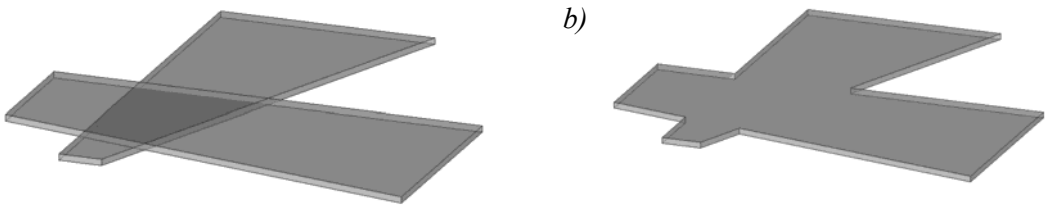


Figure 9.6-10. *Intersecting planar elements before a) and after b) performing the operation of union*

Create Surfaces of Revolution

This operation is similar to that described in Section 5.6. The only difference is that only plate models can be generated here, and the user has to specify the type (floor, roof or wall) of the newly created elements.

Division of floors and walls by beams and columns lying in their plane

This operation is used in those cases when the floor slabs and/or walls have to be divided into several parts using the lines passing through beams and columns lying in their plane as a separation line.

The most typical example of using this operation is the preparation for the transfer of information about the geometry of the model from the FORUM preprocessor to MONOLIT.

In MONOLIT the general floor layout is arranged on the orthogonal grid of nodes located at the intersections of structural members – beams, walls and columns. Floors are supported on the load-bearing walls and columns. The load on the columns and walls is transferred through a system of secondary and main beams, and also directly to the walls, if the slab is supported on them.

The considered operation is used in those cases when the model generated in FORUM has to be modified taking into account the rules provided in MONOLIT, i.e. divide the specified floor section along the beam lines.

In order to perform this operation you have to:

- invoke it by clicking the button in the **Operations with Elements** toolbar;
- select only the elements (beams, columns, floors, and walls) involved in this operations;
- click the **OK** button in the toolbar.

9.7. Project Tree

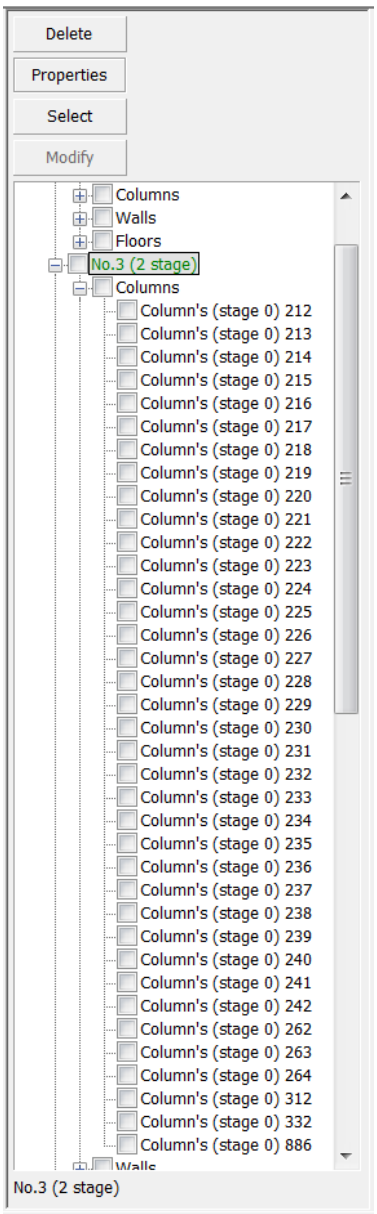


Figure 9.7-1. *Project tree*

The project tree is created automatically in the process of the generation of the model. The tree includes a panel with a set of control buttons, the actual project tree and the status bar with the name of the active module.

The tree has a multi-level structure and includes the following levels of representation of the model: model, module, group of elements, element. There are five types of groups of elements: columns, walls, beams, floors, roofs. The control of the tree is performed with the help of two types of marker-buttons of type ***a*** and ***b***.

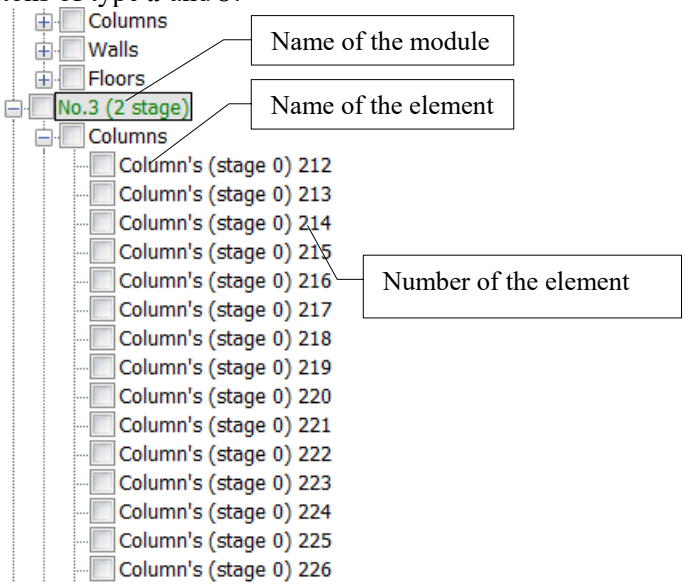


Figure 9.7-2. *Fragment of the project tree*

Markers of type ***a*** are used to unfold ⊟ or fold ⊞ a tree level beside which they stand. An unfolded group of elements **Floors** is shown in the figure. If there is no marker of type ***a***, it means, that there are no elements of the respective type in the module.

Markers of type ***b*** are used to disable (☐) and enable (☑) the display of objects on the screen. You can disable the whole model (for example, in order to make only one module visible), one or several modules, groups of elements and individual elements.

An *active module* is an important concept when working with the tree. New elements are automatically included only in the active module. The name of the active module is recorded in the status bar under the project tree, and it is marked by a gray rectangle in the tree. In order to activate the module you have to click on its name in the tree. Besides the active module a group of elements and an individual element (it is highlighted in color in the model) can be *active*.

Control Panel

Controls take up the upper part of the panel of the project tree (Fig. 9.7-3). They include the following buttons: **Delete**, **Properties**, **Select** and **Modify**.

Figure 9.7-3. *Controls on the panel of the project tree*

The **Delete** button enables to delete the active objects (selected in the tree). This applies to the whole model, and to modules, groups, and individual elements. The **Properties** button is used to access and adjust the parameters of the active element. Clicking the **Select** button selects the active object in the model. The **Modify** button is used to modify the geometry (for example, to specify the openings) of such elements as walls, floors and roofs.

View and Adjust the Parameters of Elements

Parameters of the objects selected in the tree can be viewed or adjusted using the **Properties** button.

If the model (the upper level) is an active object in the tree, then the **General Information on the Project** dialog box (Fig. 9.7-4) appears after clicking this button. It provides data on the numerical characteristics of the model. If the module is active, the **Parameters of the Module** dialog box appears (Fig. 9.7-5), which contains its name and color.

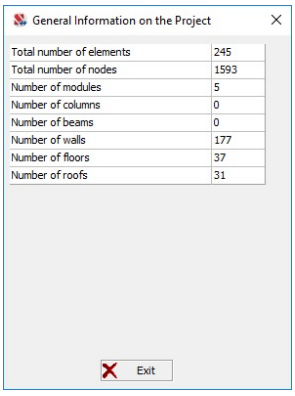


Figure 9.7-4. *The* **General Information on the Project** *dialog box*

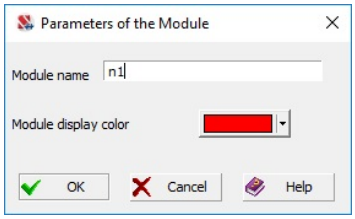


Figure 9.7-5. *The* **Parameters of the Module** *dialog box*

The color can be changed in the standard **Color** dialog box which appears after clicking on the color field. If an element is selected in the tree, for example, "floor", the **Characteristics of the Floor/Roof (Slab)** dialog box appears (Fig. 9.7-6), which provides information on the area of the section and length, or on the area of the element and thickness depending on the type of the element.

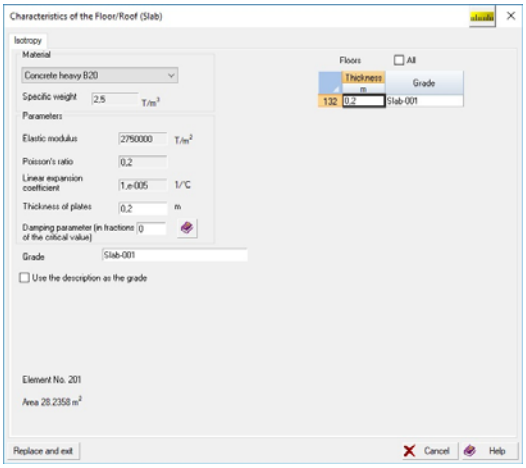


Figure 9.7-6. *The* **Characteristics of the Floor/Roof (Slab)** *dialog box*

Adjusting the Geometry of Elements

If a two-dimensional element (wall, floor, roof) is active, you can use the **Modify** button to adjust the geometry of the elements of such type. In this case adjusting means changing the borders (contour) of the element, creating window and door openings, holes etc. A special program, which is actually a modification of **Consul**, is invoked to perform the above operations.

Consul and **FORUM** operate with different objects; the concept of *contour* is used to describe them. It is a bar section in the first program, and a structural member — in the second one.

A contour can be external, i.e. a boundary of the element. Internal contour is a boundary of a hole or an opening. Vertices of the contour are nodes in terms of the model. The window of the adjusting program (Fig. 9.7-7) includes a menu, a toolbar, a work area (with scrollbars when necessary) and a status bar.

All actions are performed in the work area with a cursor. When moving the cursor over the screen or when performing some commands, the shape of the cursor changes. For example, when selecting an item from the menu or the toolbar the cursor takes the form of an arrow, when processing a command the cursor turns into an hourglass (busy cursor). If the cursor is placed over the section contour, it is displayed as a cross with its center coordinates defining its current position.

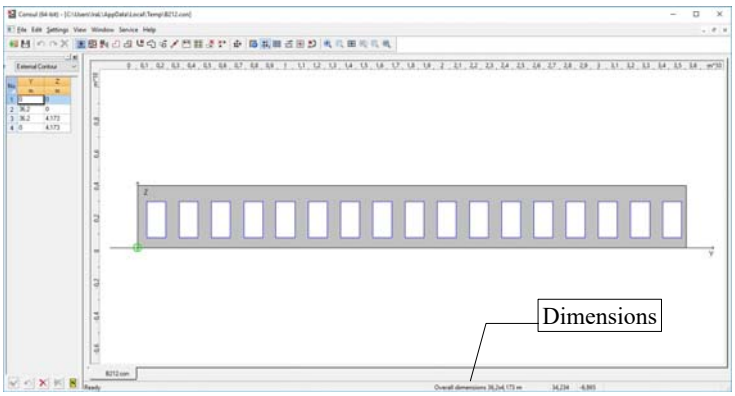


Figure 9.7-7. *Window of the program for adjusting the geometry of the elements*

When placed over the node the cursor takes the form of a cross with a target.

standard cursor — for selecting a function, a mode or an operation;

cross with a central target — for selecting one node;

cross with a rectangle — for selecting a group of nodes or elements by a rectangular or polygonal marquee;

cursor is used to move nodes, openings and holes, and to copy and duplicate them.

Coordinates of the current position of the cursor are output in the second field of the status bar.

Dimensions of the Element

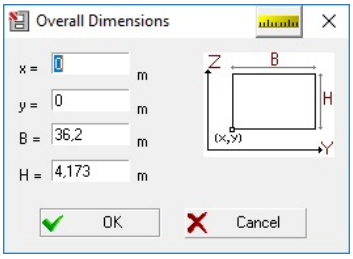


Figure 9.7-8. *The* **Overall Dimensions** *dialog box*

This operation enables to change the dimensions of the work area the element is displayed in, and it is used in those cases when the adjustment deals for example with increasing the dimensions of the element. Dimensions are specified in the **Overall Dimensions** dialog box (Fig. 9.7-8). After closing this dialog box a rectangle limiting the new dimensions of the element is displayed in the work area.

Coordinate Grid

You can use a coordinate grid to make the adjustment of elements more convenient. Parameters of a coordinate grid are specified in the **Grid Settings** dialog box (Fig. 9.7-9), which opens once you invoke the respective command. The text fields of this dialog enable you to specify horizontal and vertical grid spacing, and an angle of the grid in degrees with respect to the horizontal axis. The grid is rotated about the origin. This operation can be also invoked from the **Parameters** menu (**Grid spacing** item).

It should be noted that the grid spacing and its angle can be changed many times during the creation of the internal contours (openings and holes) or editing the external one.

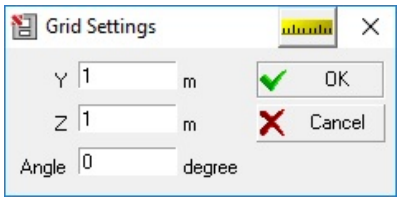


Figure 9.7-9. *The* **Grid Settings** *dialog box*

The program implements the mode of snapping the cursor to the nodes of the grid, which facilitates the adjustment of the geometry of the elements. Initialization of the snapping mode is performed by clicking the button in the toolbar. If this mode is enabled when you create and adjust the contours of elements and openings, the new nodes will be automatically snapped to the grid node nearest to the cursor.

The grid will be displayed once its parameters are entered (Fig. 9.7-9). Its visualization is turned on and off by the button , in the toolbar.

Edit the External Contour

Clicking the **Polygonal External Contour** button in the toolbar invokes the operation of editing the external contour of the element.

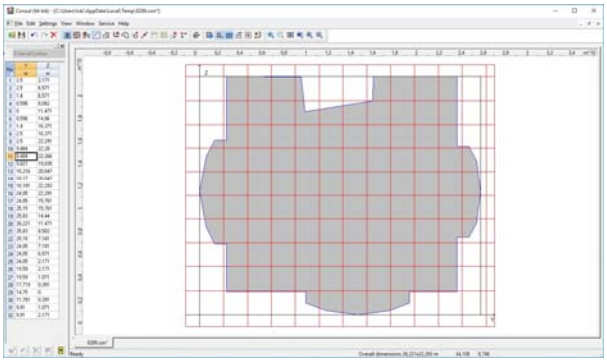


Figure 9.7-10. *Editing the external contour*

Place the cursor over any point of the contour to start the editing. After the cursor changes its shape (to a cross for an arbitrary point or to a cross with a target for a vertex), press the left mouse button and "drag" the selected point to a new position. The new vertex is fixed by double-clicking the left mouse button. An element with an edited external contour is shown in Fig. 9.7-10. Particular attention should be paid to door openings which are basically "recesses" in the external contour, and not holes.

When moving the vertices the intersection of the sides of the external contour and the intersection of sides of the internal contour with those of the external one are not allowed.

Creating Openings and Holes

The program provides three types of commands for creating the internal contours:

- creating a contour in the form of a closed polygon — button in the toolbar;
- creating a contour in the form of a circle with dynamically assigned sizes (***Edit*** menu);
- creating a contour in the form of a circle with a given radius (***Edit*** menu).

The sequence of operations for creating and editing a contour in the form of a closed polygon does not differ from that for an external contour.

Figure 9.7-11. *The* **Radius** *dialog box*

When creating an internal contour in the form of a circle with a dynamically assigned radius, place the cursor over a point of the section corresponding to the center of the circle, click and hold the left mouse button, and drag the cursor until you reach the required dimensions of the circle. Double click the left mouse button or click the left mouse button and press the spacebar on the keyboard to fix the contour of the hole. If you want to interrupt this operation, click the right mouse button.

If the operation with specifying the radius of a circle is selected, it is performed in the respective dialog box (Fig. 9.7-11). After closing the dialog box (the **OK** button) in order to create a hole you just have to click on the snap point of the center of the circle and click the left mouse button.

The number of points (nodes) on the circle is specified with the help of the operation which is initialized by clicking the button in the toolbar. The number of points on the arc is proportional to the length of the arc.

When creating polygonal internal contours, their intersection with the external one and with each other is not allowed.

When specifying the number of points on a circle, remember that if the specified number of points on the arc is too big, it can lead to the appearance of degenerated finite elements and as a result to the loss of the accuracy of the solution.

Parametric Holes

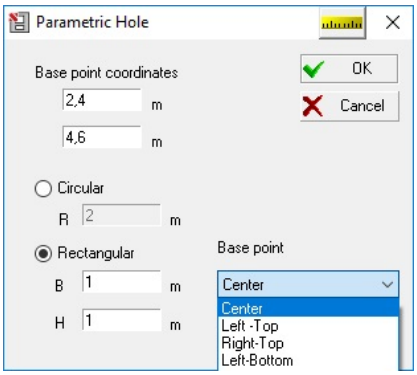


Figure 9.7-12. *The* **Parametric Hole** *dialog box*

This command is invoked from the **Edit** menu (Fig. 9.7-12) and enables you to create a rectangular or circular hole with specified dimensions and base point. Data for performing the operation are specified in the **Parametric Hole** dialog box. The type of hole (*Circular* or *Rectangular*) can be selected with the help of the radio buttons. Position of the point by which a rectangular hole will be snapped to the point with the coordinates specified in the dialog box (**Y**, **Z**) is selected from the **Base point** drop-down list. The hole will appear after closing the dialog box by clicking the **OK** button.

Delete Holes and Openings

To delete a hole or an opening, invoke this command, place the cursor over any point inside the internal contour you want to delete and click the left mouse button.

Round an Angle

An angle can be rounded by inscribing a circular arc of a given radius in it. After you invoke this command, place the cursor over a vertex of the contour (internal or external) and when the cursor takes the form of a cross with a target, click the left mouse button. In the invoked **Radius** dialog box (Fig. 9.7-13), specify a radius and click the **OK** button.

An element with rounded angles is shown in Fig. 9.7-14. The number of points (nodes) on the full circle is specified with the help of the **Number of points on the circle** operation which is initialized by clicking the button in the toolbar. The number of points on the arc will be proportional to the length of the arc.

Figure 9.7-13. *The* **Radius** *dialog box*

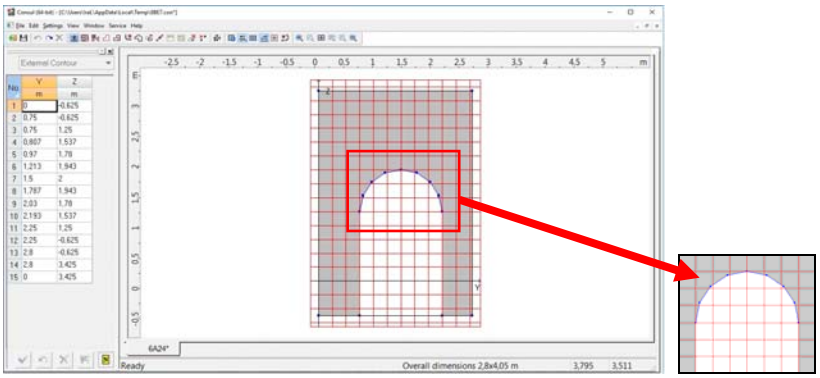


Figure 9.7-14. *A door opening with rounded corners*

Create an Arc on the Contour

This command enables to construct an arc on the internal or external contour. It is constructed through three points, two of which (the beginning and end of the arc) must lie on the contour. The contour can "bend" both inwards into the body of the element, and outwards. The first and the last point may lie on the adjacent sides of the contour. Number of nodes on the arc depends on the number of points on the full circle specified with the help of the **Number of points on the circle** operation and will be proportional to the length of the arc.

a) *b*) *c*)

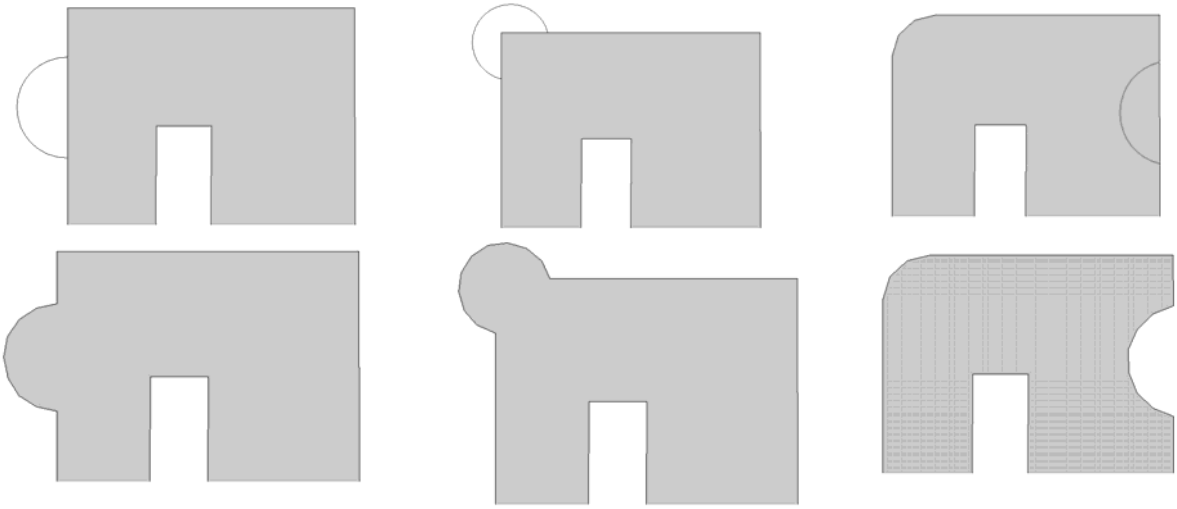


Figure 9.7-15. *Create an arc on the contour:*
a) on a straight section of the contour; b) on the adjacent sides of the contour; c) "into the body" of the contour

An arc cannot go beyond the boundary of the element. If this happens, then before constructing the arc you need to change the overall dimensions using the respective command, and then with the help of the **Move Vertices** item move the element within the new limits so that the arc does not intersect the boundary of the element. Perform the following steps to construct an arc:

- place the cursor over the beginning point of the arc and left-click;
- place the cursor over the ending point of the arc and left-click;
- place the cursor over the third point (the arc will be displayed on the screen) and left-click.

Examples of using the **Arc** command are shown in the Fig. 9.7-15. An initial arc and the final result are shown for each case.

Case *c*) shows the results of changing the contour "into the body" of the element, case *b*) — on the adjacent sides, case *a*) — on the straight section of the contour. It should be noted that in case *c*) we did not have to change the dimensions of the element.

Move Nodes

This command enables to move a group of nodes selected by a rectangular or polygonal marquee. Follow these steps to perform this operation:

- invoke the command;
- select a marquee in the drop-down menu;
- select the nodes you want to move with a marquee;
- move the cursor inside the marquee and when it changes its shape, move the marquee together with the selected nodes to a new position.

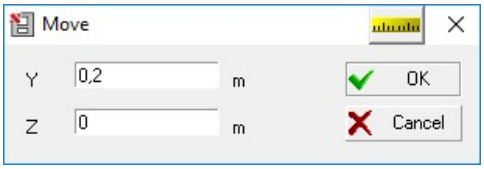


Figure 9.7-16. *The* **Move** *dialog box*

The new position of the nodes is fixed by clicking the left mouse button.

If you need to move the nodes by a specified distance, then after you move the cursor inside the marquee, click the right mouse button and specify the displacement value in the **Move** dialog box (Fig. 9.7-16).

When moving the nodes the intersection of sides of the external contour of the element, and the intersection of polygonal openings with the external contour are not allowed.

Copy Holes and Openings

This command enables to copy one or more openings (holes) selected by a rectangular or polygonal marquee. To do this, follow these steps:

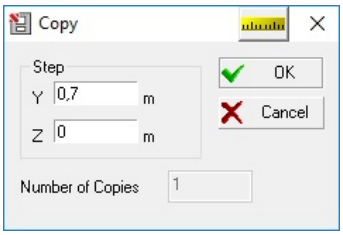


Figure 9.7-17. *The* **Copy** *dialog box*

- invoke this command;
- select the type of a marquee in the drop-down menu;
- select the openings you want to copy by this marquee (all nodes of the openings must be selected by the marquee);
- move the cursor into the marquee, and after the cursor changes its shape, click the left mouse button, and move the marquee together with the selected openings to a new position;
- confirm the new position by clicking the left mouse button.

If the copy of the hole has to be put into a certain place inside the element, then click the right mouse button after moving the cursor into the marquee, the **Copy** dialog box (Fig. 9.7-17) will appear where you have to specify the value of the ***displacement***. The openings will be copied after closing the dialog box and clicking the **OK** button.

When moving the openings their intersection with each other and the intersection of sides of the openings with those of the external contour are not allowed.

Duplicate Holes and Openings

This command enables to duplicate one or several openings (holes) selected by the marquee several times. To do this, follow these steps:

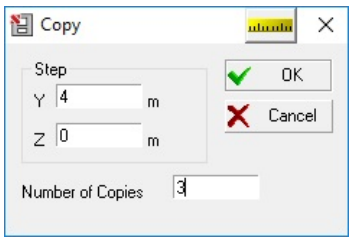


Figure 9.7-18. *The* **Copy** *dialog box*

- invoke this command;
- select the type of a marquee in the drop-down menu;
- select the openings you want to duplicate by this marquee (all nodes of the openings must be selected by the marquee);
- move the cursor into the marquee, and after the cursor changes its shape click the right mouse button;
- the **Copy** dialog box (Fig. 9.7-18) will appear where you have to specify the values of the displacement along the axes and the number of copies;
- click the **OK** button in the dialog box, it will close, and the operation of duplicating will be performed.

When copying the openings their intersection with each other and the intersection of sides of the openings with those of the external contour are not allowed.

Edit the Coordinates of the Nodes

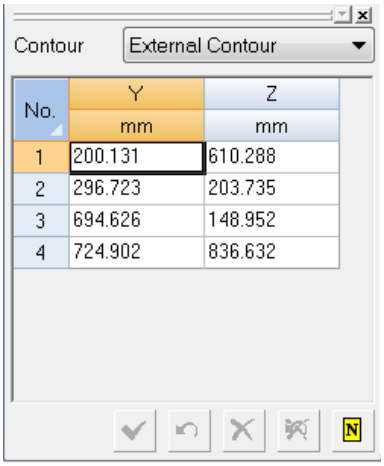


Figure 9.7-19. *Coordinates of vertices*

This command enables to edit the coordinates of the nodes in a tabular form. Click on the **Table of Vertices** button and this table (Fig. 9.7-19) will appear on the left from the work area. The dialog box includes a list of contours in the order of their creation and the table with the coordinates of vertices selected from the contour list. Perform the following steps to edit the position of nodes:

- select a contour from the list;
- change the coordinates of nodes in the table of coordinates (the nodes will be marked on the contour with color circles);
- or delete the selected node (the **Delete** and **Undo delete** buttons);
- confirm the obtained result (button).

The button cancels the last action on editing the contour performed before clicking the confirmation button .

When moving the nodes the intersection of the sides of the external contour of the element and the intersection of sides of the holes and openings with those of the external contour are not allowed.

Delete the Nodes

This command enables to delete nodes of both the external and internal contours of the element. To do this, follow these steps:

- invoke this command;
- select the type of cursor in the drop-down menu (for selecting one node or a marquee for deleting several nodes);
- click on the node you want to delete or select a group of nodes you want to delete with a marquee;
- click the left mouse button.

Show Grid

This button is used to enable/disable the display of a coordinate grid within the dimensions of the element.

Snap Cursor to the Grid

If this button is pressed, when creating and adjusting the contours of elements or openings the vertices (nodes) will be automatically snapped to the nearest node of the coordinate grid.

Nodes on the Circle

This operation enables to specify the number of nodes on the full circle which will be used when rounding angles and creating round holes. 21 points are specified by default.

When specifying the number of nodal points on the circle you should remember that by increasing their number you reduce the distance between them and degenerate elements can appear in the result in the process of generation of a finite element mesh and consequently the accuracy of solution can be lost.

Show Nodes

If this button is pressed, the nodes are displayed on the external and internal contours of the elements.

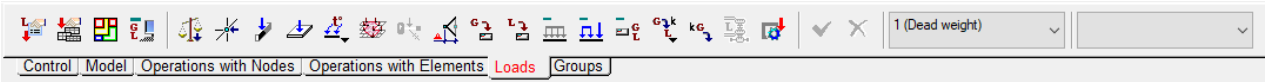
9.8. Display Filters

Filters (Fig. 9.1-1) are used to control the display of the model and perform the same functions as in the case of a finite element model.

9.9. Specifying Loads

The **FORUM** preprocessor enables to use two modes of specifying loads. The first mode is similar to specifying loads for a finite element model (see Chapter 8). Moreover, the **FORUM** preprocessor enables to specify temporary and permanent loads (including loads from the self-weight of structural members). This mode is described below and is based on the principle of snapping the load to floors (slabs) depending on the functional purpose of rooms to which they belong (except for the load from the self-weight, which is determined automatically for all the elements of the structure).

The **Loads** toolbar contains the following set of buttons for the control of the load specification process:



— parameters of the load specification mode;

— tables of loads;

— specify the types of rooms;

— delete loads.

Parameters of the Load Specification Mode

The operation is used to set the mode of formation of loadings in the process of generating a finite element model. The setting is performed in the **Characteristics of Loads** dialog box (Fig. 9.9-1).

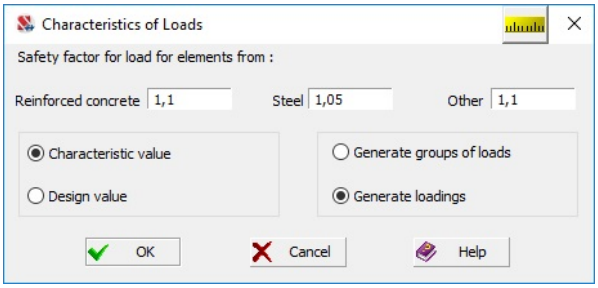


Figure 9.9-1. *The* **Characteristics of Loads** *dialog box*

The dialog box includes three text fields for specifying safety factors for load that are used when generating the design loads. The dialog box also contains two groups of radio buttons which enable to select values with which the loads will be considered in the finite element model (characteristic or design), and the representation of loads – in the form of groups of loads or loadings.

If the mode of generating groups of loads is selected, each type of load will be included in a separate group. The formation of loadings from groups is performed in the **SCAD** preprocessor.

Tables of Loads

The control of the process of formation of loads in the automatic mode of the load specification (only the automatic mode is available in this version) is performed based on the data recorded in the tables of the **Description of Loads** dialog box (Fig. 9.9-2, 9.9-3). The dialog box includes three tabs with tables, two of which contain the description of permanent and temporary loads (reduced to one square meter of floor) and the third one contains a table with the rules of distribution of permanent loads depending on the type of room (**Allowing for Permanent Loads** tab). The initial content of the tables is conventionally called "standard" and can be changed by the user.

A set of permanent loads (for example, the self-weight of floor slab, the weight of floors and the weight of partitions) and only one temporary load can correspond to each type of room.

Permanent Loads

Permanent loads can include both loads from the self-weight of the structural members and other types of load.

The list of permanent loads and their values are given in the table in the **Permanent Loads** tab of the **Description of Loads** dialog box (Fig. 9.9-2). The table contains characteristic and design values of loads, and the safety factor for load.

Load from the self-weight is formed automatically and changes with the parameters of the structural members of the model. In order to automatically take the self-weight of elements into account you have to check the respective checkbox in the considered tab.

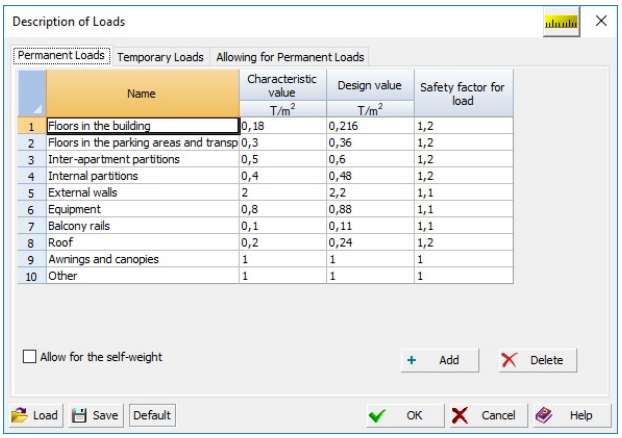


Figure 9.9-2. *The* **Description of Loads** *dialog box*, *the* **Permanent Loads** *tab*

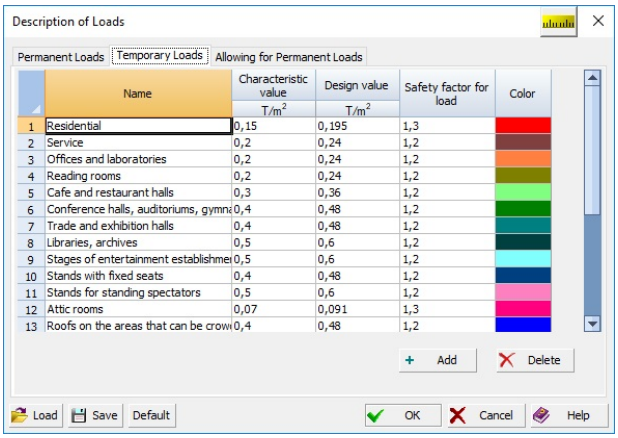


Figure 9.9-3. *The* **Description of Loads** *dialog box*, *the* **Temporary Loads** *tab*

The program enables to supplement or change the original list of loads. The buttons at the bottom of the dialog box are used to do this.

It is necessary to perform the following actions to specify a new type of load:

- click the **Add** button, and a new line will appear in the table;
- specify the name of the load in the **Name** column;
- specify the characteristic and design values of loads in the **Characteristic value** and **Design value** columns. The value of the safety factor for load will be specified automatically after switching to another tab or opening the **Description of Loads** dialog box for the second time (it is actually enough to fill any two columns — the third one will be filled automatically);
- in order to use a list of permanent loads including a newly specified type of load when working with other projects you have to click the **Save** button and specify the name of the file which can be loaded into another project with the help of the **Load** button in the standard dialog box.

  You can return the default state of the table when necessary using the **Default** button.

  Load values are given in units of force per unit area (for example, in t/m^2).

Temporary Loads

By default this table mainly corresponds to the table of SNiP “Loads and Actions”. Loads given in the table are applied only to the floor slabs. The snap of the load to the floor is specified automatically depending on the type of room specified for the considered floor in the **Characteristics of the Floor/Roof (Slab)** dialog box (Fig. 9.9-5). Thus, in order to specify the loads correctly, floors on different functional areas of the storey have to be specified by different floor elements. Assigning a type of the room to a part of the element is not provided in the software.

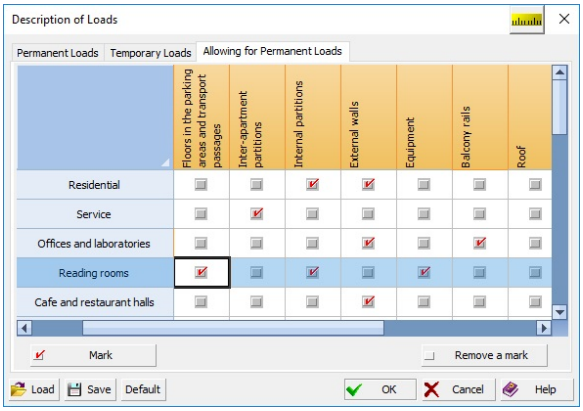


Figure 9.9-4. *The* **Description of Loads** *dialog box*, *the* **Allowing for Permanent Loads** *tab*

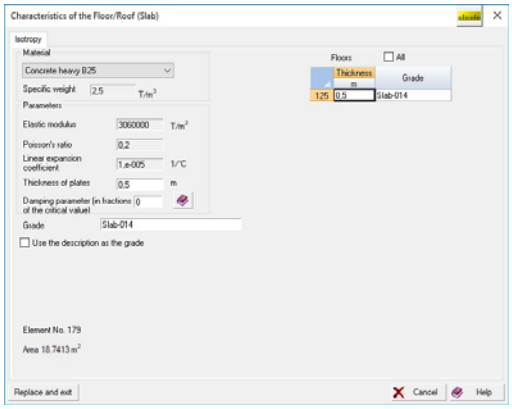


Figure 9.9-5. *The* **Characteristics of the Floor/Roof (Slab)** *dialog box*

Rules for working with the **Temporary Loads** table for changing the content of the table are similar to those described above for the **Permanent Loads** table.

Allowing for Permanent Loads

This table (Fig. 9.9-4) contains rules for snapping permanent loads to floors depending on the type of rooms. Types of rooms repeating the list in the table of temporary loads are specified in the first column of the table. The second column shows the color which will be used to display the rooms of this type in the model after activating the **Types of rooms** filter — . Other columns the names of which coincide with the list of permanent loads contain checkboxes indicating (when checked) the presence of this type of load in the room of a certain type.

If the respective checkboxes are checked for each type of room, then in the process of generating a design model the loads will be added or included into different groups the names of which will coincide with the name of the source of the load depending on whether the loadings or groups of loads are created.

Thus, the process of forming loads comes down to an automatic determination of loads from the self-weight for all elements of the model, and to the specification of a set of permanent loads and a temporary load depending on the type of room for each floor.

Peculiarities of Allowing for the Self-Weight of Enclosing Structures and Partitions

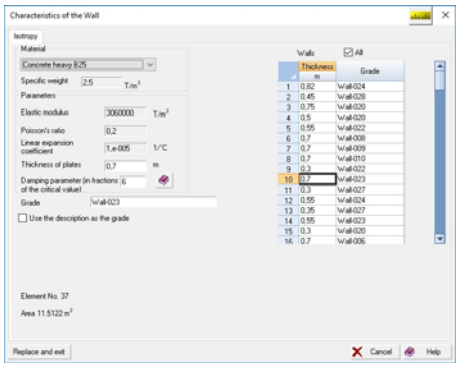


Figure 9.9-6. *The* **Characteristics of the Wall** *dialog box*

If the loads from the self-weight are automatically taken into account in the model, the loads from the enclosing structures and partitions of different type can be taken into account in two ways. Firstly, as a uniformly distributed load (in this case the respective checkboxes are checked in the table of permanent loads) and secondly, as a load "along the line". In the second case the partitions or enclosing structures have to be created in an integrated model as walls, and when describing their characteristics the "Partition" item has to be selected in the **Purpose of the wall** list in the **Characteristics of the Wall** dialog box (Fig. 9.9-6).

If the whole partition or it's part is over the opening, the load from it will not be transferred.

Specify the Types of Rooms

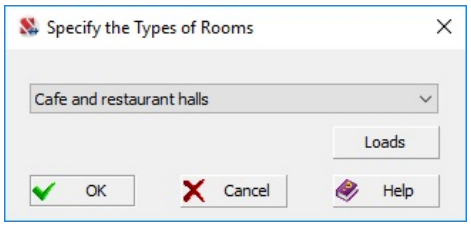


Figure 9.9-7. *The* **Specify the Types of Rooms** *dialog box*

Types of rooms can be specified both in the process of creating floor slabs, and in the finished model. In the latter case, you should use the respective operation. Once it is invoked the dialog box appears (Fig. 9.9-7), the drop-down list of which contains all types of rooms from the *Temporary Loads* table of the **Description of Loads** dialog box (Fig. 9.9-3). The **Loads** button enables to see the list (and parameters) of permanent loads corresponding to the selected type of rooms in accordance with the table of loads.

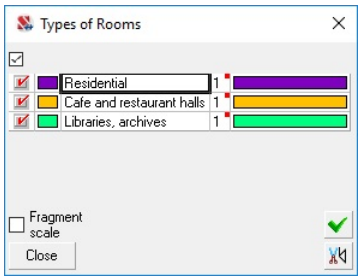
After selecting the necessary type of room in the list and closing the dialog box by clicking the **OK** button you have to select the floors in the model this type is assigned to, and click the button in the toolbar.

Delete Loads

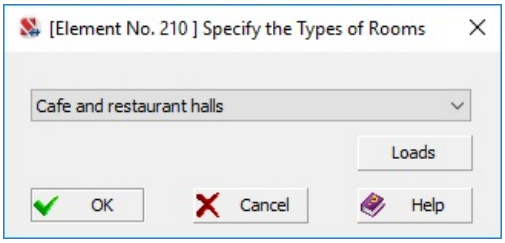
If you invoke this operation, select the floors in the model and click the button in the toolbar, then the selected elements will not have a specified type of room, and the loads will not be applied to them during the generation of the finite element model.

Control of the Permanent Loads

The **Types of rooms** filter enables to obtain a color scale of the types of rooms (Fig. 9.9-8,a) and, respectively, of the permanent loads on the floors. The control of this color scale is similar to the control of other scales, for example, of stiffness properties of elements. In order to obtain a summary table of loads applied to the floor of a room of a particular type, you have to double-click on the column of a color scale with the name of the rooms.



a)



b)

Figure 9.9-8. *The* **Types of rooms** *color scale*

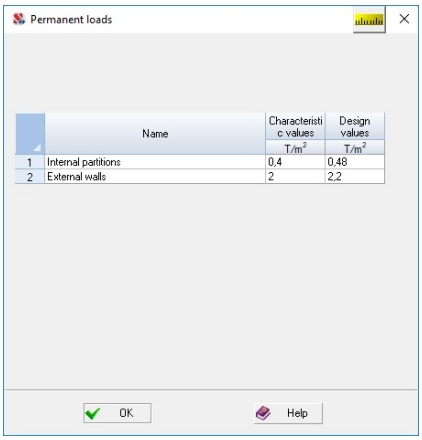


Figure 9.9-9. *The* **Permanent Loads** *dialog box*

The [**Element No.…**] **Specify the Types of Rooms** dialog box (Fig. 9.9-8, *b*) opens in the result. It enables to change the type of the room, and the **Loads** button of this dialog box enables to invoke the **Permanent Loads** dialog box (Fig. 9.9-9) with a table, which lists loads applied to the floor of this type, and contains their characteristic and design values (reduced to a unit area), and shows the total load per 1 m^2, with the exception of the self-weight.

9.10. Export the Model to SCAD

The button invoking the operation of generating the project in the input formats of the **SCAD** complex is in the **Model** section of the toolbar. The mode of exporting data into **SCAD** enables to save the information in the formats of the **FORUM** preprocessor (files with the extension **.opr**), and in the formats of the **SCAD** project (files with the extension **.spr**).

Planar objects of the following type **Floor**, **Wall**, **Roof** and bar elements — **Column** and **Beam** are transferred into **SCAD** and are represented in the complex by elements of type 1080, 1060, 1085, 1020 and 1040 respectively.

In order to make a design model from an integrated one, it is necessary to specify the types of bar elements for the respective bar objects, and perform meshing of the planar objects (the complex automatically recognizes the selected planar object as a meshing contour). All parameters of objects (properties of the material, shape and dimensions of the section etc.) are saved for all obtained finite elements. The figures show an integrated model in **FORUM** preprocessor (Fig. 9.10-1) and in **SCAD** complex (Fig. 9.10-2).

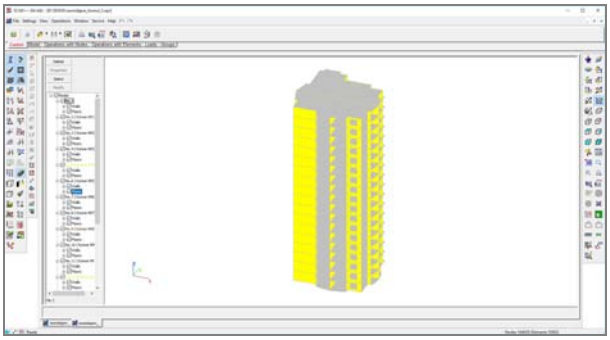


Figure 9.10-1. *Integrated model in* **FORUM** *preprocessor*

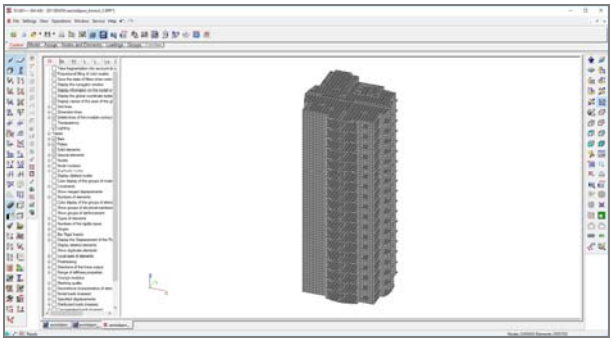


Figure 9.10-2. *Finite element model in* **SCAD** *complex*

Generation of a Finite Element Model

An automatic generation of a finite element model can be performed on the basis of an integrated model. In this case the control of the generation of a finite element mesh is performed in the **Generate the SCAD Project** dialog box (Fig. 9.10-3). Here, as in the previous case, the finite elements inherit all parameters of objects (properties of the material, shape and dimensions of the section etc.).

This dialog box appears after invoking the **Generate the resulting project** operation — in the **Model** section of the toolbar.

In order to align finite element meshes of different planar elements it is necessary to first of all align their borders, i.e. contours of the elements. A single contour division step is specified in the respective text field of the considered dialog box. This step is constant for external and internal contours of the elements, and for bar elements lying inside the contour of planar elements.

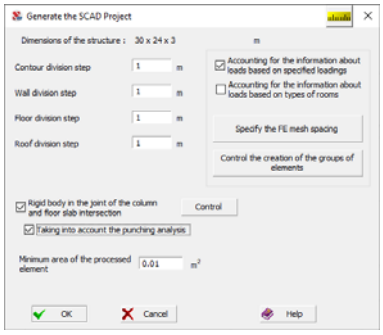


Figure 9.10-3. *The* **Generate the SCAD Project** *dialog box*

Regardless of the value of the contour division step, custom values of the mesh spacing can be specified for walls, floors and roofs. This enables, for example, to generate a relatively fine mesh on floors, where the reinforcement will be selected, and a coarse mesh on walls, which will be used in the model only to transfer wind loads.

In those cases when a detailed analysis and design has to be performed not for all modules of the design model, you can change the parameters of the mesh for certain types of elements for each module.

This is done in the **Clarification of the Parameters of Generation of the Resulting Project** dialog box (Fig. 9.10-4), which can be invoked by clicking the **Specify the FE mesh spacing** button. This dialog box contains a table with the names of modules and mesh spacing for dividing planar elements in each module. Since the contours of elements are divided with the same spacing, the alignment of the mesh is provided throughout the whole model.

Groups of elements are generated automatically together with the generation of the mesh in the design model. Finite elements that belong to one object and all elements belonging to a module are combined into groups by default. Rules for creating groups can be changed in the **Control the Creation of the Groups of Elements** dialog box, which can be invoked by clicking the respective button.

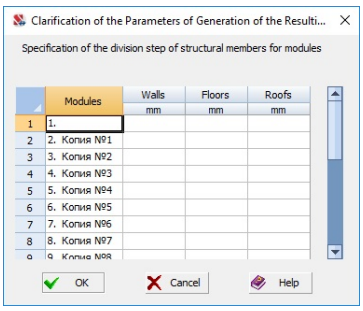


Figure 9.10-4. *The* **Clarification of the Parameters of Generation of the Resulting Project** *dialog box*

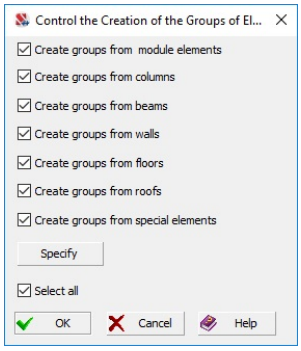


Figure 9.10-5. *The* **Control the Creation of the Groups of Elements** *dialog box*

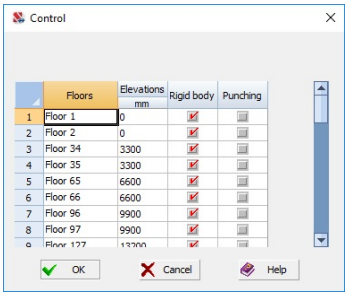


Figure 9.10-6. *The* **Control** *dialog box*

An option corresponds to each type of grouped elements in this dialog box (Fig. 9.10-5). Moreover, you can invoke the project tree by clicking the **Specify** button, where you can include or exclude them from the groups by checking or unchecking the respective checkboxes.

If the **Rigid body in the joint of the column and floor slab intersection** checkbox is checked in the **Generate the SCAD Project** dialog box, then in the process of generation of the finite element mesh this mesh will be generated taking into account the sizes of the column sections at the points of junction of columns to the floor slabs, and the finite elements within the column sections will be replaced by the rigid bodies. If the **Taking into account the punching analysis** checkbox is checked as well, both rigid bodies and finite elements in the column section will be generated in the joint (Fig. 9.10-9). Right-clicking on the columns *Rigid body* or *Punching* invokes a menu which enables to check or uncheck all checkboxes, or invert the selected checkboxes.

Clicking the **Control** button on the right from the considered options invokes the **Control** dialog box (Fig. 9.10-6), where you can check the respective checkboxes to indicate the floor slabs for which the mesh will be generated taking into account the sizes of the column sections and the punching analysis.

After closing the **Generate the SCAD Project** dialog box by clicking the **OK** button, a standard **Save As …** dialog box appears where you have to specify the name of the file of the resulting model.

The process of generating a finite element mesh is reflected by an indicator (Fig. 9.10-7) in the message window and can be interrupted by clicking the **Cancel** button.

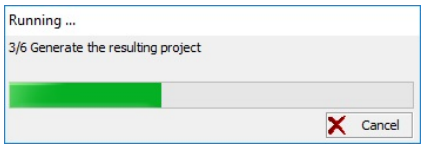


Figure 9.10-7. *The* **Generation process** *message window*

Every time you specify the spacing of a finite element mesh, you should keep in mind that if the spacing is too small, the dimension of the problem can exceed the capabilities of your computer of storing work files and files of the results of calculation. It is recommended to specify a small spacing only for those objects of the model for which it is necessary (it is usually done in those cases when it is expected to perform the selection of reinforcement).

Modeling the Joint of the Column and Floor Slab Intersection

If the **Refinement of the mesh under the columns** checkbox is checked in the **Generate the SCAD Project** dialog box (Fig. 9.10-3), then at the generation of a finite element model the mode of generating a mesh modeling the intersection area will be activated in the areas of the column and slab intersection. The

area is a rigid body the dimensions of which are equal to the overall dimensions of the rectangle corresponding to the column section. Depending on the position of the column with respect to the edge of the slab the area will have a different configuration. For example, Figure 9.10-8 shows the configuration of a rectangular column bearing areas on the plate in the middle of the plate - a) and at the edge of - b), c), d) these configurations and the corresponding rigid bodies. For example, Figure 9.10-8 shows the configuration of the bearing areas of the rectangular column on the slab in the middle of the slab – *a*) and on the edge – *b), c), d)* and rigid bodies corresponding to these configurations.

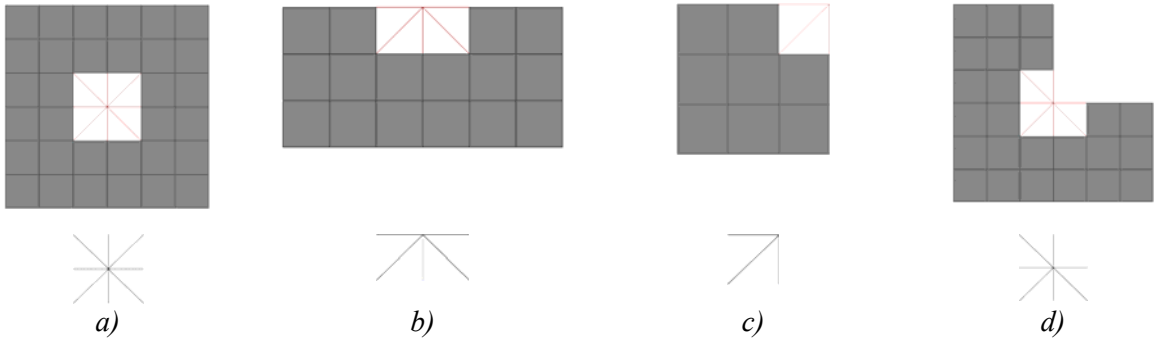


Figure 9.10-8. *Modeling the area of intersection of a column and a floor slab*

In some cases the program can not automatically generate the intersection area (for example, if the distance between the columns or between the face of the column and the edge of the slab is very small). The results of the automatic mesh generation should always be controlled and, if necessary, the adjustments should be made in the design model.

The result of the generation also depends on whether the **Taking into account the punching analysis** checkbox is checked (Fig. 9.10-9):

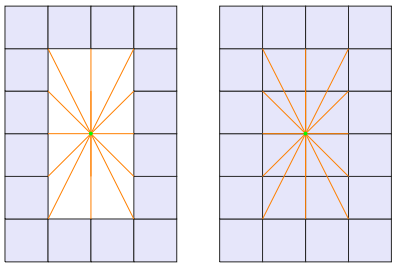


Figure 9.10-9. *Joints (without the punching analysis option and with it).*

Accounting for Loads

Parameters of a finite element model are specified in the **Generate the SCAD Project** dialog box, which can be invoked by the **Generate the resulting project** operation. Loadings or groups of loads will be generated only when one of the checkboxes **Loads based on the specified loadings** or **Loads based on room types** is checked. In the first case, no additional information is required. Consider the case of generating loads based on room types.

If the generation of groups of loads is provided in the settings of the project, a separate group will be created for each type of load, for example "Self-weight", "Floors in the building" etc. In the case of the generation of loadings the distribution of loads among different loadings is performed by the user.

The distribution is performed in the **Generate Loadings** dialog box (Fig. 9.10-10), which appears after clicking the **OK** button in the **Generate the SCAD Project** dialog box. It is recommended to use the following procedure in this dialog box:

- specify the name of the loading in the **Name of the loading** field;
- use the radio buttons to select the type of loads included in the current loading (permanent or temporary), and the list of the respective loads will be shown in the left table;
- select the load included in the current loading in the table and transfer it into the right table (the list of loads included in the current loading) by clicking the button with a single arrow;

- repeat the last action for other loads included in this loading (if all loads from the left table are included in the loading, you can use the button with two arrows);
- click the **Save as new** button to save the loading under the specified name.

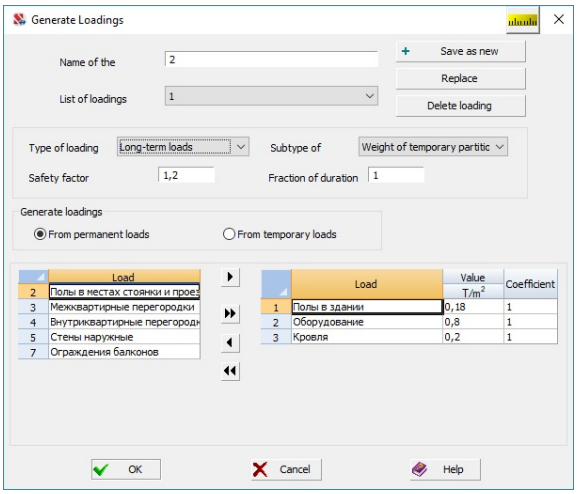


Figure 9.10-10. *The* **Generate Loadings** *dialog box*

The load value is specified in the second column of the right table, and the coefficient with which this load will be included in the loading is specified in the third one. The coefficient can be changed if necessary. You should keep in mind that the values given in the table correspond to the settings of the load generation mode, i.e. if the design values were specified in the settings, they will be in the table as well (in this case the multiplication by the safety factor for load is already performed). An exception is the load from the self-weight, the value of which is not given in the table, because it depends on the section and material of different elements.

Results of Generation Dialog Box

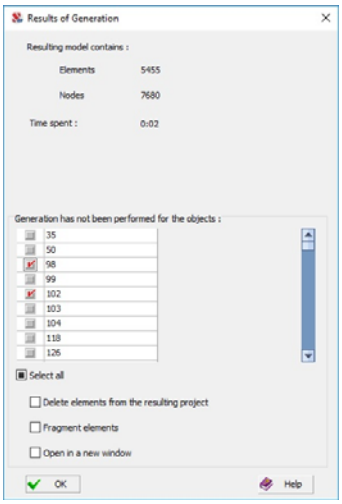


Figure 9.10-11. *The* **Results of Generation** *dialog box*

The **Results of Generation** dialog box (Fig. 9.10-11) appears after completing the generation of a finite element design model. It provides the information on the number of elements and nodes obtained in the resulting model, and also a list of planar objects for which the mesh generation failed for some reasons. By default these objects will be included in the resulting project as contours (elements of type 1080, 1060, 1085). If you check the **Delete elements from the resulting project** checkbox, the contours will be deleted. If the **Fragment elements** checkbox is checked, only these elements will be left on the screen of **FORUM**.

If you check the **Open in a new window** checkbox before closing this dialog box, the generated model will be loaded automatically.

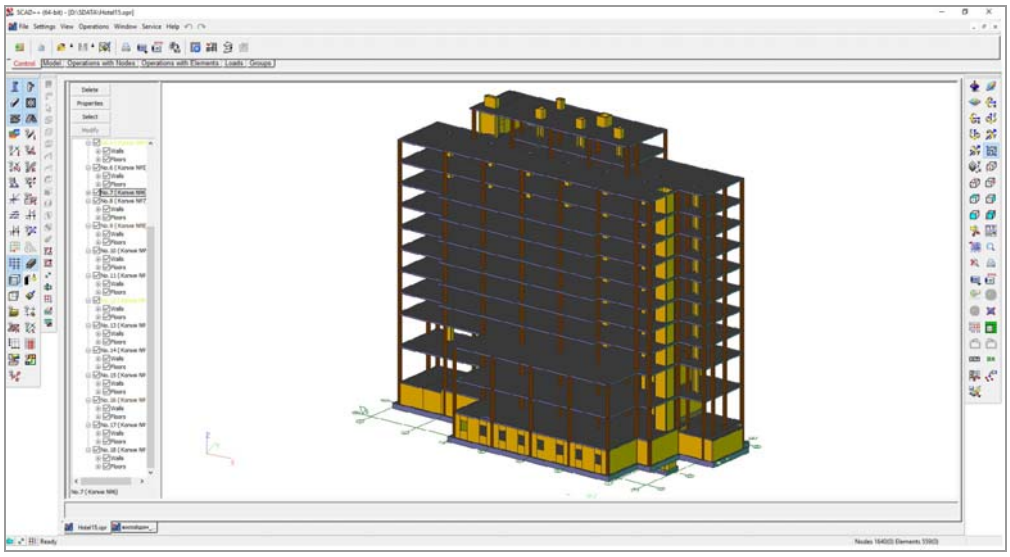


Figure 9.10-12. *Model generated in the* ***FORUM*** *preprocessor*

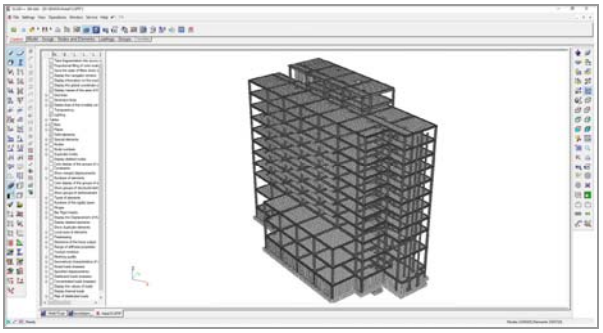


Figure 9.10-13. *Finite element model in* **SCAD** *complex*

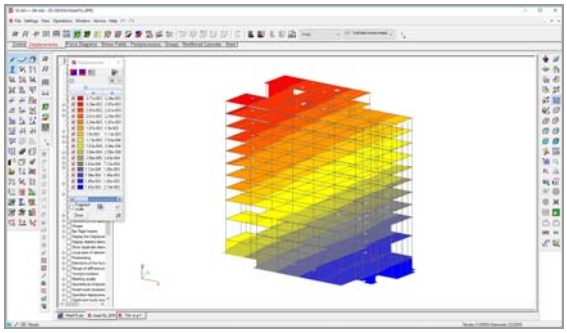


Figure 9.10-14. *Results of the calculation*

Fig. 9.10-13 shows the generated finite element model of the problem given in Fig. 9.10-12. Fig. 9.10-14 shows this model at the analysis of the results of the calculation in **SCAD** complex.

When generating a finite element model the directions of local axes of plate elements are not tracked automatically. By default, the direction of the output of forces in finite elements occupying a vertical position in the resulting model will coincide with the direction of the Z axis of the global coordinate system, and in other elements – with the direction of the X axis. If, on the basis of any considerations, for example, the direction of the layout of reinforcement bars, another direction of the output of forces is preferable, it can be specified with the help of the respective operation in **FORUM** or in a finite element preprocessor.

9.11. Connection with Architectural and Graphic Systems

Creating a design model for the strength analysis of bearing structural members of buildings and structures is quite a difficult problem. A design model is based on the geometric sizes and topology of the architectural part of the project. The widespread introduction of specialized software systems into the architectural design has enabled not only to automate the design process, but also to get a numerical model of the design object, i.e. its description in the database of the architectural system. Therefore the problem of organizing access to this information and creating a design model on its basis is very important.

Naturally, the design model is not a copy of the architectural solution. However, the main dimensions, snaps of the columns and bearing walls, floor outline, the position of openings and holes largely repeat those specified by the architect. This enables to perform an automated generation only of the integrated

design model based on the data of the architectural solution. Automatic generation of a detailed design model (but not complete) on the basis of the integrated model is possible only for a very narrow class of simple objects. This is due to the fact that most systems of strength analysis of structures are based on the finite element method, and for their design models, in addition to the geometric data taken from the architectural design and supplemented by generating a finite element mesh you also have to specify the boundary conditions, the data on the physical and mechanical properties of materials, and the information on loads. This requires specifying additional information necessary to perform the strength analysis.

It is necessary to perform the following actions to generate a design model on the basis of an architectural solution:

1. Generate the integrated design model of the building consisting of such objects as columns, beams, walls, floors (slabs) and the roof on the basis of the internal representation of data of the architectural system;
2. Delete objects from the structural model which are not included in the design model, for example, partitions, enclosing elements, architectural details, etc;
3. Adjust, if necessary, the position of objects in the design model and supplement it with new elements not taken into account in the architectural solution;
4. Automatic or user-controlled generation of a finite element mesh with the simultaneous specification of stiffness properties of finite elements;
5. Specify the boundary conditions of elements;
6. Specify loads and special initial data.

The last two operations are performed in **SCAD** complex, where additional modifications can be made to the design model.

Software tools for exporting data into formats of the **FORUM** preprocessor project are developed for such popular architectural systems as Revit Structure, ArchiCAD®, ALLPLAN®, and others.

9.12. Data Transfer from the FORUM Preprocessor to Monolit

This operation enables to prepare the initial data for **Monolit** based on the information on the structural (enlarged) elements created in the **FORUM** preprocessor (Fig. 9.12-1). In the general case, if the parameters of the structural elements and their position in the model do not contradict the rules used in **Monolit**, a file with the .opr extension will be generated after performing this operation, which can be imported into this program (Fig. 9.12-2).

The initial data can include structural elements of slabs, columns, beams, and walls. All these elements must belong to one storey (module).

If the model created in **FORUM** consists of one module, and the elements included in this module form a ribbed floor slab, then the file of this project with the .opr extension can be loaded into **Monolit** without using the considered operation. All elements with the characteristics contradicting the **Monolit** parameters will not be imported into the resulting project.

If the original project in **FORUM** includes floors lying in different levels, consists of several modules, or only a part of the floor is transferred to **Monolit**, the following procedure is recommended to prepare the model for export:

Variant A.

Fragment the original model in such a way so that only the exported part remains in the window (if the entire module is exported, all other modules have to be deactivated);

Invoke the **Export to Monolit** operation;

The **Export to Monolit** dialog box will appear, where you have to select the **Visible fragment** option;

Close the dialog box by clicking the **OK** button;

Specify the name of the file with the .opr extension with the exported model in the standard **Save As …** dialog box.

Variant B.

Select all elements you want to export in the design model;

Invoke the **Export to Monolit** operation;

The **Export to Monolit** dialog box will appear, where you have to select the **Selected elements** option;

Close the dialog box by clicking the **OK** button;

Specify the name of the file with the .opr extension with the exported elements in the standard **Save As…** dialog box.

Variant C.

Select the floors you want to export;

Invoke the **Export to Monolit** operation;

The **Export to Monolit** dialog box will appear, where you have to select the **Selected floors and adjacent elements** option;

Close the dialog box by clicking the **OK** button;

Specify the name of the file with the .opr extension with the exported elements in the standard **Save As …** dialog box.

The **Import** item of the **File** menu enables to load the project with the .opr extension to **Monolit**.

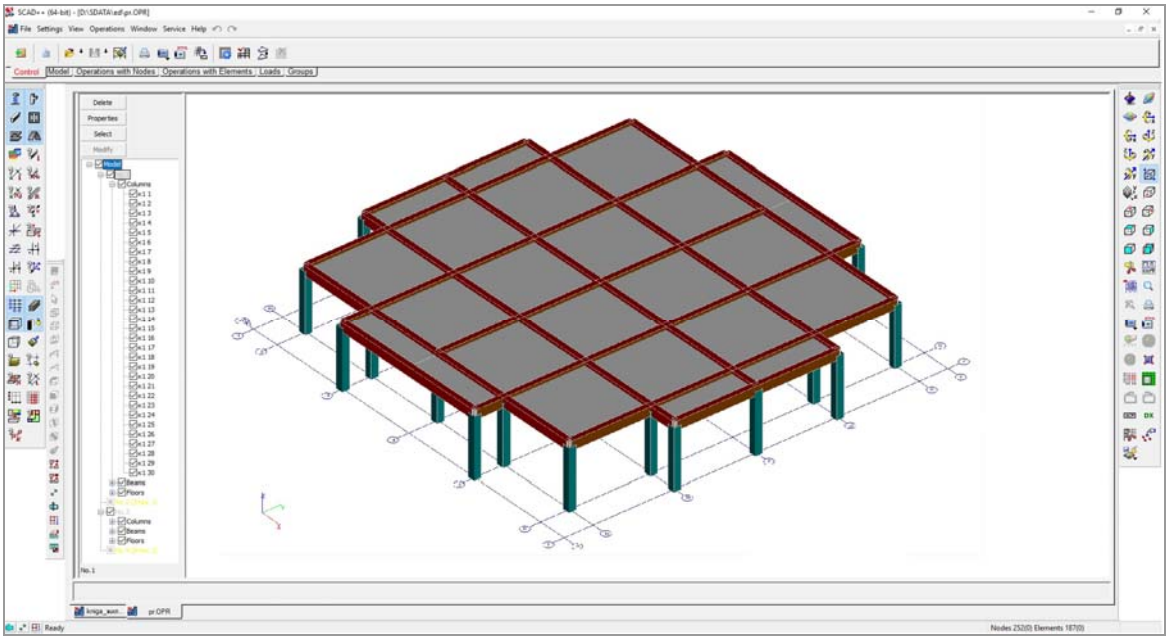


Figure 9.12-1. Model in the **FORUM** preprocessor

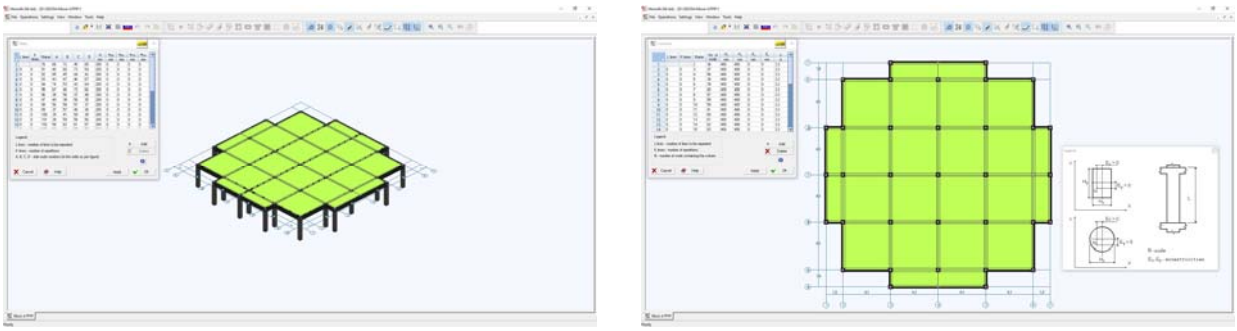


Figure 9.12-2. Model imported into **Monolit**

9.13. Example of Creating a Model in the FORUM Preprocessor

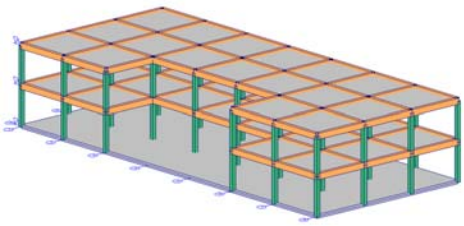


Figure 9.13-1. *Design model*

As an example, let's consider the generation of a very simple model (Fig. 9.13-1) in the **FORUM** preprocessor. The following procedure can be recommended to do this:

– (*action* 1) click the **New project** button — in the **Control** section of the toolbar, specify the identification data of the project and the units of measurement of the main values in the **New Project** dialog box (hereinafter, it is assumed that the dialog box is closed after specifying the data by clicking the **OK** button);

– (2) the **Save As…** dialog box will appear where you have to specify the name of the file the project will be saved in;.

– (3) click the **Specify grid lines** button — in the **Model** section of the toolbar and specify the parameters of grid lines in the **Specify Grid Lines** dialog box (click the filters button — **Display grid lines** to display the grid lines).

Figure 9.13-2. *The* **Create Nodes at the Intersection of Grid Lines** *dialog box*

– (4) Click the **Generate nodes on the grid** button — in the **Model** section of the toolbar and specify in the dialog box (Fig. 9.13-2) the boundaries of the area within which the nodes will be created on the intersection of grid lines. The elevations are considered as coordinates of nodes along the Z axis. The nodes will be displayed in the model after clicking the filters button — **Nodes**.

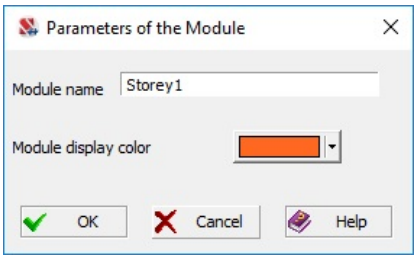


Figure 9.13-3. *The* **Parameters of the Module** *dialog box*

– (5) Click the **Create module** button — in the **Model** section of the toolbar and specify the name of the module and the color which will be used to display the module in the model in the **Parameters of the Module** dialog box (Fig. 9.13-3). A section (empty for now) with the elements of a new module will appear in the project tree after performing this operation.

The newly created module automatically becomes active, i.e. it will contain all the elements created immediately after creating the module.

– (6) Click the button with the image of the created element (for example, column) — in the **Model** section of the toolbar and specify the name of the column, the type of the section (parametric or rolled steel) and the input method in the **Rigidity of Bar Elements** dialog box (Fig. 9.13-4).

There are two methods for creating a column.

If the **Column height** radio button is selected, the column is created by specifying one snap node of the column, and the column height is defined by the value specified in the text field (if the value is negative – the column will be directed downwards from the snap point). Naturally, columns created

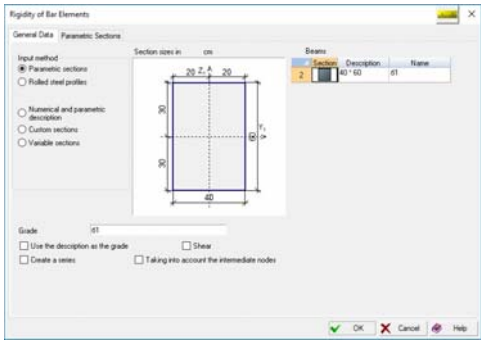


Figure 9.13-4. *The* **Rigidity of Bar Elements** *dialog box*

in such a way are always vertical. Specify the height of the column of 4 m.

The column section and the characteristics of the material are specified in the **Parametric Sections** tab (Fig. 9.13-5). Let's specify a square section 40x40 cm and click the **OK** button in the dialog box.

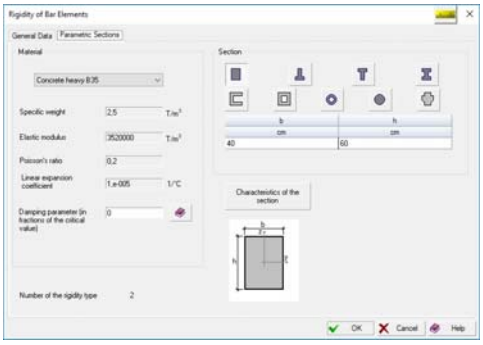


Figure 9.13-5. *The* **Rigidity of Bar Elements** *dialog box the* **Parametric Sections** *tab*

- (7) Place the cursor in the work area and click the right mouse button. The **Select Cursor** menu will appear, where you have to select the **Immediate execution** option and the **Cross with a frame** cursor.
- (8) Select all nodes lying at the 0,0 level with a rectangular marquee, and we will obtain a model shown in Fig. 9.13-6.
- (9) Beams — are always created by stretching the "rubber band" between two nodes. There are no restrictions on the position of the beam in the model. **General Data**, **Parametric Sections,** and **Rolled Steel Profiles** tabs are similar to the respective tabs used when creating columns.

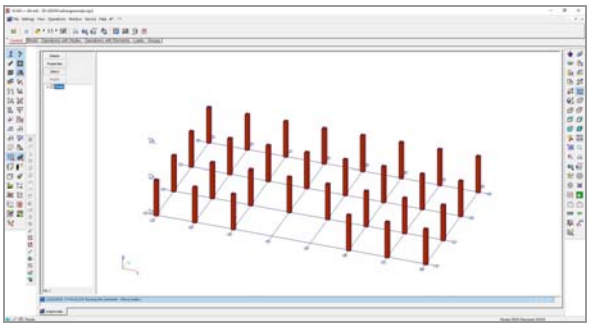


Figure 9.13-6. *Model after creating columns*

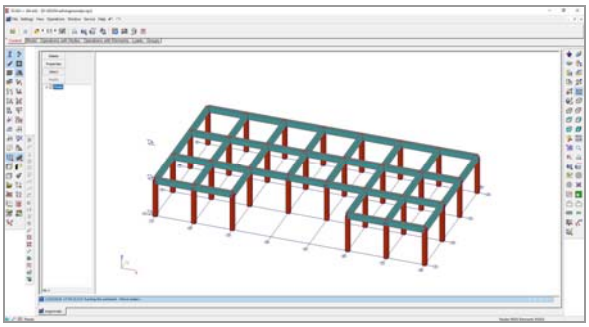


Figure 9.13-7. *Model after creating beams*

Let's specify a **rectangular** section 40×60 cm for beams and create beams by connecting the upper nodes of edge columns along numbered and lettered grid lines with a "rubber band". The obtained model is shown in Fig. 9.13-7.

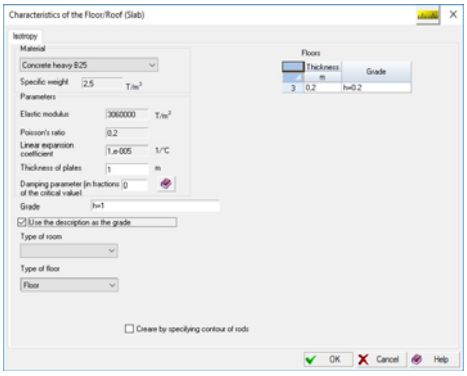


Figure 9.13-8. *The* **Characteristics of the Floor/Roof (Slab)** *dialog box*

- (10) The **Characteristics of the Floor/Roof (Slab)** dialog box (Fig. 9.13-8) appears after invoking the operation of creating a floor slab — . You have to specify the name of the floor slab (or of the group of floor slabs), its thickness and material in this dialog box.

Let's specify concrete B25 and thickness 0,2 m for the floor slab.

A floor is created as a flat multi-node element of an arbitrary shape. It is done with the help of a "rubber band" which is stretched from node to node and ends by clicking on the first node of the floor for the second time.

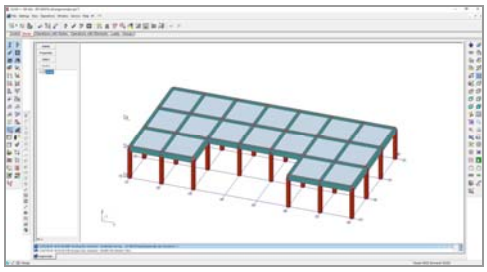


Figure 9.13-9. *Model after creating a floor slab*

The position of the created floor is shown in Fig. 9.13-9.

– (11) Let's use the operation of copying to create the second storey. In the **Copy the Model** dialog box (Fig. 9.13-10) specify the type of the object you want to copy — **Copy the selected module**, direction (Z) and step of copying (4 m), and the number of copies (1).

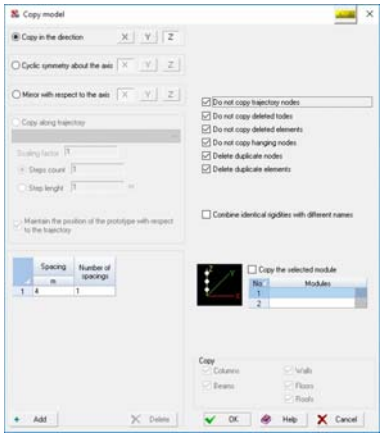


Figure 9.13-10. *The* **Copy the Model** *dialog box*

After specifying these data and closing the dialog box by clicking the **OK** button, the resulting model will appear on the screen (Fig. 9.13-11).

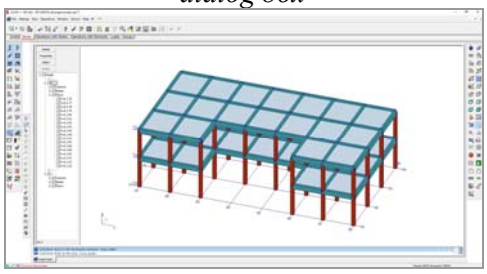


Figure 9.13-11. *Model after copying the second storey*

Model obtained in the result will include two storeys with the height of 4 m.

– (12) Let's create a foundation slab according to the rules given above (*action 10)*. The final model is shown in Fig. 9.13-12.

We have generated quite a typical integrated model in the result. The following operations were used to create it:

- creating grid lines and generating nodes on them;
- specifying parameters and elements of columns, beams and floor slabs;
- copying modules;
- 3D display of the model.

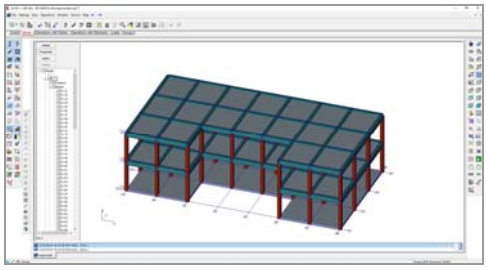


Figure 9.13-12. *Final model in the* **FORUM** *preprocessor*

– (13) The **Generate the resulting project** operation in the **Model** section of the toolbar enables to make a finite element model from an integrated one. The **Generate the SCAD Project** dialog box (Fig. 9.10-3) appears after invoking this operation, where you can specify the parameters necessary for the automatic generation of a finite element mesh. You have to specify the contour division step and mesh spacing for each type of element here.

Specify the spacing of 1 m for contours and all types of elements and click the **OK** button in the dialog box (Fig. 9.13-13). After closing the dialog box the standard **Save As …** dialog box appears where you can specify the name of the file of the resulting model.

The message window which accompanies the process of mesh generation provides comments to the operations performed and an indicator of the progress of the process.

The generation ends with the **Results of Generation** dialog box (Fig. 9.13-14). It provides the information on the number of elements and nodes obtained in the resulting model, and also a list of planar objects for which the mesh generation failed for some reasons.

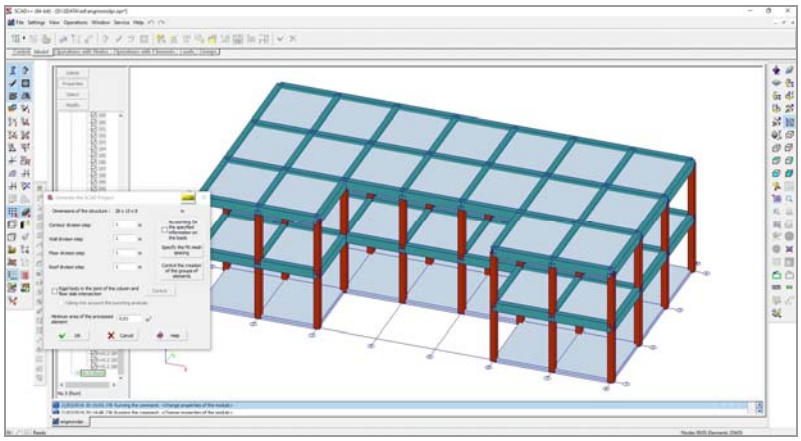


Figure 9.13-13. *Generate the **SCAD** project*

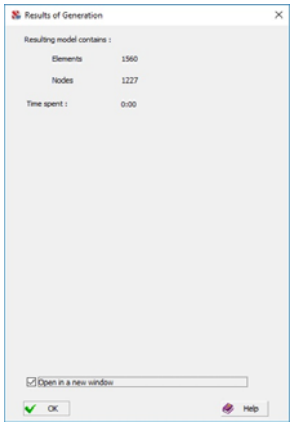


Figure 9.13-14. *The* **Results of Generation** *dialog box*

If you check the **Open in a new window** checkbox in the **Results of Generation** dialog box, the resulting model will be loaded automatically. It should be noted that all the information on the model including grid lines is transferred from **FORUM** to **SCAD**. The model in the SCAD preprocessor and the results of the calculation in the postprocessor are shown in Fig. 9.13-15.

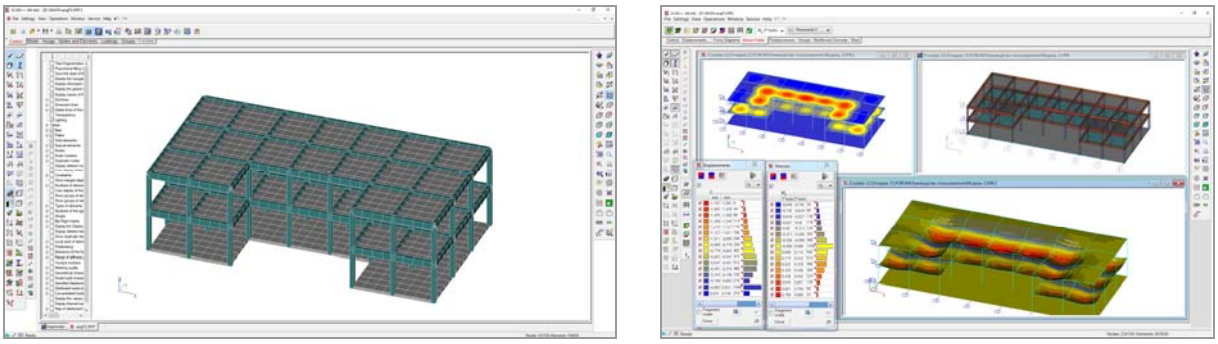


Figure 9.13-15. *Design model in the* **SCAD** *preprocessor and postprocessor*

10. Control of the Display of the Design Model

Various display control operations can be used when creating a design model and analyzing the results. They include a set of operations of fragmentation and positioning of the image of the model which can be invoked from the **Visualization** toolbar, and different display filters which enable to highlight and mark nodes and elements of the model based on the specified criteria. The **Display Filters** toolbar contains the filter control buttons.

10.1. Visualization Toolbar

The movable **Visualization** toolbar (Fig. 10.1-1) contains buttons for the control of the visualization of the design model. They enable to rotate the image of the model about the given axes, project the model on the planes of the rectangular coordinate system, and to perform different fragmentation operations.

Operations of the control of visualization have a priority over other operations. This means that if the mode of specifying the initial data is active (e.g., the mode of specifying the constraints) and one of the operations of visualization is invoked, the control will be given to the latter. The previously invoked mode stays active, and you can continue to work with it after performing the operation of visualization.

The position and sizes of the **Visualization** toolbar are saved and automatically restored after loading the complex. The button of setting and hiding the **Visualization** toolbar is located in the **Control** section of the toolbar.

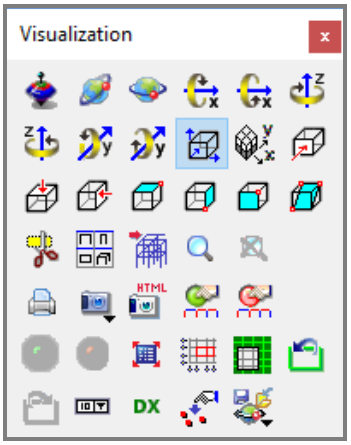


Figure 10.1-1. *The* **Visualization** *toolbar*

Rotation of the Model

The model can be rotated about the given axis by clicking the respective buttons in the toolbar. Each click rotates the model by one step, the value of which can be set using the **Set the rotation angle** button . In order to rotate the model by the specified step (30^0, 45^0, 60^0, 90^0, 120^0, 135^0, 180^0) you just have to place the cursor over the respective rotation button, and select the step from the drop-down list using the right mouse button. In order to restore the initial position of the model, click the button — **Restore the initial projection of the model**.

Set the Rotation Angle

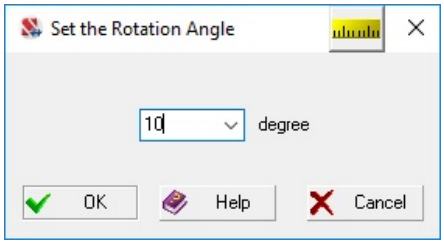


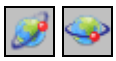
Figure 10.1-2. *The* **Set the Rotation Angle** *dialog box*

The **Set the Rotation Angle** dialog box (Fig. 10.1-2) is used to set the rotation step. The value of the angle can be selected or specified by the user in the window of the list. The specified step also acts in the case of the active operation of continuous rotation.

Continuous Rotation of the Model

If the button of the mode of continuous rotation of the model is *active*, clicking any rotation button will invoke the continuous rotation of the model in the given direction. In order to stop the rotation of the model you have to depress the button of continuous rotation.

Orbit



These buttons enable to invoke the mode in which the movement of the mouse (when the left button is pressed) rotates the model on the screen. The direction in which the model has to be rotated is defined by the direction of movement of the mouse. The free rotation can be performed by clicking the button , and the rotation only about the Z axis of the global coordinate system and the X axis of the screen projection — by clicking the button (which is usually used when working with models of buildings and structures). You can also rotate the model about the Z and X axis without clicking the button , and using a special combination Shift+"the middle mouse button". The center of rotation is the center of the fragment (including the grid lines and the origin of the global coordinate system, if the display of these objects is requested).

Project the Model on the Coordinate Plane

These operations enable to project the image of the model onto one of the planes of the global coordinate system. The selection of the projection plane is performed with the help of the respective buttons in the toolbar. The projection directions shown in the icons of the buttons are provided by default. In order to view the model “from the other side” you should use the operation of rotation about the respective axis having set the rotation angle of 180° beforehand, or select the necessary position of the model in the drop-down menu, which appears after right-clicking (Fig. 10.1-3).

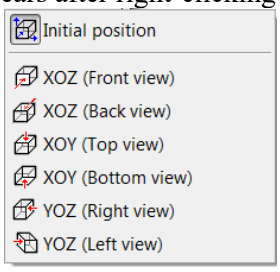


Figure 10.1-3. *Projection menu*

Click the button — **Restore the initial projection of the model** if you want to return the image of the model to its initial projection.

Isometric Projection

This operation enables to obtain an isometric projection of the image of the model. If you click this button, the projection will be generated in such a way so that the view of the observer will be directed towards the first octant. The selection of other octants becomes available in the drop-down menu which appears after right-clicking (Fig. 10.1-4.) The used octant numbering is given in Figure 10.1-5.

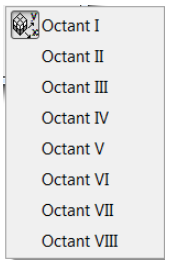


Figure 10.1-4. *Menu of isometric projections*

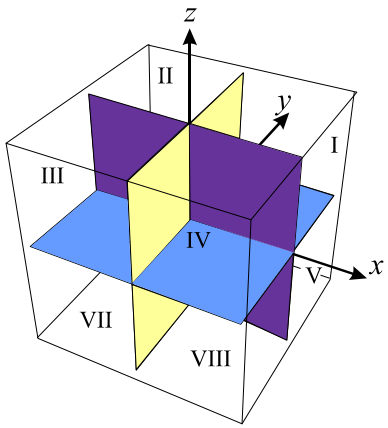


Figure 10.1-5. *Octant numbering*

Selecting a Planar Fragment

In order to select the fragment of the model lying in the plane parallel to one of the planes of the global coordinate system, it is necessary to specify the plane by clicking the respective button in the toolbar, and then select the node in the model belonging to this plane. More than one node belonging to different fragments of the model can be selected, and all these fragments will be selected in the result. In those cases when a fragment lies in an arbitrary plane, you have to select three nodes lying in the plane of the fragment. All elements and nodes belonging to this plane will be highlighted. If the button **Inverted fragmentation** is pressed, all nodes and elements *not* belonging to the respective plane will be selected.

If the selection is correct, click the green button to perform the operation, and if a mistake was made, click the red button, and repeat all the above actions.

When setting the modes of the cursor in the **Select Nodes and Elements** toolbar (see Section 6.1) you can reject clicking the confirmation button, i.e. automatically perform the active operations immediately after selecting the object of the operation. This mode can be set by activating the **Immediate execution** option — button in the **Control the Selection** toolbar. In this case the confirmation of the fragmentation is not required, and the fragment will be selected at once after selecting the node (nodes).

Click the button — **Initial display of the model** if you want to return to the image of the complete model.

Select a Fragment with a Marquee

In order to work with an arbitrary fragment of the design model you should use the operation of fragmentation by a marquee (**Fragmentation by a "marquee"**). A marquee can be rectangular or it can

have an arbitrary shape (Fig. 10.1-6 and 10.1-7). After selecting the operation a cursor of the rectangular marquee appears automatically. You can change the type of the marquee in the menu which appears after right-clicking.

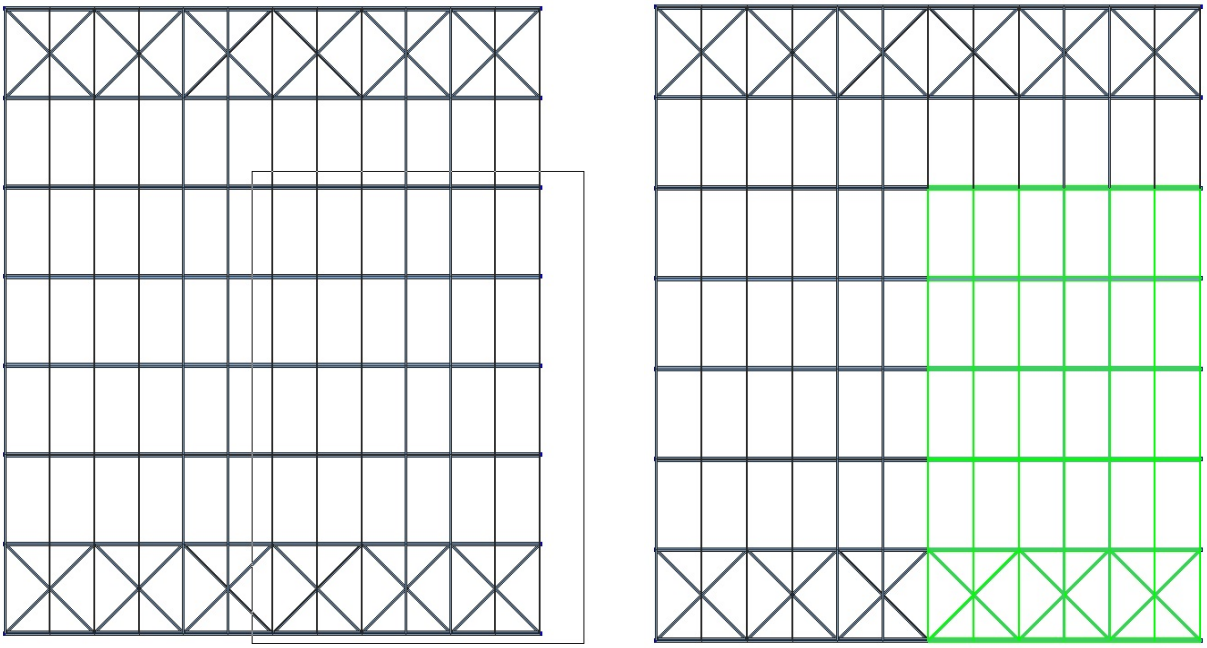


Figure 10.1-6. *Fragmentation by a rectangular marquee*

In order to select a fragment by a rectangular marquee it should be selected in such a way so that all the nodes and elements which should be included in the fragment are inside the marquee.

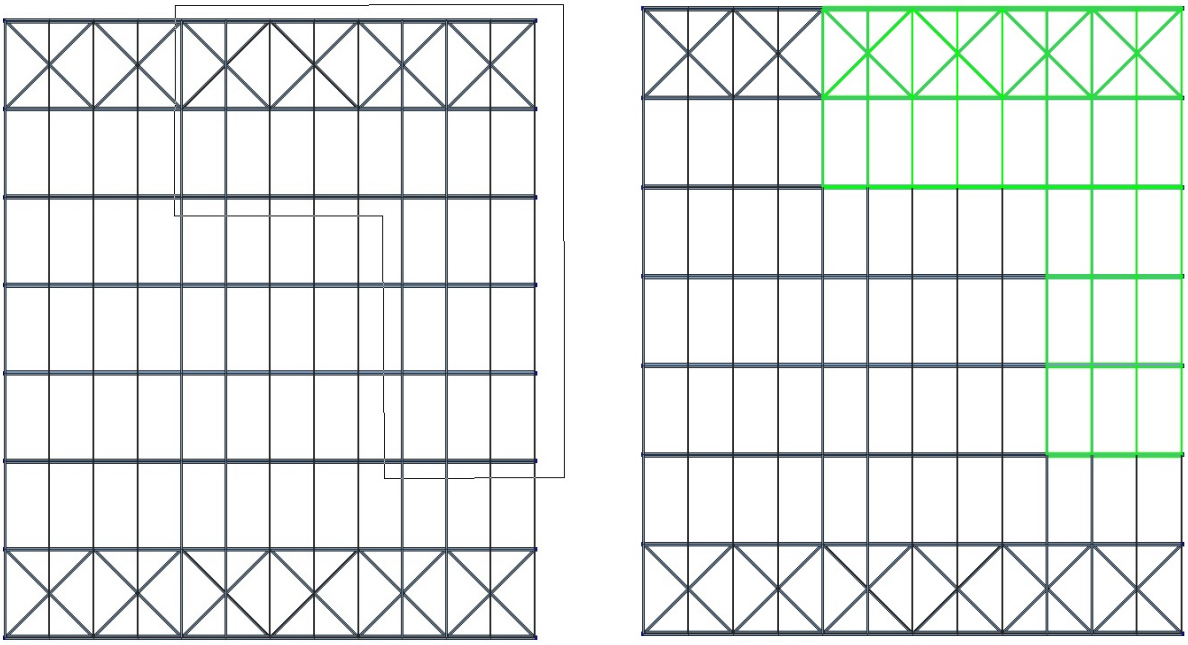


Figure 10.1-7. *Fragmentation by a polygonal marquee*

Marquee of an arbitrary shape is specified by a consecutive fixation of the vertices of the area of the fragment by the **left** mouse button. The area is closed, i.e. the first and the last points are connected automatically after double **left**-clicking. After setting the marquee the nodes within this area and elements all the nodes of which are inside this area will be highlighted.

If the button **Inverted fragmentation** is pressed, all nodes and elements *not* inside the marquee will be selected.

If the fragment is selected correctly, the confirmation is performed by clicking the green button . The rejection of the fragmentation is performed by clicking the red button .

If when setting the modes of the cursor in the **Select Nodes and Elements** toolbar the **Immediate execution** option is selected, the confirmation of the fragmentation is not required, and the fragment will be selected at once after closing the marquee. Only the elements all the nodes of which are inside the area of the marquee will be included in the fragment, and also individual nodes inside this area.

If there are no elements inside the fragment, only nodes will be displayed on the screen after clicking the button — **Nodes** in the **Display Filters** toolbar (Fig. 10.2-1).

Click the button — **Initial display of the model** if you want to return to the image of the complete model.

Cut on Projections

This operation enables to select a fragment of the design model by consecutively clipping an area on the projections of the model on the coordinate planes by a marquee. Four windows appear on the screen after its initialization. Three of them display the projections of the model on the XOZ, XOY and YOZ planes, and the fourth one displays the general view of the model and the control menu (Fig. 10.1-8).

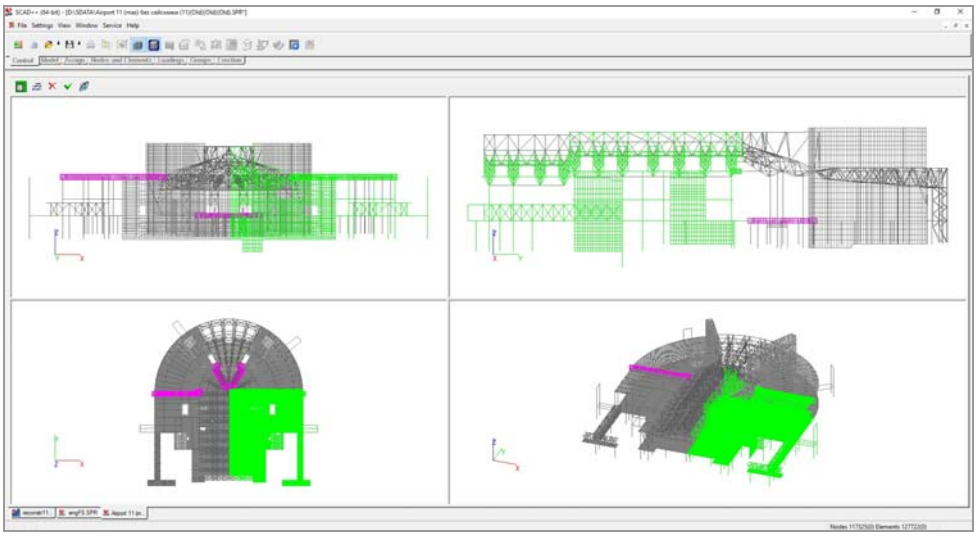


Figure 10.1-8. *Windows with the projections of the design model*

The rules for working with the operation of clipping on the projections are as follows:

- the windows of the projections are independent, and any of them can be increased or decreased;
- clipping can be performed in any order, many times and in any window;
- setting the cursor of a marquee (rectangular or of an arbitrary shape) and working with the cursors are similar to the function of selecting a fragment with a marquee;
- part of the design model within the fragment is highlighted;
- the clipping mode can be inverse, and in this case the fragment will include everything that was not included in the selected area;
- direct and inverse clipping may repeatedly alternate;
- setting the clipping modes and exiting the functions are performed with the help of the toolbar.

Figure 10.1-9. *Clipping control toolbar*

The clipping control toolbar (Fig. 10.1-9) enables to set the modes of inverted fragmentation — and displaying free nodes (not belonging to the elements) — , rotate the model — , cancel the fragmentation — or output the obtained fragment in the work area of the screen — .

You can exit the cut on projections mode by clicking the button in the toolbar. Click the button — **Initial display of the model** if you want to return to the image of the complete model.

Fragmentation on the Grid Lines

If grid lines were specified when generating the design model, a fragment can be selected by specifying the clipping area with the specified names of grid lines and/or elevations serving as the boundaries of this area. A ***clipping*** area can be selected in the **Fragmentation on the Grid Lines** dialog box (Fig. 10.1-10).

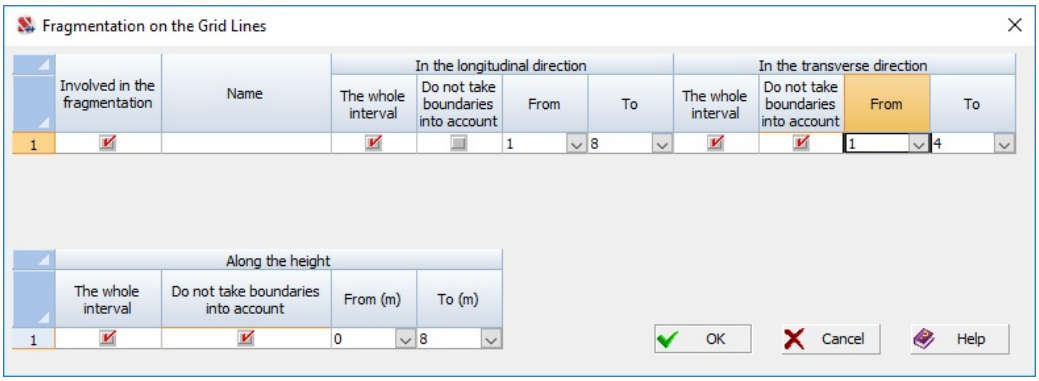


Figure 10.1-10. *The* **Fragmentation on the Grid Lines** *dialog box*

In order to select the necessary part of the model, you should perform the following actions for the respective areas of grid lines:

- check the **Involved in the fragmentation** checkbox;
- uncheck **The whole interval** checkbox if it is necessary to select not the whole interval but only a part for a certain segment;
- select the beginning and the end with the help of the **From** and **To** columns for the longitudinal and transverse directions;
- perform the similar actions with the elevations.

If **The whole interval** option is used, and a part of the model lies beyond the specified grid lines (for example, the elevations were not specified), then in order to obtain the full display of the model in the given direction you should check the **Do not take boundaries into account** checkbox.

Undo and Redo Fragmentation of the Design Model

These operations enable to perform the transition from one fragment of the design model to the other and back in the order of performing the fragmentation. Fragmentation can be performed by clipping a part of the model by a rectangular marquee or polygon, clipping on projections, by selecting the given plane or with the help of color scales.

Zoom in the Image

The **Zoom in the image** operation enables to zoom in the image of the design model and its fragment. Unlike the fragmentation by a marquee where only the selected part of the model is displayed on the screen, this operation enables to work with the whole model. The scale of the image changes in such a way so that the selected part of the model occupies the whole work area of the screen (Fig. 10.1-11), and the access to the other, invisible, part is performed by moving the image of the model (you should move the image while holding the middle mouse button). Thus, the user can move the image on the screen.

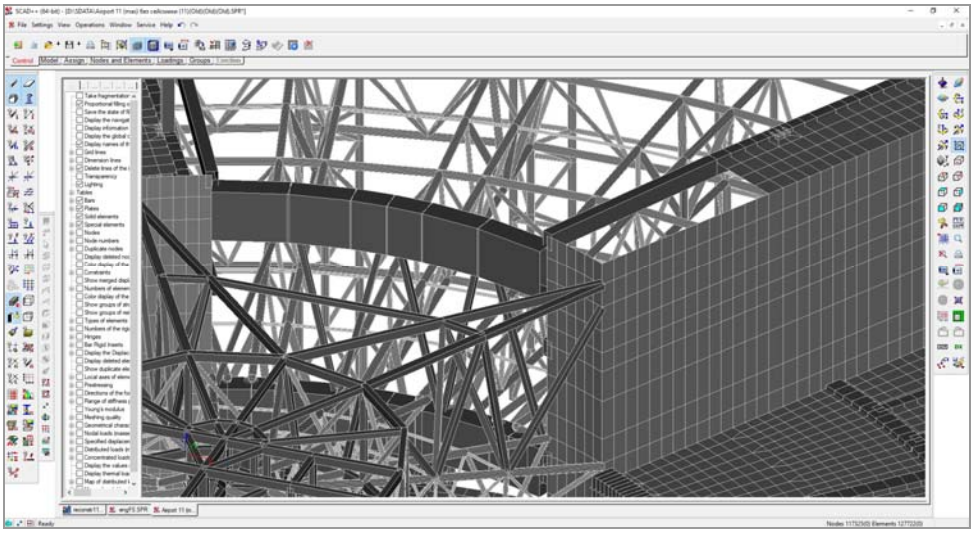


Figure 10.1-11. *Zoomed-in view*

After initializing the operation by clicking the respective button the magnified image is set similarly to selecting a rectangular fragment of the model.

If the proportions of the rectangle are close to the proportions of the window (height to width ratio), the obtained image of the part of the model will correspond to the fragment in the marquee. Otherwise, the fragment displayed on the screen will be slightly larger than expected.

The **Restore the original size** button restores the original size of the image.

Full Screen Mode

Full screen mode is used when working with large design models and enables to increase the work area by temporarily hiding the Windows taskbar, menu and the heading of the window. The image occupies the entire physical screen. Clicking the button for the second time returns the window to its initial state. If the window does not have the maximum size, invoking the operation leads to its maximization.

Rendering Engine

The software provides a variety of rendering engines. The first one is based on the DirectX technology. This technology provides the fast rendering of the design models. The high operating speed of DirectX is largely ensured by the fact that the technology is supported by all modern video cards. One of its disadvantages is the fact that in some cases the digitization and putting other text information over the image can have not very high quality. DirectX is used by default (except for the cases when **SCAD** detects problems with the support of DirectX — for example, when the operating system does not have the necessary version of DirectX). In this case the software *automatically* switches to the second rendering engine based on the GDI technology native for Windows. The user can also change the rendering engine. In order to do it, click the button in the toolbar and select an alternative rendering engine from the drop-

down menu. Switching to GDI is recommended only in the case when it is necessary to change the display of the text information. You should not do it in other cases, since the operating speed of GDI is much less than that of DirectX.

You should also keep in mind that you can not obtain an image with the effects of transparency, lighting and perspective in GDI.

Fragmentation Using the Tree

For the projects of the **FORUM** type this operation invokes a project tree where you can exclude individual modules (or parts of modules) from the fragment of the model.

Fragmentation of the Selected Objects

If there are selected nodes and/or elements in the design model, the fragment containing only the selected objects: selected nodes, selected elements and nodes belonging to them, will be left on the screen after clicking this button.

Fragmentation of Not Selected Objects

If there are selected nodes and/or elements in the design model, the fragment containing only objects which were not selected will be left on the screen after clicking this button, i.e. all the selected elements, selected nodes and the elements adjacent to them will be removed from the current fragment.

Search of Nodes with the Given Characteristics

This operation enables to select nodes occupying a certain position in the model or specified by their numbers. The search is performed for the following factors:

- distance from the point with the specified coordinates;
- distance from the specified nodes;
- distance from the specified elements;
- search among the nodes with the specified numbers.

These factors can be used in any combination in the search of nodes. Nodes satisfying all the above factors will be selected in the result. The search area can be specified as a sphere (**Radius** radio button) or as a parallelepiped (**Range** radio button). A parallelepiped is generated by changing the coordinates of the considered node or point to the following values $\pm\Delta x$, $\pm\Delta y$, $\pm\Delta z$.

Distance from the point with coordinates — the search of nodes lying in the vicinity of the point with the specified coordinates is performed;

Search among the nodes with numbers — the search of nodes with the specified numbers is performed;

Distance from nodes — all nodes lying in the vicinity of the nodes specified in the list are selected according to the rules described for the **Distance from the point with coordinates** operation;

Distance from elements — the search of nodes lying in the vicinity of the element is performed. Depending on the option defining the search area the evaluation of whether an *i*-th node is in the vicinity of the specified element is performed with respect to the nodes of the element (option — **Distance from nodes**) or any point belonging to the element (option — **Distance from body**). It is considered that a node satisfies the search conditions if at least one node of the element (**Distance from nodes**) or at least one point belonging to the element (**Distance from body**) is within the search area with the considered node in its center.

The control of the selection is performed in the dialog box which appears after invoking the operation. Depending on whether the **Select elements** checkbox is checked only nodes will be selected in the result of the search or nodes with their adjacent elements.

If the **Inversely** checkbox is checked, all objects the properties of which do not match those specified in the dialog box are selected.

The objects will be selected after closing the dialog box by clicking the **OK** button.

Save the Screen Image

After invoking this operation a menu appears, where you have to select the format of the copied image (vector or raster) and specify the storage method – in a file or in the clipboard.

The operation enables to obtain a metafile (extension **.wmf**), enhanced metafile (extension **.emf**) or placeable metafile (extension **.wmf**), or a raster image in ***.bmp**, ***.png**, ... formats. The obtained image can be included in the report, for example, when preparing the results of the calculation in the **Documenter** or in MS Word.

If you choose to save to disk, a standard dialog box appears, where you have to select the necessary format, specify the file name and the name of the catalogue it is saved into (it is saved into the working catalogue by default).

Save as HTML

The display of the design model, including all types of the graphical representation of the results of the calculation can be saved in the HTML format. This enables to view the image using the tools for visualization of HTML files of different browsers which have a WebGL support. Mouse buttons and a scroll wheel are used for the control of the display, including:

- ***rotation of the model*** on the screen is performed by moving the mouse while **holding the left mouse button**;
- ***an image*** can be ***zoomed out*** by **rotating the scroll wheel forward**;
- ***an image*** can be ***zoomed in*** by **rotating the scroll wheel backward**;
- ***an image*** can be ***moved*** on the screen by moving the mouse while **holding the scroll wheel**.

If the obtained image has to be put on the web-site or transferred to another computer, the x3dom directory, which is created in the same location as the HTML-file, has to be copied together with the HTML-file.

It should be noted that displaying images with a large number of texts (digitization of nodes/elements/results etc.) is extremely time-consuming for all existing browsers and can cause them to hang or fail to display the necessary information. Therefore, an image should be saved only in the case of a relatively small model or a fragment of a model.

Save and Restore the View of the Model on the Screen

When generating a design model and analyzing the results of the calculation, as a rule, a relatively small number of views of the model or of its fragments on the screen is used. A particular view can be used in the subsequent sessions if the information on it is saved as a file with data. This information is universal and can be used when working with any other design model.

In order to save the information on the current position, perform the following actions:

- select a view with the help of the display filters and click the **Save/Restore model position** button — ;
- a menu will appear where you have to select the **Save model position** item, and then select a directory and specify the file name in the standard **Save As…** dialog box;

- close the dialog box by clicking the **Save** button.

In order to restore a particular position of the model, perform the following actions:

- click the **Save/Restore model position** button — ;
- a menu will appear where you have to select a file name;
- after loading the information the model or its fragment will take the position on the screen corresponding to the selected view.

10.2. Display Information on the Design Model

General Comments on Displaying the Information on the Model

The control of the display of information on the design model is performed with the help of the filters. They enable to set not only the type of the displayed information, but also the rules of the visualization. The movable **Display Filters** toolbar contains the controls of the filters (Fig. 10.2-1).

In addition to the display control operations the filters toolbar also contains buttons which are often used when working with the model. They include, for example, buttons for obtaining information on a certain node or element, cancelling the selection of all objects, determining the distance between the specified nodes etc.

The multi-tab **Set the Information Display Filters** dialog box (Fig. 10.2-2) enables to set many filters. In order to invoke the dialog box, place the cursor over the **Display Filters** toolbar and ***right***-click. The setting is performed by enabling/disabling a number of different settings in the respective tabs of the filters. For example, you can set the display rules in the **Bars** and **Plates** filters, in accordance with which the elements occupying a certain position in the model will be shown (see Fig. 10.2-21).

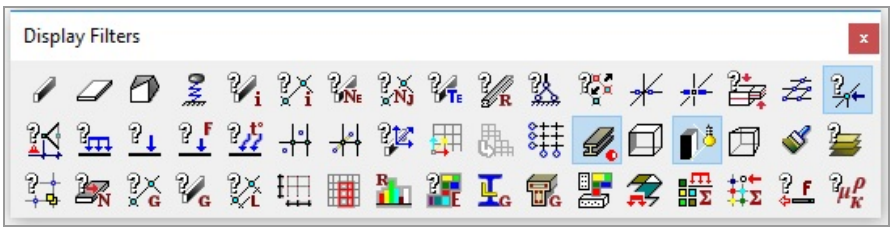


Figure 10.2-1. *The* **Display Filters** *toolbar*

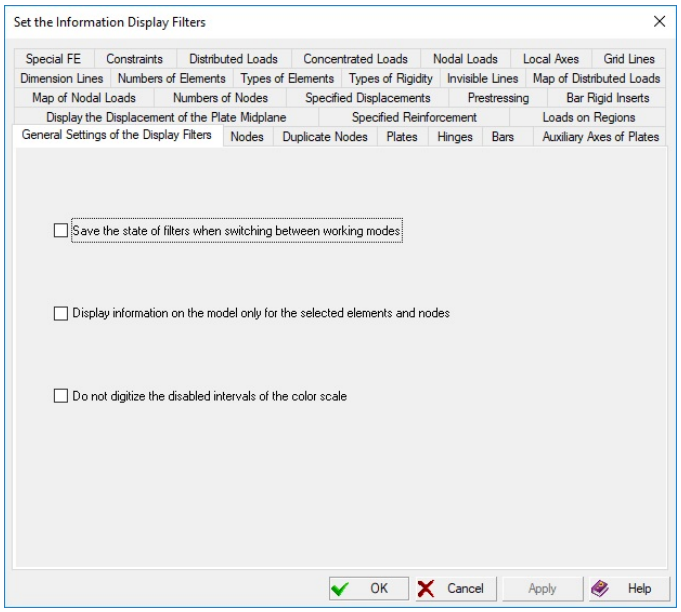


Figure 10.2-2. *The* **Set the Information Display Filters** *dialog box*
(*the* **General Settings of the Display Filters** *tab*)

Figure 10.2-2 shows the **General Settings of the Display Filters** tab of this dialog box which enables to:

- save the state of filters when switching between working modes;
- when the filters are active display the information not for all elements or nodes of the design model but only for the selected ones. Checking/unchecking the **Display information on the model only for the selected elements and nodes** checkbox will affect the display settings in the same way;
- do not digitize the elements/nodes corresponding to the disabled intervals of the color scale.

It is necessary to pay attention to some timing aspects of the visualization dealing with the filters setting. Even if you use the DirectX rendering engine, the output of the text information takes quite a long time. The more setting options are simultaneously enabled for different filters, the longer it will take to perform the functions of displaying the model on the screen.

The output time will increase insignificantly for small design models. However, if the model contains tens of thousands nodes and elements, the user will have to be patient and wait until it is displayed on the screen taking into account all the settings.

If the user has changed the setting of a certain filter, the respective button will have an additional indicator showing that the standard settings have been changed. For example, the display filter of bar elements (in the standard and modified states) is as follows: , .

The **Show/hide visualization toolbar** button is in the **Control** section of the toolbar.

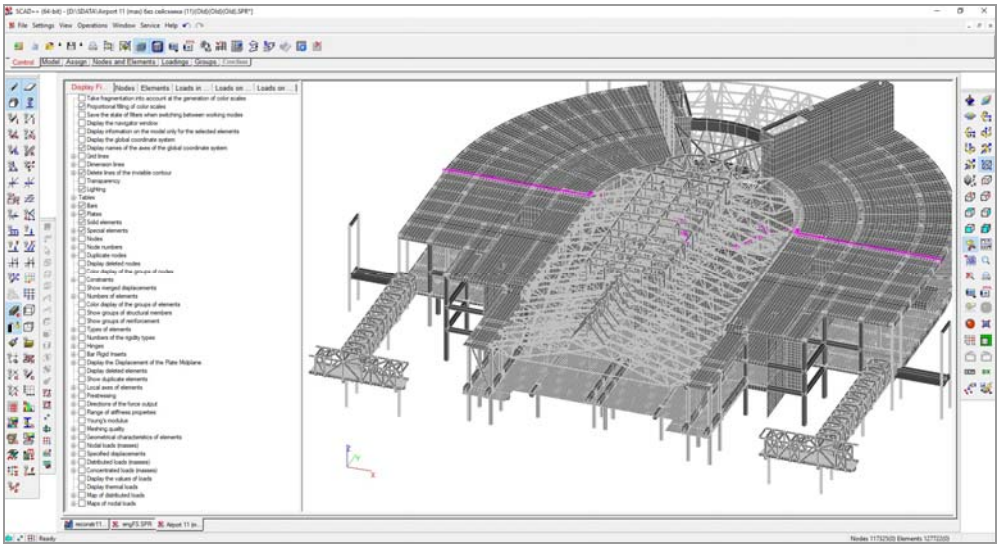


Figure 10.2-3. *Tree for setting the display filters*

The settings of the display filters are duplicated in the form of a tree in the left part of the window of each task. You can hide or expand the tree to the necessary width by moving the separator line (Fig. 10.2-3). The filters settings performed in the tree are duplicated in the filters panel and vice verse. Moreover, the **View** menu also contains the operation of disabling the tree and it has to be performed before loading the first task.

Saving and Resetting the Filters

The current state of the filters can be saved in the files with the extension **.SCADFilters**. Thus, it becomes possible to create a series of filter settings and enable the necessary variant depending on the working mode (pre- or postprocessor, bar or combined model etc.). The state of the display filters and of the visualization of the results is saved. When you load the task for the first time all filters will be in the initial state.

Working with files of the filters state is performed from the **Settings** menu in the **Save the state of filters…** (saving the current state) and **Reset filters…** (reading the necessary variant of the filters setting) sections. The access to the last several files is possible directly from the menu.

The compatibility of file formats for saving the filter settings is not guaranteed at the transition to new versions of SCAD. If the compatibility is not provided, a respective message will be generated if you try to load a file. In this case the files have to be created again.

Setting the Output of the Digital Information

Type, size, style and color of the font of the digital information on the design model are specified in the **Font Setting** tab (Fig. 10.2-4) of the **Parameters** dialog box, which is invoked from the **Settings** section of the ***Settings*** menu (or by clicking the button in the toolbar).

The description of this dialog box is given in section 34.3.

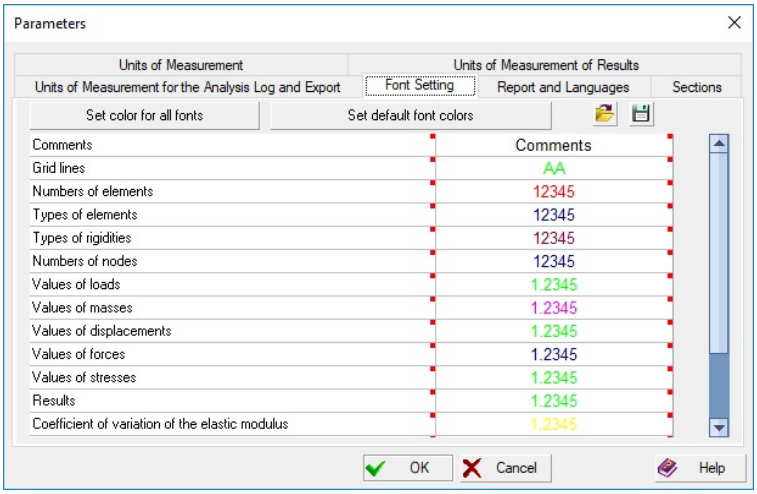


Figure 10.2-4. *The* **Parameters** *dialog box, the* **Font Setting** *tab*

Print the Design Model

Any displaying of the design model on the screen can be printed. The **Control** section of the toolbar and the **Visualization** toolbar contain buttons for invoking the printing mode. You can enter the information falling in the special field of the printed document in the **Text of Comments for Printing** dialog box (Fig. 10.2-5), which appears after invoking the mode.

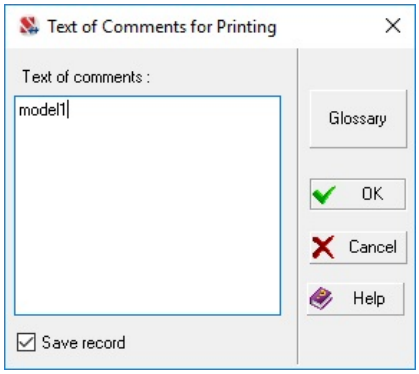


Figure 10.2-5. *The* **Text of Comments for Printing** *dialog box*

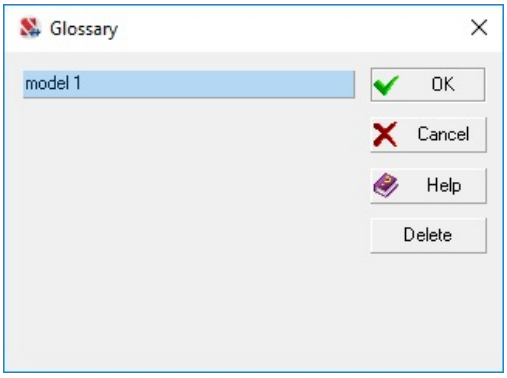


Figure 10.2-6. *The* **Glossary** *dialog box*

Printer setting is performed in the dialog boxes of the driver software supplied by the manufacturers of the equipment together with the printers. You should pay attention to such settings as the position of the image on the page: *Portrait format* (vertically) or *Landscape* (horizontally), and also to the color and

resolution controls. The isofields and isolines are printed on the color printer in the colors specified on the screen.

If the **Save record** checkbox is checked, the text of the comment will be saved and you will not have to enter it again, and you can just output it using the **Glossary** button of this dialog box (Fig. 10.2-6). You have to select a line with the necessary text in this dialog box and click the **OK** button; and the selected text will be transferred as the text of the comment. You can delete the unnecessary text using the respective button.

When printing in color you should pay special attention to the color of the text information (numbers of nodes and elements, types of rigidity, values of displacements etc.). Light colors (white, yellow, light gray etc.) are not visible on the paper. You should select such a font color which will be visible not only on the screen but on the paper as well.

The image is printed in accordance with the selected rendering engine (DirectX or GDI). Thus, if DirectX is selected, the program will print a raster image, and in the case of GDI a vector image will be printed. Usually the results of printing are almost the same, but if the image is output to a PDF-file (and largely magnified in the PDF viewer) the final images may differ significantly. The quality of the raster image in PDF-files can be improved if you increase the resolution of the output file, disable the reduction of the resolution for images, disable the compression of images with losses etc. in the settings of the respective PDF-printer. If all these actions do not give the necessary result, you can change the rendering engine to GDI before printing (you should remember about such limitations of GDI as the absence of support of lighting, transparency, perspective, etc.).

Reset Display Filters

If in the process it becomes necessary to cancel all filter settings made earlier, you can use the **Reset display filters** button. After pressing it, all filters will be disabled, and their settings will be transferred to the state assigned to them by default.

Display Information on the Nodes

This section considers functions of the display filters designed for working with a design model dealing with the display of nodes and information on them.

Output Numbers of Nodes

This filter is used to display the numbers of nodes on the design model. The filter settings (Fig. 10.2-7) enable to output the numbers and/or names of nodes, and the coordinates of nodes.

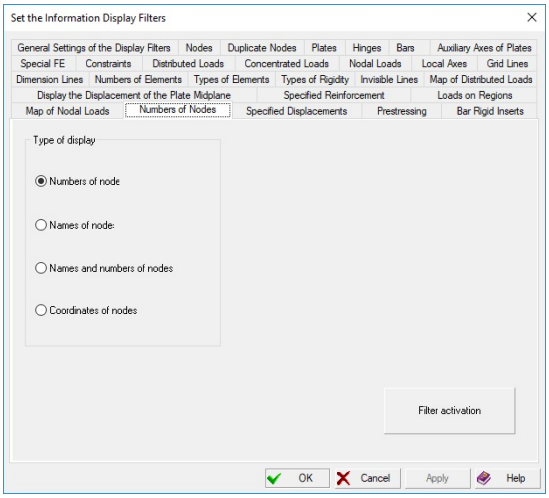


Figure 10.2-7. *The* **Numbers of Nodes** *tab*

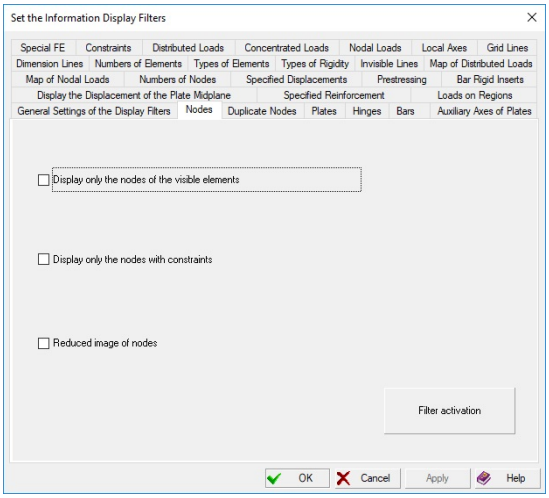


Figure 10.2-8. *The* **Nodes** *tab*

Display Nodes

Nodes are displayed on the design model as bordered squares the color of which depends on the performed operation. The following colors for marking nodes are assumed by default:

- displaying a node in its original form — blue;
- selected node in the modes of the parameter specification — red;
- selected node in the fragmentation modes — green;
- deleted and duplicate nodes — yellow.

This filter is enabled automatically in some modes, for example, in the modes of assembling the model from submodels, creating nodes, fragmentation etc., but it can be disabled by the user. You can cancel the automatic display of nodes by unchecking the respective checkbox in the **Environment Settings** dialog box, which is invoked by the respective command in the ***Settings*** menu.

The following rules for displaying nodes on the screen can be specified in the **Nodes** tab of the **Set the Information Display Filters** dialog box (Fig. 10.2-8):

- display only the nodes of the visible elements;
- display only the nodes with constraints;
- reduced image of nodes.

The size of the image of a node can be changed with the help of the respective operation in the **Design Model** tab of the **Environment Settings** dialog box.

Display the Deleted Nodes

If the operation of packing data was not performed, deleted nodes and elements are present in the model in the hidden form. This filter enables to display the deleted nodes. If necessary the nodes can be restored with the help of the respective operation in the **Nodes and Elements** section (if you invoke this operation, the filter of displaying the deleted nodes will be enabled automatically).

Filter setting is not provided.

Display Duplicate Nodes

Once this filter is enabled, the search of the duplicate nodes is performed, which are highlighted in yellow in the model by default. “Duplicate” nodes are nodes the distance between which is less than the value specified in the environment settings.

When setting the filter (Fig. 10.2-9) you can change the accuracy of estimation of the duplicate nodes, hide duplicate nodes with the specified merged displacements, and hide duplicate nodes used in one element.

Display Constraints

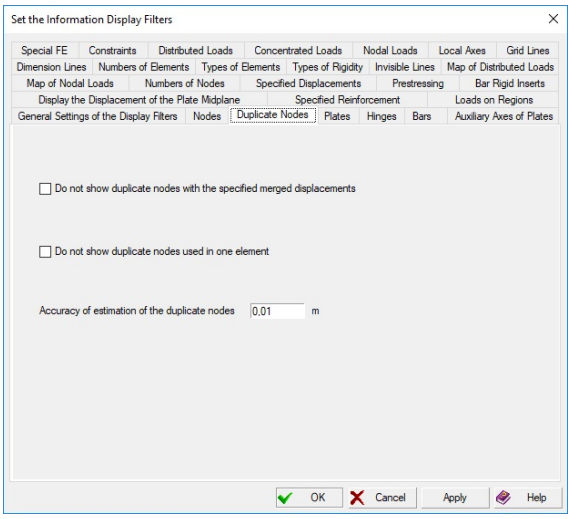


Figure 10.2-9. *The* **Duplicate Nodes** *tab*

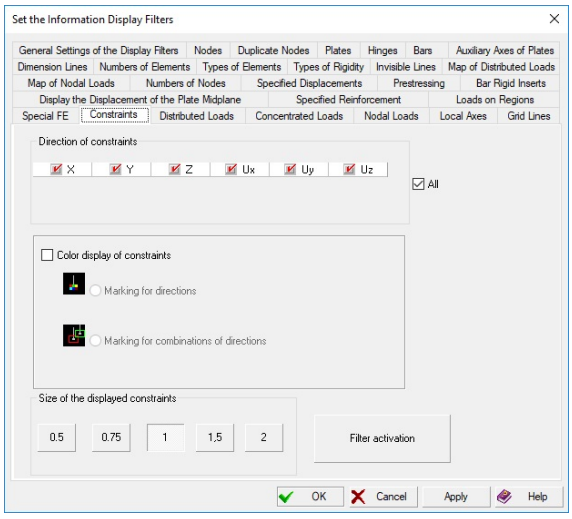


Figure 10.2-10. *The* **Constraints** *tab*

Constraints in nodes of the design model are displayed as rectangles. You can select the size of rectangles in fractions of the maximum value in the **Constraints** tab (Fig. 10.2-10) of the filters settings.

The control of the filter enables to display the constraints on the model only for the specified directions and to select the mode of the color display of constraint with marking for directions or for combinations of directions.

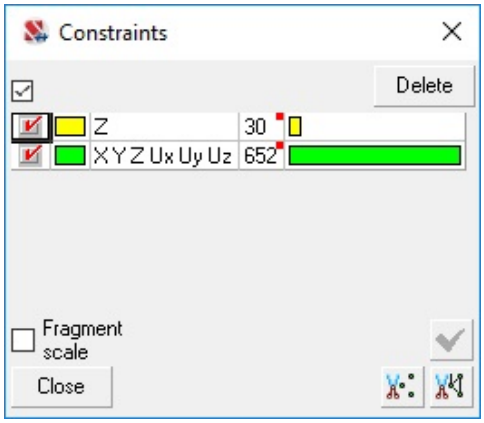


Figure 10.2-11. *The color scale of the display of* **Constraints**

The following displaying of constraints is possible if the color marking is enabled:

- rectangles with color fields, the color of which indicates the direction of the specified constraints. The upper three color fields in rectangles correspond to the constrains in linear directions, and the lower ones correspond to the constraints in angular directions. The setting of the color scale is performed in the **Color Settings** dialog box (see Sec. 10.5);
- the color scale of constraints (Fig. 10.2-11), where all the specified combinations of constraints in nodes are given. Using the possibilities of the color scale you can, for example, display only the nodes with the selected combinations (by checking/unchecking the checkbox) or delete the constraints from the nodes by clicking the **Delete** button.

The column marked with a red square indicates the number of nodes with constraints of the given type, and the next column provides this information in the form of a diagram.

Display Groups of Merged Displacements

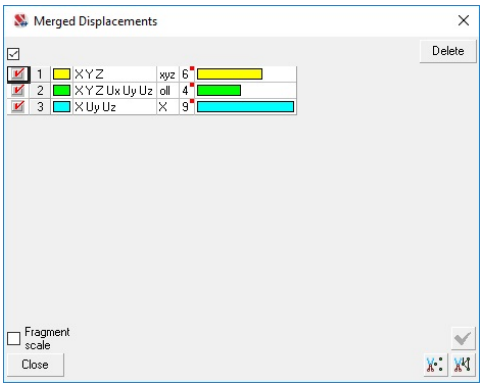


Figure 10.2-12. *Color scale of the mode for displaying the merged displacements*

When this filter is invoked a color scale appears (Fig. 10.2-12), where each group of nodes with merged displacements has its corresponding color and a line with the directions of merged displacements. The work with a color scale is performed according to the rules described above. Nodes included in the groups are marked with rectangles of the corresponding color in the design model.

You can cancel the merging of displacements for nodes of the selected groups using the **Delete** button.

The column marked with a red square indicates the number of nodes included in the group of merged displacements, and the next column provides this information in the form of a diagram.

Determine the Distance between Nodes

This operation enables to determine the distance between two nodes of the design model. Once it is invoked the **Distance Between Nodes** dialog box (Fig. 10.2-13) appears displaying the results of the measurement.

In order to determine the distance select the first node in the model (let's call it the basic node), which is marked by a yellow ring, and then the second one (it is marked by a purple ring). Coordinates of the selected nodes, displacements between nodes given by three coordinates and the distance between them are displayed in the respective text fields.

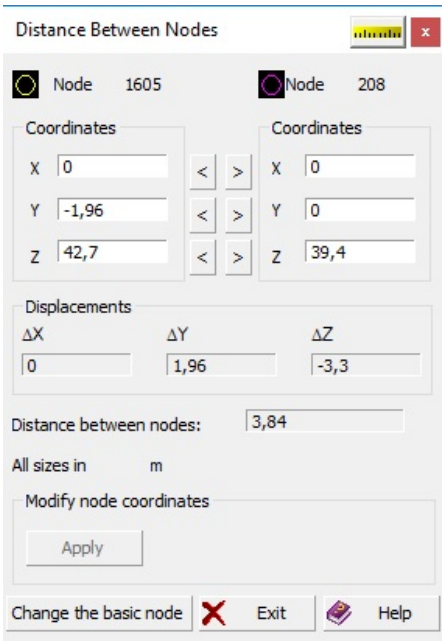


Figure 10.2-13. *The* **Distance Between Nodes** *dialog box*

In order to determine the distance between the basic node and the other node of the design model you just have to select this node in the model without activating the filter again. In order to change the basic node, use the respective button in the dialog box. You can specify a new basic node after clicking it.

The coordinates of nodes can be corrected in the initial data input mode, i.e. they can be moved into the points with other coordinates. In order to do it, specify the new values of the coordinates in the respective text fields and click the **Apply** button.

Values of the coordinates of one node can be assigned to the other one with the help of a set of buttons located between the text fields of the node coordinates in the **Distance Between Nodes** dialog box (Fig. 10.2-13).

Coordinates can be moved only between the values of one direction. Otherwise, they have to be input using the keyboard. In order to perform the operation click the button near the text field of the changed coordinates and click the **Apply** button after duplicating the value into the field of the other node. The results are displayed in the design model.

The standard operation **Undo** is used to undo an operation.

If this operation is invoked in the postprocessor, the **Displacement** button appears, which enables to obtain the displacements of the selected nodes, displacements for each coordinate axis and the total displacement in the special dialog box (Fig. 10.2-14).

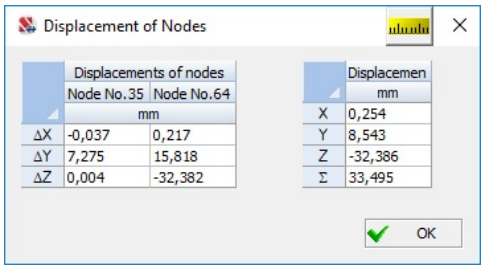


Figure 10.2-14. *The* **Displacement of Nodes** *dialog box*

Information on the Node

This operation enables to obtain the complete information on the given node, including the information on the coordinates, constraints, loads, adjacent elements, and displacements (in the analysis of the results of the current loading). The **Information on the Node** dialog box contains the controls of the operation (Fig. 10.2-15).

In order to obtain the information click the filters button and specify the method for selecting the node in the model in the dialog box. In order to find a node you can specify its number in the **Node number** field and click the **Find** button or select a node with a cursor if the **Indication on the model** button was pressed before. The selected node is marked by a green circle in the model, and the information on it will be displayed in the dialog box.

The coordinates of the node are output in the **Coordinates** group, and the values of displacements are output in the **Displacements** group (only in the mode of the analysis of the results).

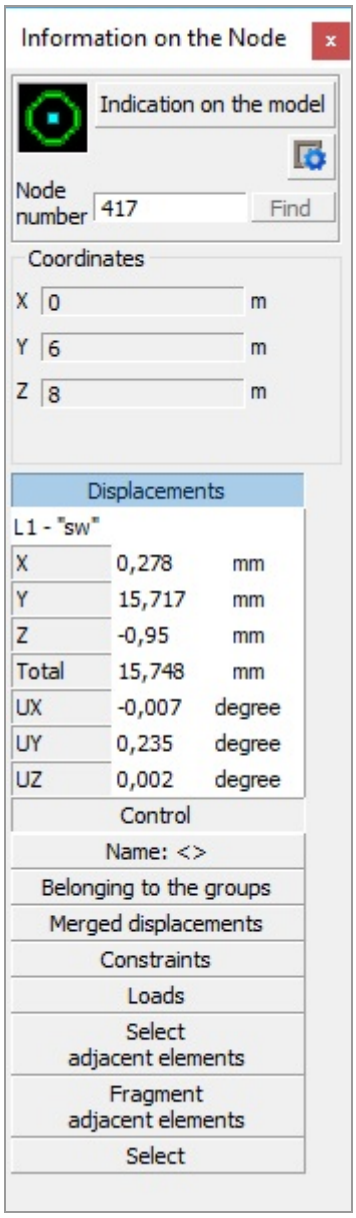


Figure 10.2-15. *The* **Information on the Node** *dialog box*

In the mode of generating the design model the coordinates of the node can be changed directly in the **Information on the Node** dialog box. The changes are saved by clicking the **Apply** button. The **Constraints** button invokes the dialog box where you can view (and change in the preprocessor) the constraints in the selected node.

Clicking the **Loads** button invokes the **Loads on Node** dialog box (Fig. 10.2-16) with the information on all loads applied to the node in each loading. This mode enables to selectively delete the loads from the current (active) loading or change the load values. In order to delete it select a line with the load you want to delete in the list which is specified as **Current** (loading), and click the **Delete** button.

The value of the load selected in the list is output in the respective information window. In order to correct the load value, select it in the list, change the value in the text field and click the **Modify** button.

The load will be deleted or changed only in the case if the loading is saved using the button in the **Loadings** section.

The **Reactions in constraints** and **DCR** (design combinations of reactions in constraints) buttons will be available in those cases when the respective calculations were performed, and a constraint was imposed in the considered node. Once they are invoked, tables with the results of these calculations will be displayed in the respective dialog boxes.

If the design combinations of displacements were obtained in the result of the calculation, the **DCD** button will be available in the **Information on the Node** dialog box. Clicking it makes the table with the combinations of displacements accessible.

Similarly, if the design combinations of punching were obtained in the result of the calculation, the **Design combinations of punching, Punching loads** and **Punching loads for combinations of loadings** buttons will be available in the **Information on the Node** dialog box. Clicking them makes the tables with the results accessible.

The **Fragment adjacent elements** button enables to create a fragment containing only all the elements adjacent to the node. Clicking the **Select adjacent elements** button makes all the elements adjacent to the node selected.

If necessary, the considered node can be deleted or marked as selected. The respective buttons are used to perform these operations.

The **Belonging to the groups** button invokes the dialog box containing a table with all groups of nodes the selected node belongs to. You can exclude a node from the group by unchecking the checkbox near the respective group and clicking the **OK** button. The **Show all groups** checkbox enables to obtain the list of all groups of nodes. If you check the checkbox near the group the node does not belong to and click the **OK** button, the node will become an element of this group.

If a design model has groups of merged displacements, similarly to groups of nodes you can check whether a selected node belongs to a certain group of merged displacements and/or exclude or include it in a group (Fig. 10.2-18).

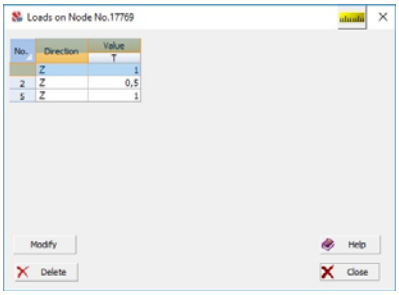


Figure 10.2-16. *The **Loads on Node** dialog box*

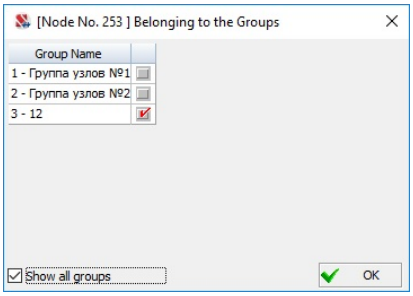


Figure 10.2-17. *The **Belonging to the Groups** dialog box*

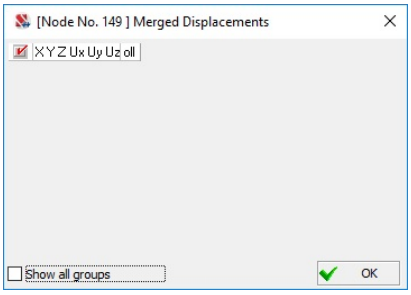
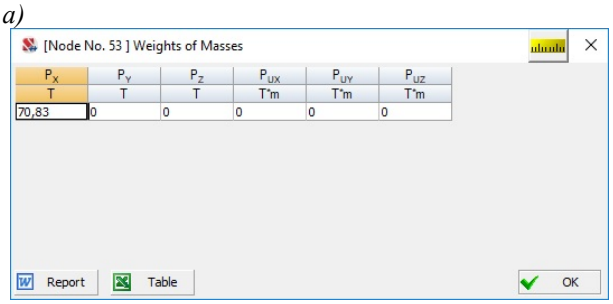
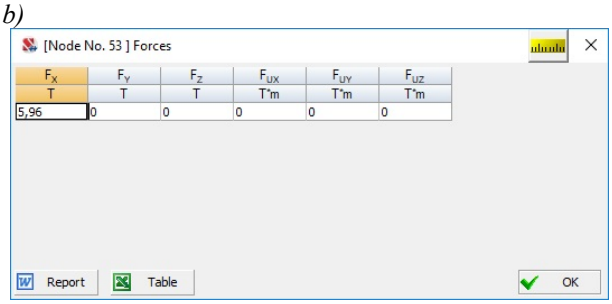


Figure 10.2-18. *The **Merged Displacements** dialog box*

a)



b)



c)

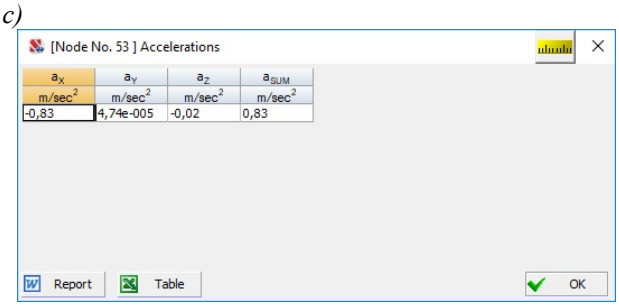


Figure 10.2-19. *Dynamic data on the node: a) nodal masses; b) inertial forces; c) accelerations*

Values of displacements and rotation angles will be displayed for the selected node in the graphical analysis of the results of the calculation for the current loading (oscillation modes, buckling modes, combinations of loadings). If the direct integration of the equations of motion is analyzed, it becomes possible to view the graphs of the variation of the components of displacements with time (see Chapter 15), and if the non-linear analysis is performed, you can view the stepwise displacement history of the nonlinear loading (provided that the number of steps with the saved results is greater than one). The postprocessor also has buttons which enable to obtain tables with reactions in constraints (see Sec. 12.6), design combinations of displacements (see Sec. 18.8), and the loads from the fragment of the model (see Sec. 22.3). If the current loading is a dynamic one, the respective buttons enable to obtain a tabular representation of the weights of nodal **masses**, dynamic **forces** and **accelerations** (Fig. 10.2-19).

The **Information on the Node** dialog box may occupy any position on the screen (or outside the work area of **SCAD**). You do not have to close it when performing the operations of specification or correcting the geometry of the model.

In those cases when the set of buttons or information provided by default in the **Information on the Node** dialog box includes items which are not used or rarely used, it is possible to configure the dialog box to display only the necessary data. To do this, use the **Settings** operation, which is invoked by clicking the respective button in the considered dialog box or by right-clicking. After invoking this operation, a menu appears, which enables to include or exclude these items from the dialog box. The settings will be used later in all dialog boxes and for all models. If the user has changed the setting, the button **Settings** will have an additional indicator showing that the standard settings have been changed.

Display Information on the Elements

You can output the following information with the help of the display filters for finite elements:

- numbers, types of elements, numbers and names of the types of rigidity;
- presence of hinges and rigid inserts in bar elements;
- directions of the local axes of elements and the force output axes in plate and solid elements;
- diagrams of deflections and forces, combinations of forces, reinforcement for bar and plate elements (in the information on element in the mode of the analysis of results).

Most of the information can be output in the digital form or in the form of a color scale. It should be noted that only one color scale for elements can be output at a time.

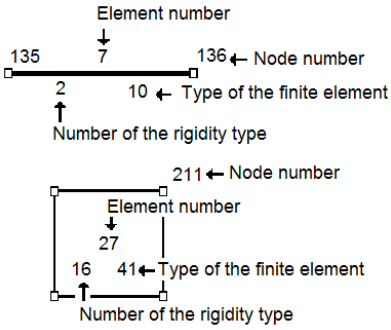


Figure 10.2-20. *Position of attributes in bar and plate elements*

If you activate the filters with the attributes of the elements, the information will be output in accordance with the model shown in Fig. 10.2-20. "Readability" of the information will largely depend on the sizes of the elements on the screen. If the sizes are small, the attributes usually can not be written within the field of the plate or bar. In these cases it is recommended to select a fragment with the necessary part of the model or set the **Zoom in the image** mode in the **Visualization** toolbar.

The current setting of the filters is saved before closing the project or exiting to the project tree. In the initial state (by default) the information is output in the numerical form for all nodes and elements of the model.

Display Filters of Elements

These filter buttons are used to enable or disable the display of elements of a certain type — bars, plates, solid or special elements, and if you are working with enlarged elements they are used to enable or disable the display of columns, beams, walls, slabs and roofs. All buttons are active in the initial state, i.e. all types of elements are displayed in the model. If they are disabled, the respective types of elements will not be displayed in the model.

A more detailed setting of filters is performed in the multi-tab **Set the Information Display Filters** dialog box in the **Bars** and **Plates** tabs (Fig. 10.2-21), where the display rules are set according to which the elements occupying a certain position in space in the model will be displayed.

The tab for setting the display of special finite elements (Fig. 10.2-22) contains checkboxes for different types of special elements, checking or unchecking which enables to display only the necessary types of elements on the screen. There are additional checkboxes for selecting the directions of constraints of finite rigidity.

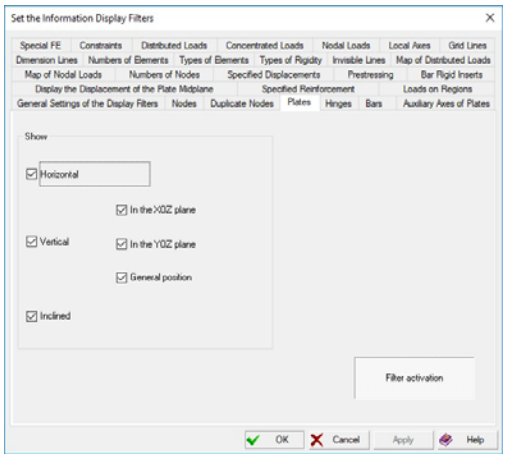


Figure 10.2-21. *The* **Set the Information Display Filters** *dialog box*, *the* **Plates** *tab*

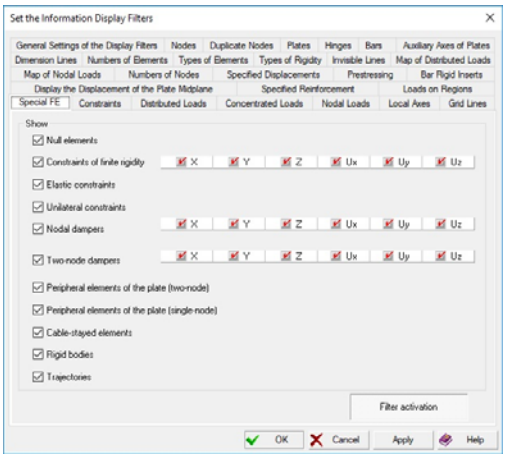


Figure 10.2-22. *The* **Set the Information Display Filters** *dialog box*, *the* **Special FE** *tab*

Display Numbers of Elements

The **Numbers of Elements** tab of the **Set the Information Display Filters** dialog box contains the controls for displaying the numbers of elements (Fig. 10.2-23). This tab enables to specify the display of numbers on the model depending on a certain type of elements, or on the position of the elements in the model.

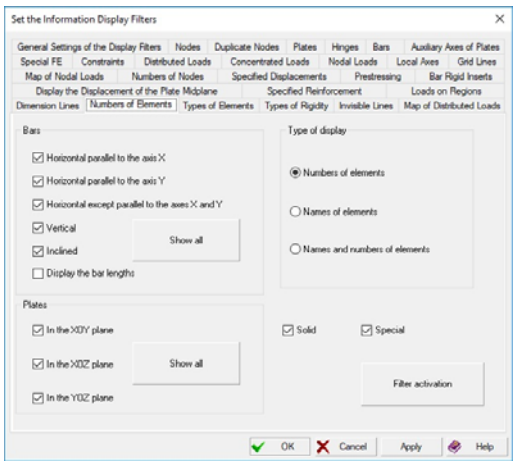


Figure 10.2-23. *The* **Numbers of Elements** *tab*

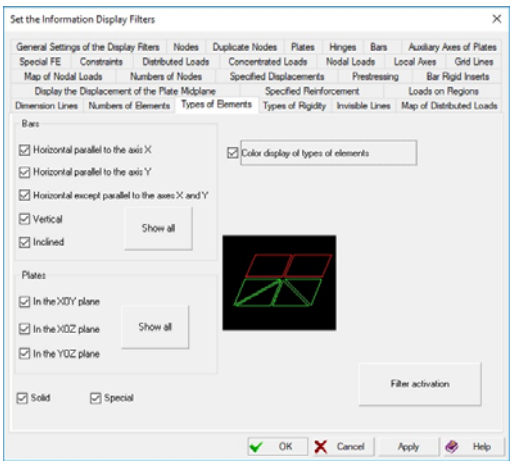


Figure 10.2-24. *The* **Types of Elements** *tab*

Checkboxes of the **Bars** group are used for working with bar elements and enable to specify the output of numbers of bars depending on their position in the model. If the **Display the bar lengths** checkbox is checked, the values of lengths of elements are displayed instead of the numbers of elements.

Checkboxes of the **Plates** group are used for specifying the rules for outputting the numbers of plate elements depending in their belonging to the coordinate planes. Moreover, there are two checkboxes for enabling or disabling the display of numbers of solid and special elements. You can also display names of elements instead of their numbers, or display the names and numbers simultaneously.

The **Font Setting** tab enables to specify the type, color and size of the font (see Sec. 34.3).

Display the Types of Elements

The **Types of Elements** tab of the **Set the Information Display Filters** dialog box contains the controls of the filter of the element type display (Fig. 10.2-24). The controls of this tab are used to set the rules for displaying the information depending on the type of elements, and on their position in the model. Moreover, you can also use colors for the indication of different elements by their type on the model.

Four groups of checkboxes are provided for setting the filter. First three groups enable to set the filters similarly to the filter of the numbers of elements. Checkboxes of the fourth group determine the rules of the color indication.

The color indication mode (the **Color display of types of elements** checkbox is checked) enables to assign a color for each type of finite element. The **Types of FE** dialog box with a color scale appears (Fig. 10.2-25). The controls of this dialog box enable to perform the following operations:

- check or uncheck all the checkboxes of the color scale (the **Check/uncheck checkboxes**);
- apply the specified combination of checkboxes of the color scale to the model (the **Apply** button);
- fragment the model in such a way so that it has only the elements the type of which coincides with the checked checkboxes of the color scale (the **Fragment** button);
- disable the mode of the color indication (the **Close** button).

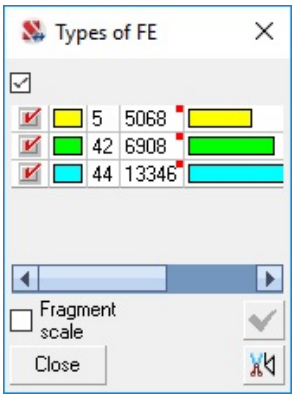
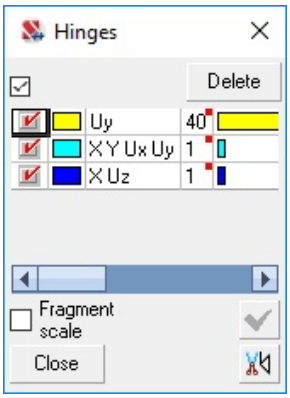


Figure 10.2-25. *Color scale of the indication of the types of elements*

a)



b)

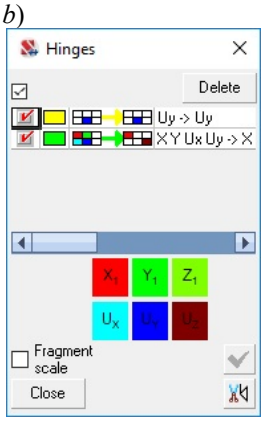


Figure 10.2-26. *Color scales of hinges*
a) as color checkboxes; b) by coloring the bars

The column marked with a red square indicates the number of elements of the respective type, and the next column provides this information in the form of a diagram.

In order to change the color double click on the color field and select a new color in the standard Windows dialog box. Double clicking on the cell with a type of the finite element invokes the dialog box where you can change the type of FE.

When displaying the numbers of the types of elements in the numerical form the type, color, and size of the font are specified in the **Font Setting** tab (see Sec. 34.3).

Display Hinges

Regardless of the specified direction(s) the hinges are displayed in the model as rings at the ends of the flexible part of the bar elements or near the nodes (depending on the parameters of the hinges).

The settings of the filter provide the color display of hinges and elements containing hinges (Fig. 10.2-26). In the first case each combination of released constraints at the ends of the flexible part of the elements obtains its color in the color scale and in the model. In the second case bars are marked in color in the model depending on the combinations of hinges specified for them.

The column marked with a red square indicates the number of hinges of the given type in the model, and the next column provides this information in the form of a diagram.

Display Deleted Elements

If the operation of packing data was not performed, deleted nodes and elements are present in the model in the hidden form. This filter enables to display the deleted elements. If necessary the elements can

be restored with the help of the respective operation in the **Nodes and Elements** section (if you invoke this operation, the filter of displaying the deleted elements will be enabled automatically).

Filter setting is not provided.

Display Duplicate Elements

Once this filter is enabled, the search of the duplicate elements is performed, which are highlighted in yellow in the model by default. “Duplicate” elements are elements of the same type all the nodes of which coincide.

Display Types of Rigidity

The setting of the display filter of the types of rigidity is performed in the **Types of Rigidity** tab (Fig. 10.2-27) similarly to the setting of other filters.

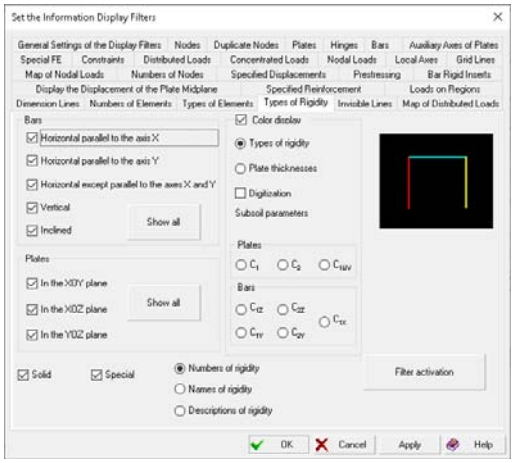


Figure 10.2-27. *The* **Types of Rigidity** *tab of the* **Set the Information Display Filters** *dialog box*

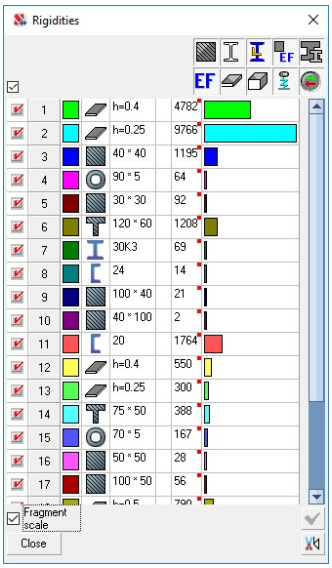


Figure 10.2-28. *Color scale of the indication of numbers of the types of rigidity*

If you specify the color display of the types of rigidity, you will obtain a scale shown in Fig. 10.2-28, where each type of rigidity, except for color, has a corresponding button with an icon displaying the type of elements or a section of bar elements. The next column contains the following characteristics of the type of rigidity:

- bar elements, specified by an explicit description of the section, — sizes of the section or grade of the section from the assortment of rolled profiles;
- bar elements, specified by the numerical description of the rigidity —axial stiffness;
- plate elements — thickness of the element;
- solid elements — elastic modulus of the material;
- rigid bodies, elastic constraints, constraints of finite rigidity — direction of the constraint;
- unilateral constraints — rigidity of the constraint etc.

If the names were given to the types of rigidity, they are output in the column following that with the characteristics. The last two columns contain data on the number of elements with this type of rigidity given in the numerical form and in the form of a diagram.

Bar elements the rigidity of which is defined by the characteristics of parametric sections will have icons in accordance with the section type — , etc. Sections from the assortment of rolled profiles are displayed by “their” icons: , , … If the rigidities are described numerically, and also for special

finite elements the button will have the following form . If the numerical-parametric description of rigidities is used, the button will take the following form — . And the button appears for sections which were prepared with the help of **Section Builder**, **Consul** or **TONUS**. Elements without specified stiffness properties are displayed in yellow (by default) in the model and are indicated by the icon in the scale.

The general symbol — is used for all types of plate elements, and — is used for solid ones.

Clicking these buttons invokes dialog boxes with the respective stiffness properties where you can perform ***only the correction*** of the parameters of the types of rigidity. They differ from the similar dialog boxes which are invoked from the **Assign** toolbar and are described in detail in Sec. 7.2 in that they do not have the **OK** button, and the new parameters are saved by clicking the **Replace and exit** button.

When specifying the color display in the **Set the Information Display Filters** dialog box you can output the color scales of the thicknesses of planar elements, subsoil parameters C_1, C_2 or C_{1UV} of plates or those C_{1Y}, C_{2Y}, C_{1X}, C_{1Z} or C_{2Z} of bars (Fig. 10.2-29). You can switch to the coefficients of another type directly in the dialog box of the color scale of the subsoil parameters (for example, switch from the Winkler coefficients C_1 to the Pasternak coefficients C_2).

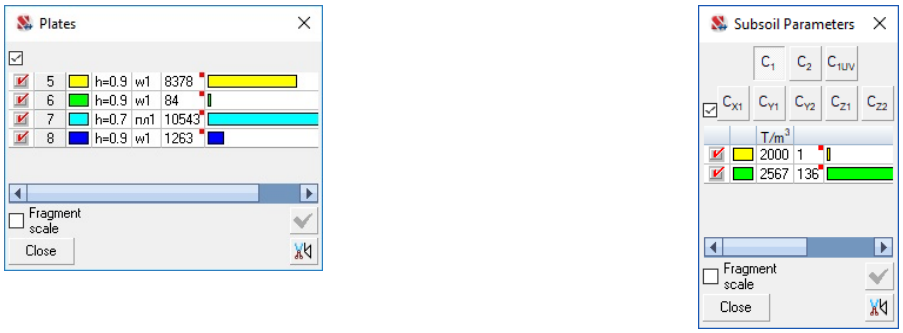


Figure 10.2-29. *Color scales of the indication of the thicknesses of plates and the subsoil parameters*

Moreover, you can use the settings to display the names of the types of rigidity or descriptions of rigidity on the design model, and simultaneously output the color display and the numbers of the types of rigidity (digitization).

It should be noted that the output of the color scheme of rigidities (and other characteristics as well) is possible not only on the "wire" model, but also at the active filter for deleting the lines of the invisible contour, including the displaying of sections of bars and thicknesses of plates (Fig. 10.2-30).

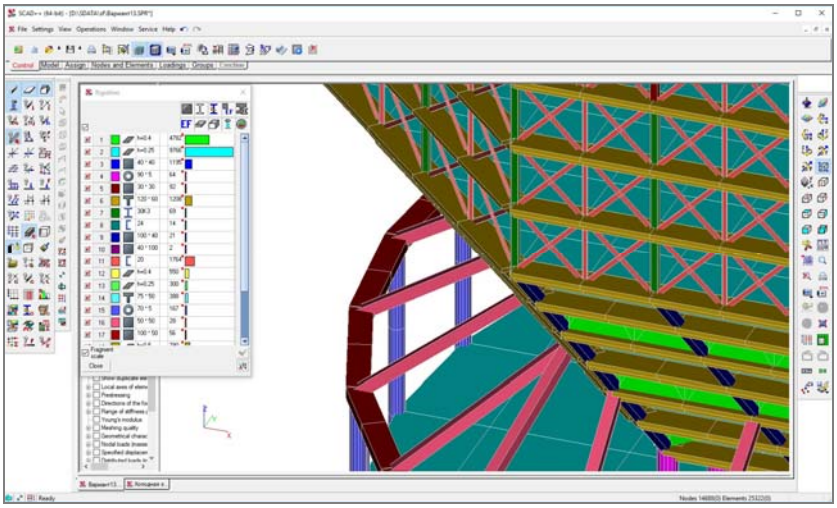


Figure 10.2-30. *Color display of the types of rigidity taking into account the sections of bars and thicknesses of plates*

Ranges of Stiffness Properties

This operation enables to perform a comparative analysis of the stiffness properties of elements. The analysis is performed for the bar and plate elements for the following parameters:

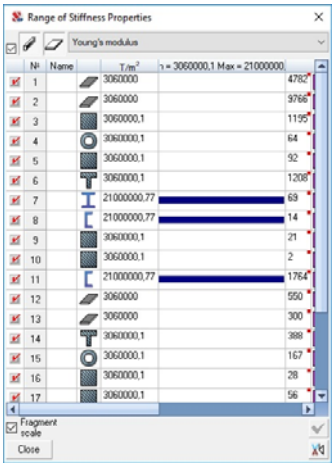


Figure 10.2-31. *The* **Range of Stiffness Properties** *dialog box*

- bars and plates — elastic modulus, Poisson's ratio, specific weight, damping parameter;
- bars — axial and bending stiffness;
- plates — thickness, cylindrical stiffness...

The spectrum of stiffness properties is output in the respective dialog box (Fig. 10.2-31), where you can specify the following settings:

- select the type of the analyzed factor from the drop-down list;
- using the checkboxes in the table select only the types of rigidity you are interested in;
- double left click on the line of the respective type of rigidity to invoke the dialog box where you can correct the stiffness properties.

The column marked with a red square indicates the number of elements with the respective type of rigidity, and the next column provides this information in the form of a diagram.

In all other respects working with this dialog box is similar to working with any other color scale.

Young's Modulus

This filter enables to obtain a color scale with the values of the elastic modulus in different finite elements of the design model.

Display Rigid Inserts

Rigid inserts are displayed on the design model in the scale and for the directions specified for them. It should be noted that if the display filter of rigid inserts is disabled, a line connecting the nodes will be displayed in the model, but not the element. This line may not correspond to the actual position of the element in the model. In some cases this filter is activated automatically in the postprocessor. This occurs, for example, at the display of the diagrams of forces (since they relate to the flexible part of the bar) or at the display of the deformed model.

The setting of the display filter of rigid inserts in performed in the **Bar Rigid Inserts** tab (Fig. 10.2-32). Three types of displaying can be selected here: the geometric one described above, the color scale (Fig. 10.2-33) or simultaneously both the geometric display and the color scale.

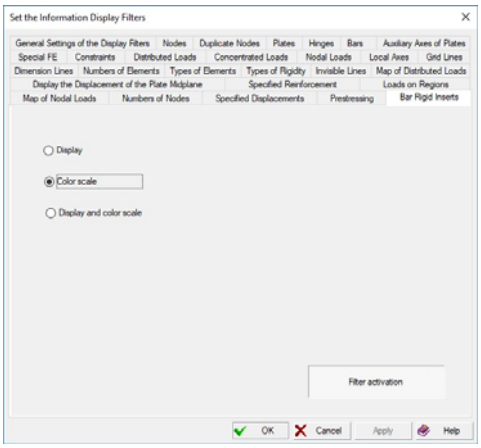


Figure 10.2-32. *The* **Bar Rigid Inserts** *tab*

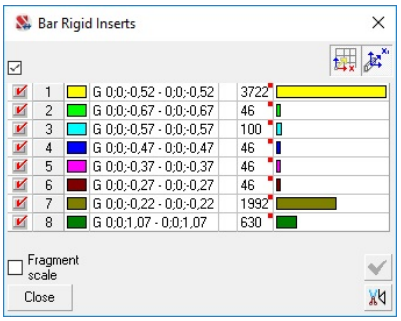


Figure 10.2-33. *The* **Bar Rigid Inserts** *dialog box*

Display the Displacement of the Plate Midplane

Data on the displacement of the plate midplanes can be displayed similarly to the rigid inserts of bars: geometrically, as a color scale, or simultaneously both the geometric display and the color scale.

Display Directions of the Local Axes of Elements

Local axes of elements are displayed in color as three axes located in the center of the element. Moreover, plate elements can be displayed in color depending on the direction of the local Z_1 axis. The setting of the filter is performed in the **Local Axes** tab (Fig. 10.2-34).

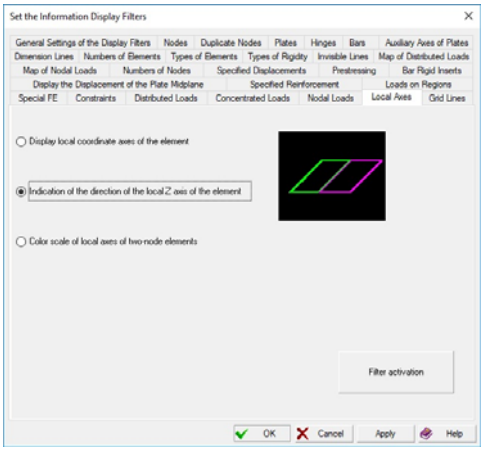


Figure 10.2-34. *The* **Local Axes** *tab*

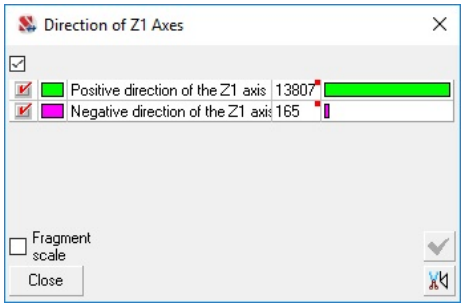


Figure 10.2-35. *The* **Direction of Z_1 Axes** *dialog box*

If the radio button of the color indication of plates is enabled, the **Direction of Z_1 Axes** dialog box of the color scale appears after activating the filter (Fig. 10.2-35). The controls of this dialog box enable to select a fragment of the model which includes only the plates the local Z_1 axis of which is directed upwards (positive direction) or downwards. For elements parallel to the YOZ plane the positive direction of the Z_1 axis coincides with the direction of the X axis of the global coordinate system. For elements parallel to the XOZ plane the positive direction of the Z_1 axis coincides with the direction of the Y axis of the global coordinate system. The rules for working in this dialog box are the same as for the color scale. It should be noted, that the concepts of “positive” and “negative” directions are conditional. The upward direction of the Z_1 axis is assumed to be positive for the horizontal elements.

If the **Color scale of local axes of two-node elements** checkbox is checked, two-node elements with the values of the angles of orientation of the principle axes of inertia different from the default ones (specified by the user) are displayed in color in the design model.

Display Directions of the Force Output

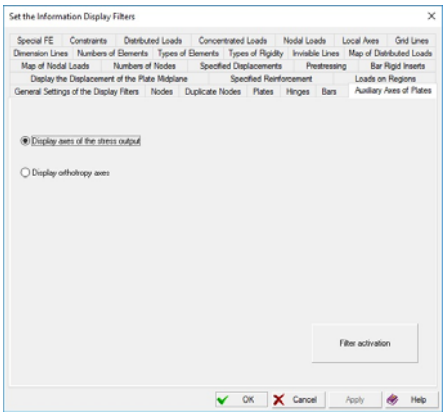


Figure 10.2-36. *The* **Auxiliary Axes of Plates** *tab*

This filter is used to display the directions of the output of forces in plate and solid elements. Two arrows are output in the center of each solid element indicating the direction of the X_N and Y_N axes of the output of forces. After activating the filter an arrow is output in the center of each plate element the direction of which coincides with the direction of the output of forces specified for it. Forces in the given direction are considered as those directed along the local X_N axis. In the initial state the direction of arrows coincides with the direction of the local X_l axis of elements.

The filter setting (Fig. 10.2-36) enables to switch from the display of the output of forces to the display of the orthotropy axes.

Information on the Element

This mode enables to obtain information on the type of the finite element, number of sections of the output of forces, stiffness properties, applied loads, angles of orientation of the local axes, hinges and rigid inserts. The diagrams of the deflections and forces, tables with the values of the design combinations of forces and other results are output in the mode of the analysis of the results. The **Information on the Element** dialog box (Fig. 10.2-37) contains the controls of these operations.

In order to obtain the information click the filters button and specify the method for selecting the necessary element in the dialog box. In order to find an element you can specify its number in the **Element number** field and click the **Find** button or select an element with a cursor if the **Indication on the model** button was pressed beforehand.

The selected element is marked by a green ring, and the information on it is output in the dialog box where an element is displayed corresponding to its position in the structure, its nodes and their numbers, and also the directions of the local coordinate system. The node of the origin of the local coordinate system is marked by a yellow rectangle. The image can be rotated with the help of the mouse. Double left-clicking restores the image to its initial position.

Depending on the type of the element, specified data and the performed calculations the dialog box may contain a certain set of buttons which enable to view the respective information, and to change it in the mode of the data preparation. This dialog box provides the access to the following main data.

The **Nodes** button invokes the dialog box (Fig. 10.2-38) which contains a table with the list of nodes of the element. You can change the numbers of nodes when preparing the data. If an element with additional nodes on the sides is considered, node number 0 in the part of the table listing the additional nodes means that there is no additional node. You can attach/detach additional nodes by changing 0 to the number of the existing node and vice verse.

Type of element — type of the finite element is specified here, **Number of the rigidity type** — number of the rigidity type, **Number of sections** — the number of intermediate sections where the forces are calculated is output for bar elements.

Clicking the **Loads** button invokes the **Loads on Element** dialog box (Fig. 10.2-39) with the information on loads applied to the element in each loading. This mode enables to selectively delete the loads from the current (active) loading. In order to delete it select a line with the load you want to delete in the list which is specified as **Current** (loading), and click the **Delete** button.

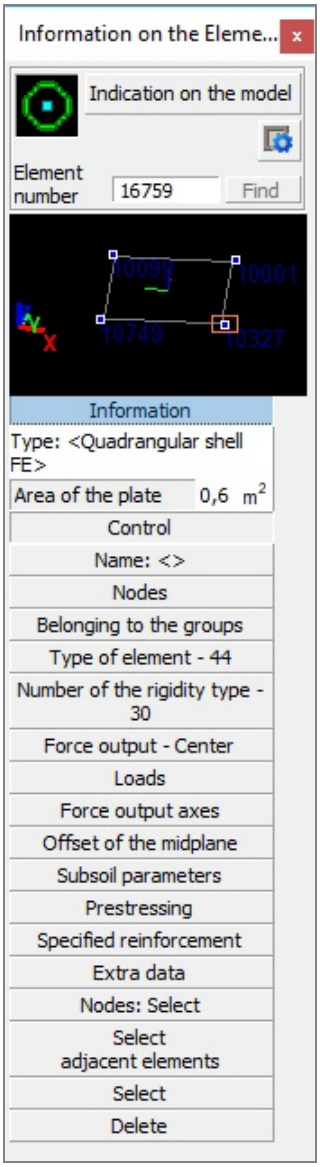


Figure 10.2-37. *The* **Information on the Element** *dialog box*

The value of the load selected in the list is output in the respective information window. In order to correct the load value, select it in the list, change the value in the text field and click the **Modify** button. The load will be deleted or changed only in the case if the loading is saved using the button in the **Loadings** section.

The **Number of the rigidity type** button invokes the dialog box with the stiffness properties of the element which can be corrected. The main thing you should remember when using this operation is that only the characteristics of the analyzed element will be changed which corresponds to creating a new rigidity type.

If it is necessary to change the characteristics of a certain type of rigidity, this operation has to be performed by invoking the respective operation from the **Assign** section or using the color indication of the numbers of the rigidity types (Fig. 10.2-28).

The **Orientation of the Axes of Inertia** dialog box can be invoked for two-node elements by clicking the **Local axes** button, the **Hinges** button invokes the **Boundary Conditions of Bars** dialog box, and the **Rigid inserts** button invokes the **Rigid Inserts** dialog box where you can correct the parameters specified earlier. When preparing the data you can change the **Subsoil parameters** and **Prestressing** for the selected element with the help of the respective buttons. If necessary the considered element can be deleted or marked as selected. The **Delete** and **Select** buttons respectively are used for performing these operations.

Clicking the **Select adjacent elements** button makes the current element and all the elements having common nodes with the current selected.

It should be noted that the modification of the characteristics of elements is available only in the preprocessor. Using the above functions in the mode of the analysis of the results is purely informative.

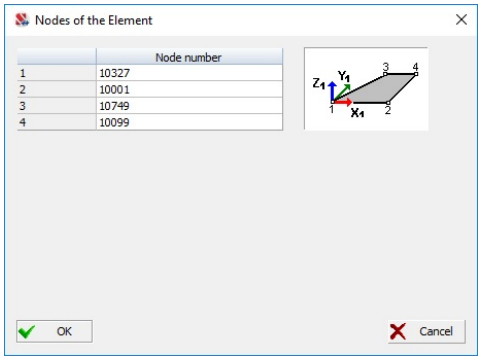


Figure 10.2-38. *The* **Nodes of the Element** *dialog box*

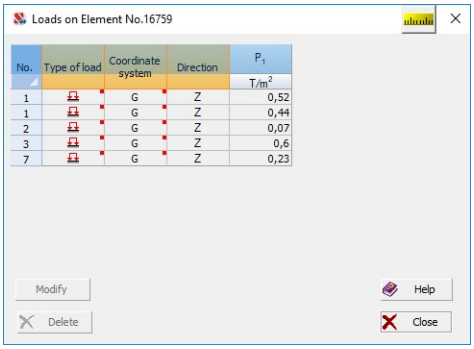


Figure 10.2-39. *The* **Loads on Element** *dialog box*

Diagrams of deflections (in the local coordinate system) or forces can be plotted in the mode of the analysis of the results of the calculation for bar elements. Dialog boxes with diagrams (Fig. 10.2-40 and 10.2-41) can be invoked by clicking the respective buttons. The obtained diagrams can be output in the report document by clicking the **Report** button, or tables with the values of factors on the basis of which the diagrams were plotted can be loaded into Excel®.

If the direct integration of the equations of motion is analyzed (Chapter 15), you can view the graphs of the variation of the components of forces (stresses) with time.

The **Stresses in sections** button enables to analyze the distribution of normal, shear or equivalent (according to the Mises theory) stresses in sections of bar elements for the current loading (or the combination of loadings) (Fig. 10.2-44). The **Stresses** button invokes the dialog box with the table of stresses for plate and solid elements (Fig. 10.2-42). Similarly, the **Principal and equivalent stresses** button enables to obtain a table with the respective information (Fig. 10.2-43).

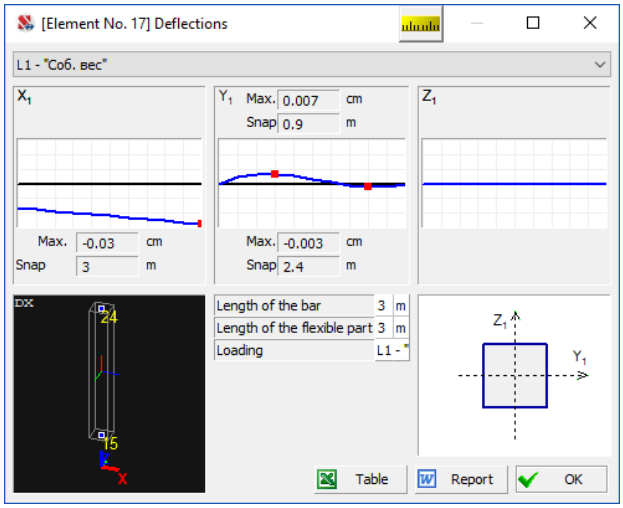


Figure 10.2-40. *Displacement (deflection) diagrams*

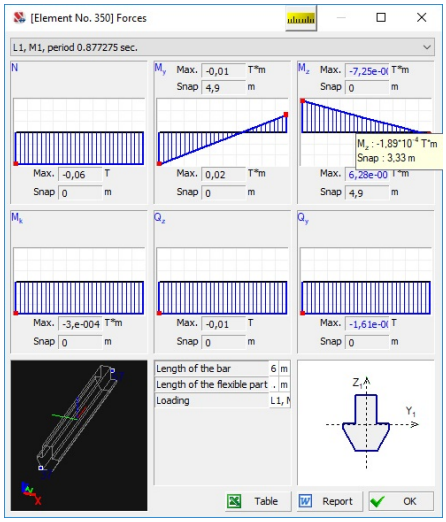


Figure 10.2-41. *Force diagrams*

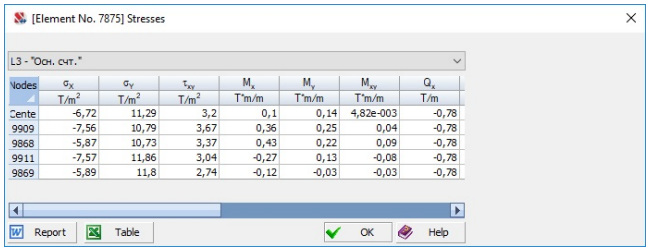


Figure 10.2-42. *The* **Stresses** *dialog box*

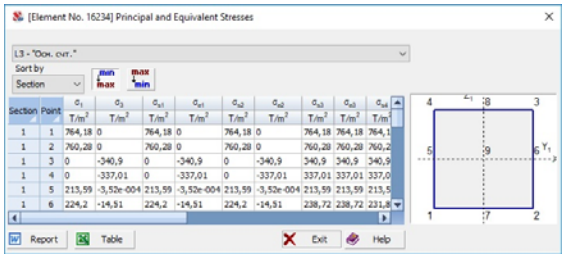


Figure 10.2-43. *The* **Principal and Equivalent Stresses** *dialog box*

The **Belonging to the groups** button invokes the dialog box (Fig. 10.2-45) which provides a list of groups containing this element. The **Show all groups** checkbox enables to expand the list and obtain a full set of groups. Each line of this list has a checkbox indicating whether an element belongs to a group. You can add the selected element to the group or delete it from it by checking or unchecking the checkboxes. The *Type of group* column shows whether a group is general or it is a group of structural members (steel or reinforced concrete). The **View** button enables to view the groups of structural members.

If the generation of design combinations of forces has been performed, you can use the **DCF** button to obtain the information on the design combinations of forces in the selected element in the tabular form in the special dialog box (see Sec. 18.12). A similar dialog box (invoked by clicking the **DC of deflections** button) provides information on the design combinations of deflections in bar elements.

When performing the calculation for the progressive collapse the **DCF (progressive collapse)** button becomes available and the tables will include data corresponding to the rules of this calculation (see Chapter 23). Moreover, in this case you can obtain the information on the utilization factors of restrictions for the considered element by clicking the **Progressive collapse** button.

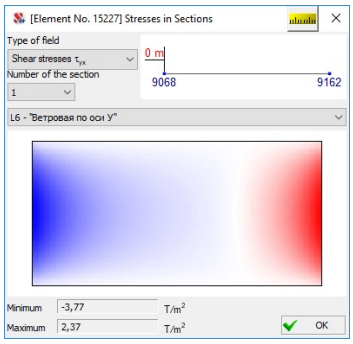


Figure 10.2-44. *The* **Stresses in Sections** *dialog box*

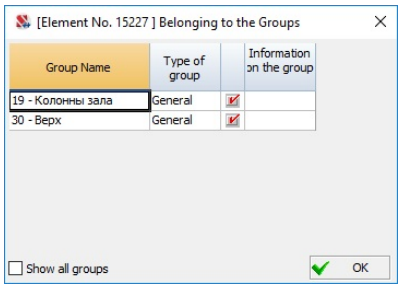


Figure 10.2-45. *The* **Belonging to the Groups** *dialog box*

If the element belongs to a group of structural steel or reinforced concrete members, the buttons which enable to obtain information on the results of the selection of reinforcement or check according to the design codes for reinforced concrete or steel structures will appear for this element. It is described in detail in Chapters 19, 20.

The **Information on the Element** dialog box can occupy any position in the work area or outside it. You do not have to close this dialog box when performing the operations of specification or correcting the geometry of the model.

In those cases when the set of buttons or information provided by default in the **Information on the Element** dialog box includes items which are not used or rarely used, it is possible to configure the dialog box to display only the necessary data. To do this, use the **Settings** operation, which is invoked by clicking the respective button in the considered dialog box or by right-clicking. After invoking this operation, a menu appears, which enables to include or exclude these items from the dialog box. The settings will be used later in all dialog boxes and for all models. If the user has changed the setting, the button **Settings** will have an additional indicator showing that the standard settings have been changed.

Displaying Loads, Masses and Specified Displacements

This section is devoted to the consideration of the issue of displaying loads in **SCAD**.

Display Nodal Loads

The rules for displaying loads of this type are specified in the **Nodal Loads** tab (Fig. 10.2-46). Controls of this tab enable to select a load by its type (forces, moments) and direction of the action for the current loading. If the lines with the load values are selected in the **Load value** list, only the loads the values of which coincide with the selected values will be displayed in the model (if they are specified in the current, i.e. active loading).

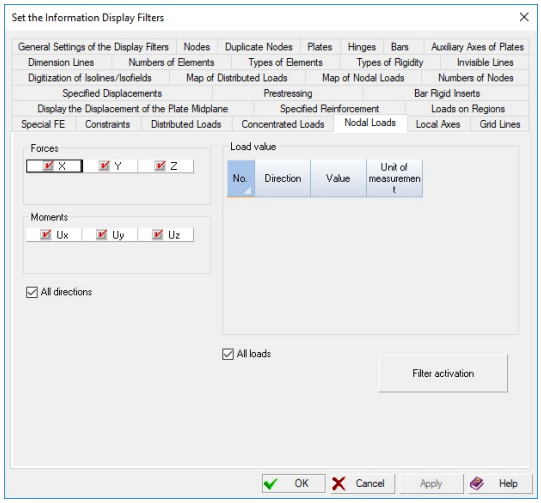


Figure 10.2-46. *The* **Nodal Loads** *tab*

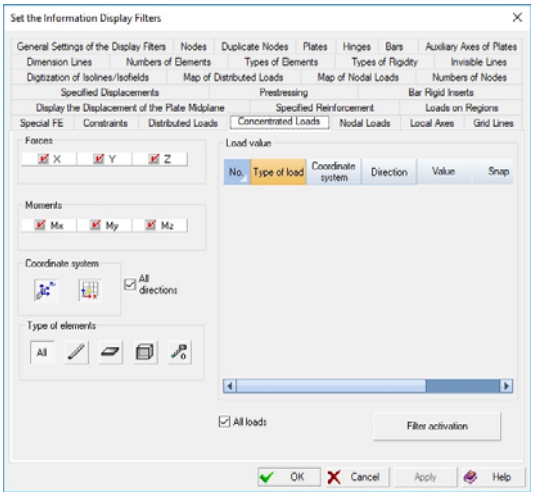


Figure 10.2-47. *The* **Concentrated Loads** *tab*

Nodal loads are displayed in the following way:

- forces — as an arrow along the respective axis;
- moments — as a double arrow along the axis about which the moment acts.

In both cases the direction of the arrows is defined by the sign of the force or moment.

By default the forces are displayed as red arrows, and moments — as green arrows.

It should be noted that a force is positive if it is directed against the axis, and a moment is positive if its vector determined according to the right-hand screw rule acts against the axis direction.

Display Local Concentrated Loads

The rules for displaying loads of this type are specified in the **Concentrated Loads** tab (Fig. 10.2-47). Controls of this tab enable to select loads for displaying in a loading by their type (forces, moments), direction of action, type of elements, and the coordinate system they are applied in (local, global).

Moreover, the tab also contains a list of load values similar to the **Nodal Loads** tab. Concentrated loads are displayed in the following way:

- forces — as an arrow along the respective axis;
- moments — as a double arrow along the axis about which the moment acts.

The sign convention coincides with that described for nodal loads but taking into account the coordinate system (local or global).

Display Local Distributed Loads

The **Distributed Loads** tab (Fig. 10.2-48) is used for specifying the rules for displaying loads of this type. Controls of this tab enable to select loads by their type (forces, moments), direction of action, type of elements, and the coordinate system they are applied in (local, global). Moreover, the tab also contains a list of load values similar to the **Nodal Loads** tab.

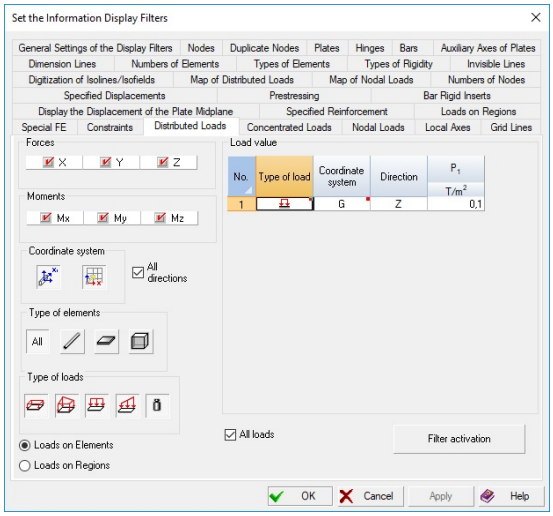


Figure 10.2-48. *The* **Distributed Loads** *tab*

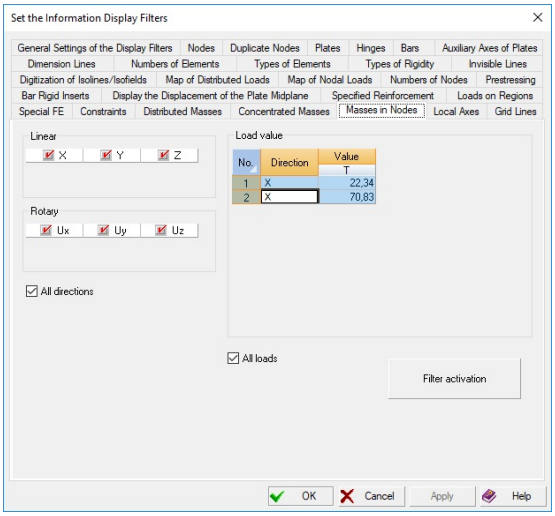


Figure 10.2-49. *The* **Masses in Nodes** *tab*

The **Distributed Loads** tab enables to switch to a mode where you can set display filters for the area loads by selecting a type, direction or a certain set of loads.

The **uniformly distributed** loads on bar elements are displayed in the following way:

- forces — as a bound pair of arrows applied to the nodes of elements along the respective axis;
- moments — as a bound pair of double arrows applied to the nodes of the element along the respective axis about which the moment acts.

The loads on plate elements are displayed in the following way:

- forces — as three or four bound arrows (depending on the number of nodes in the element), applied to the nodes of elements along the respective axis;
- moments — as three or four bound double arrows applied to the nodes of the element along the respective axis about which the moment acts.

The uniformly distributed loads on solid elements are displayed as an arrow applied to the center of the element and directed along the respective axis.

Trapezoidal loads on bar elements are displayed in the following way:

- forces — as a bound pair of arrows applied to the element along the respective axis;
- moments — as a bound pair of double arrows applied to the element along the respective axis about which the moment acts.

The following system of designations is assumed for plate elements:

- forces — as three or four bound arrows (depending on the number of nodes in the element), applied to the nodes of elements along the respective axis;
- moments— as three or four bound double arrows applied to the nodes of the element along the respective axis about which the moment acts.

By default the forces are displayed as red arrows, and moments — as double green arrows. The sign convention coincides with that described for nodal loads but taking into account the coordinate system (local or global) in which the loads are applied.

Display Thermal Loads

Thermal loads are displayed as wavy arrows applied to the centers of the elements for which this type of load is specified.

The setting of display of this type of load is not provided.

Display Masses and Dynamic Actions

If the active loading is a dynamic one, the icons for displaying nodal, concentrated and distributed loads change, and the user can obtain a graphical display of masses.

Tabs similar to those for setting the display of loads are used for the control of the display of masses and dynamic actions (see Fig. 10.2-49). The controls of the output are in many respects similar to those described above for nodal loads.

Masses are displayed in the model as lines without arrows (they do not have a sign) along the respective axis.

Display Specified Displacements

This filter is used for displaying loads specified as the displacement of the constraint (calculation for specified displacements). Similarly to nodal loads a setting which enables to select the directions of action or individual values that have to be visualized is provided for the loads of this type. The value of the specified displacement can be output on the screen using the filters button **Values of loads** — .

Generate a Map of Distributed Loads

The operation of generating a map of distributed loads is used for the control of total uniformly distributed loads specified in the current loading. In the result of performing this operation elements of the design model are displayed in different colors depending on the value of the total load.

The setting is performed in the **Map of Distributed Loads** tab (Fig. 10.2-50), which is invoked when setting the filter and has the following parameters:

- **Type of elements** — bars or plates;

- **Coordinate system** for specifying the loads — local or global;
- **Direction** of the load action — X, Y or Z;
- **Range of loads** — all loads or the loads in the given range (in the last case it is necessary to specify the range of the load values).

The values of the total loads can be output in the centers of elements with the help of the filters button **Values of loads** — .

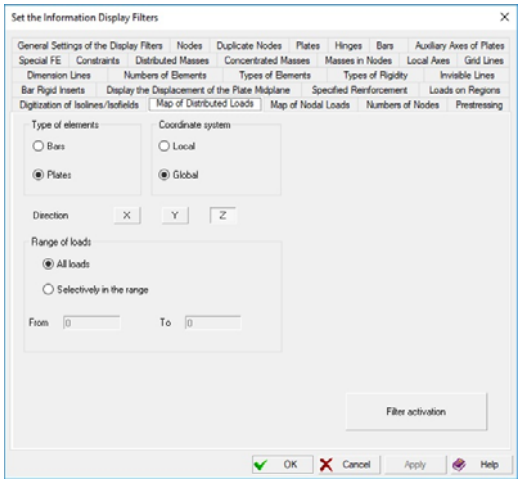


Figure 10.2-50. *The* **Map of Distributed Loads** *tab*

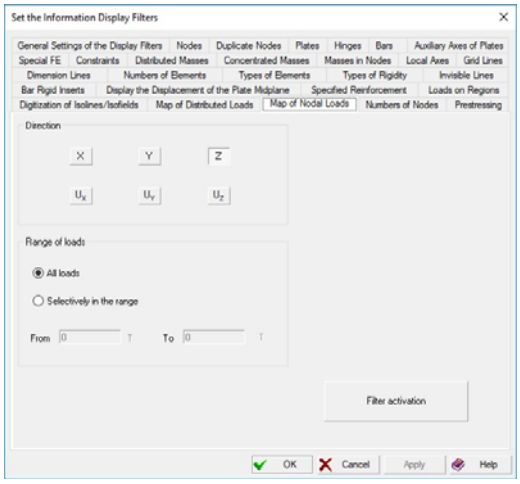


Figure 10.2-51. *The* **Map of Nodal Loads** *tab*

Map of Nodal Loads

The operation of generating a map of nodal loads is used for the control of the total nodal loads specified in the current loading. In the result of performing this operation the nodes of the design model are indicated by color markers depending on the value of the total load.

The setting is performed in the **Map of Nodal Loads** tab (Fig. 10.2-51), which is invoked when setting the filter and has the following parameters:

- **Direction** of the load action — X, Y, Z, U_X, U_Y, U_Z;
- **Range of loads** — all loads or the loads in the given range (in the last case it is necessary to specify the range of the load values).

The values of the total loads can be output with the help of the filters button **Values of loads** — .

Indication of the Number of Loads

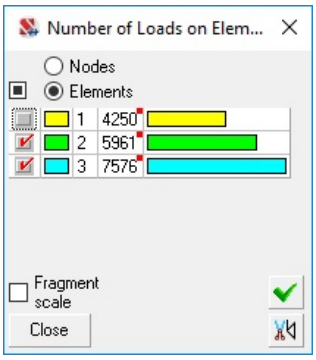


Figure 10.2-52. *Color scale of the* **Indication of the number of local loads**

This filter is used for displaying a color map of the number of loads on the element/node. Each element/node of the model or fragment will be displayed in a certain color dependingon the number of loads applied to it in the given loading. A standard color scale enables to select the type of the objects (nodes or elements) and control the display including the fragmentation of elements depending on the number of loads (Fig. 10.2-52) .

Output the Values of Loads and Masses

The values of loads are output according to the following rules:

- for nodal and concentrated loads — next to the arrow displaying the load;
- for uniformly distributed loads on bars — in the center of an area bounded by a pair of arrows;
- for uniformly distributed loads on plates and solid elements — in the center of an element;
- for trapezoidal loads — values of the load next to the arrows;
- for masses — similarly to nodal loads (letter M stands before the value of the mass);
- for dynamic actions (pulse, impact, harmonics) prefixes IM, SM, HM respectively are used when outputting loads.

The type, color and size of the font for indicating the load value can be changed in the **Font Setting** tab of the **Parameters** dialog box which can be invoked from the ***Settings*** menu.

When outputting the loads their image is scaled in accordance with the value of the load. The scaling is performed separately for forces and moments.

Display Axes of the Global Coordinate System

When working with the model the global coordinate system is always displayed in the lower left corner of the window and shows the current position (rotation) of the design model. After activating the filter the coordinate system is moved into the point with the coordinates 0, 0, 0.

Setting of this filter is not provided.

Display Grid Lines

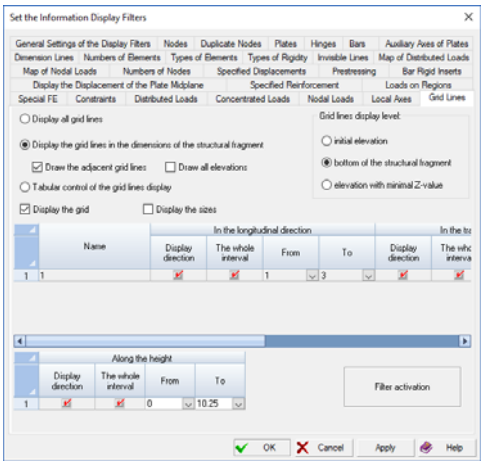


Figure 10.2-53. *The* **Grid Lines** *tab*

This operation enables to display the specified grid lines and elevations in the model. The **Grid Lines** tab of the **Set the Information Display Filters** dialog box contains the controls for displaying the grid lines (Fig. 10.2-53). The controls of this tab enable to perform the following actions:

- set the mode for displaying all or individual groups of grid lines and marks, fully or only in the specified interval for each direction;
- select the rules for displaying marks of grid lines;
- specify the display of grid lines only in the dimensions of the structural fragment;
- enable the **Display the grid** mode;
- display the dimension lines.

In order to control the selective display of grid lines for each direction select the **Tabular control of the grid lines display** radio button and perform the necessary settings in the tables.

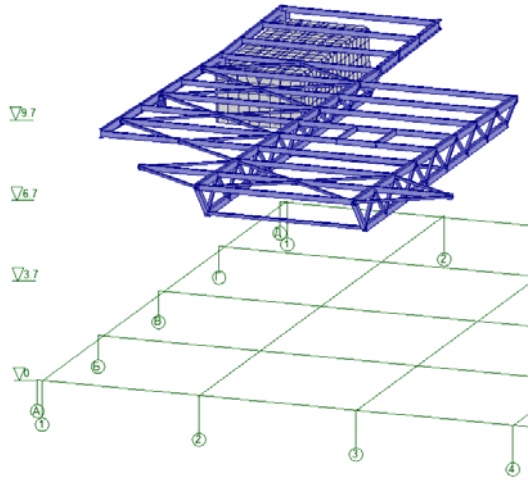
The checkboxes in the **Display direction** column enable to display or hide the grid lines in longitudinal or transverse directions, and marks as well. If **The whole interval** checkbox is not checked, you can specify the marks of grid lines which define the interval displayed on the screen. In this case you will obtain access to the **From** and **To** lists. A mark of the first grid line of the interval is selected in the first list, and that of the last one — in the second list. This selective display can be specified for each group of grid lines.

Grid lines can be displayed together with the dimension lines. In order to do this, check the **Display the sizes** checkbox.

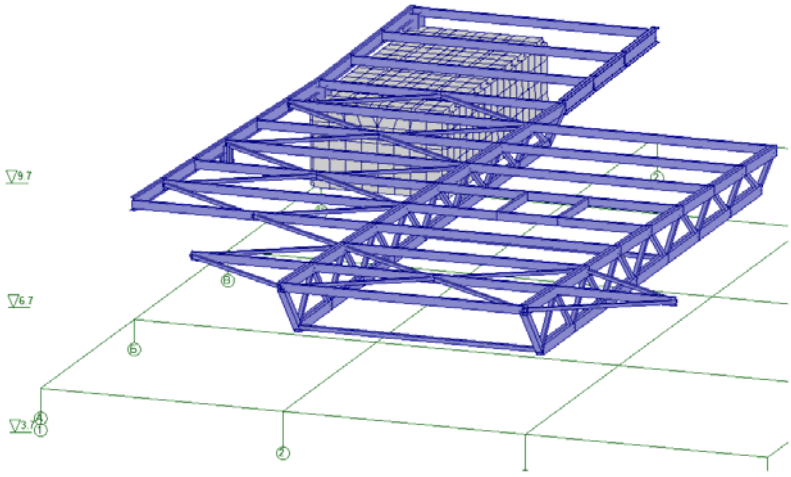
By default, the elevations are drawn on a vertical line that intersects the horizontal plane at the origin of the system of grid lines selected by the user. If this system is not displayed, the elevations are displayed at the bottom left corner of the model.

The horizontal plane for drawing the grid lines can be selected using one of the following options:

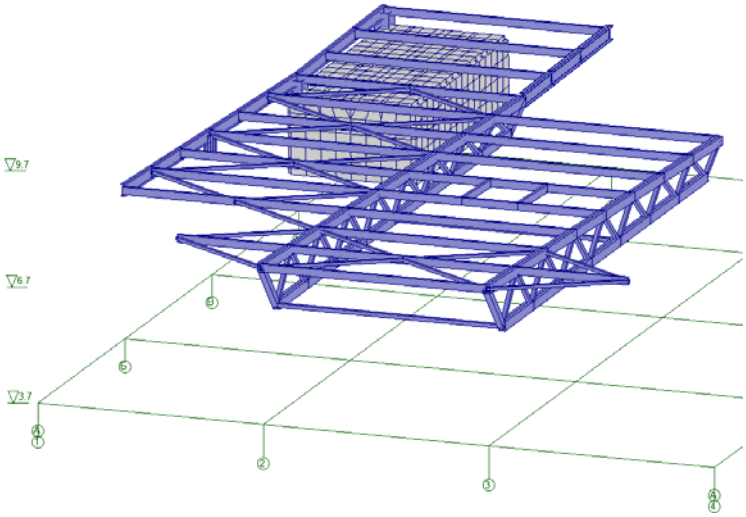
At the zero elevation



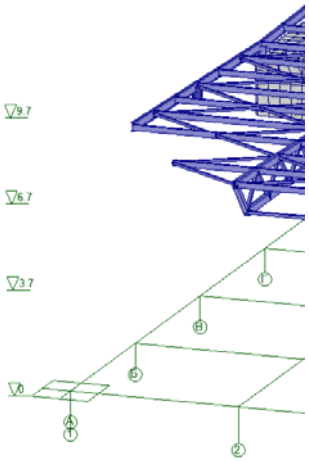
At the bottom of the model or of its fragment



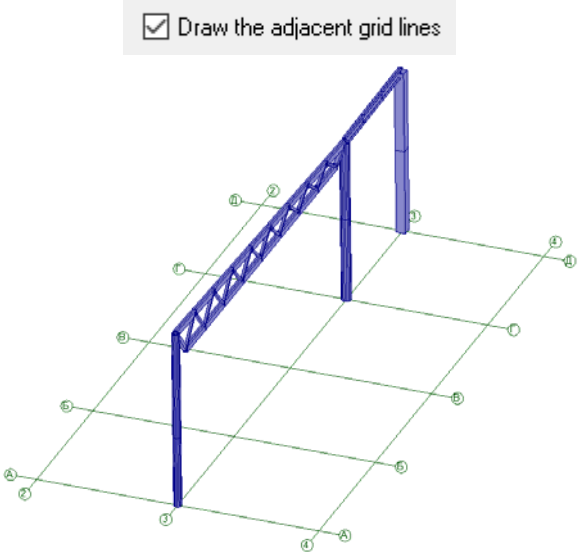
At the elevation with the minimum Z coordinate

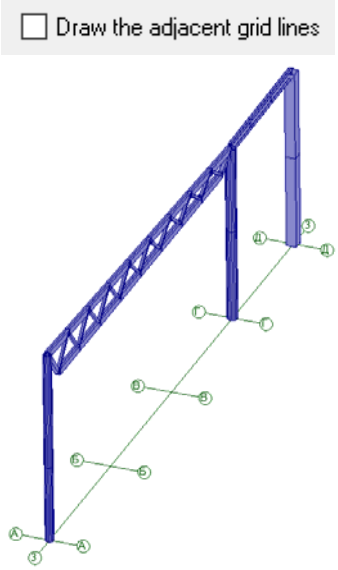


If the grid lines are displayed in the dimensions of the structural fragment, the **Draw all elevations** checkbox enables to see the elevation of the selected fragment more clearly.



If the **Display the grid lines in the dimensions of the structural fragment** radio button is selected, all grid lines intersecting the fragment are displayed by default together with the adjacent ones, which allows to uniquely determine the position of the fragment. An additional option is provided to control the display of grid lines for the most common case, when only one system of rectangular grid lines is specified (or the display of all other grid lines is disabled using the table): you can uncheck the **Draw the adjacent grid lines** checkbox and only the grid lines intersecting the fragment will be displayed.





Display Dimension Lines

This operation enables to display the dimension lines on the screen. They are set along the bottom and left edges of the work area. Filter settings are performed in the **Dimension Lines** tab (Fig. 10.2-54) and enable to display the grid. Dimension lines and the grid are output only in the case if the fragment or the whole model are projected onto one of the planes of the global coordinate system (Fig. 10.2-55). The spacing of dimension lines can be selected automatically or specified by the user.

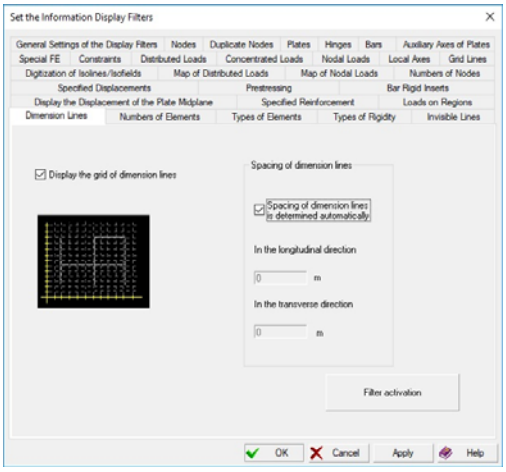


Figure 10.2-54. *The* **Dimension Lines** *tab*

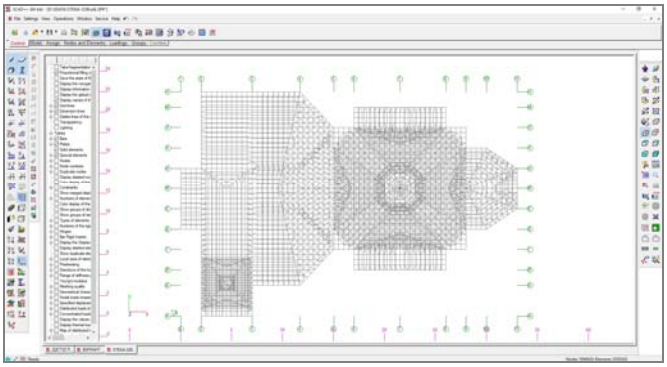


Figure 10.2-55. *Display of the dimension lines and the grid*

Deleting the Lines of the Invisible Contour

The following modes of display of the design model can be set in the **Invisible Lines** tab of the **Set the Information Display Filters** dialog box (Fig. 10.2-56):

- deleting the lines of the invisible contour or the contour display;
- the same with the three-dimensional display of the profile of bar elements;
- the same with the display of the thickness of plates.

  You should take into account the following facts when invoking this operation:
- when the GDI rendering engine is used the time needed for deleting the lines of the invisible contour depends on the sizes of the design model and can be quite considerable for large systems. If the options for displaying the profiles of bar elements and thicknesses of plates are active, the time for deleting increases considerably;
- the **Environment Settings** dialog box enables to specify the gap between the bar elements (the **Offsets for displaying the profiles** group of buttons in Fig. 10.4-1). The gap is necessary to correctly display the intersection of elements at the three-dimensional display of profiles;
- the gap defines the distance from the end of the bar to the node in fractions of the sizes of the section; if a zero gap is specified, the intersection of bars coming to one node is unpredictable and the time necessary to delete it can be increased in several times;
- the profiles of short bars the length of which is commensurable with the sizes of their cross-section are not output in the GDI rendering engine.

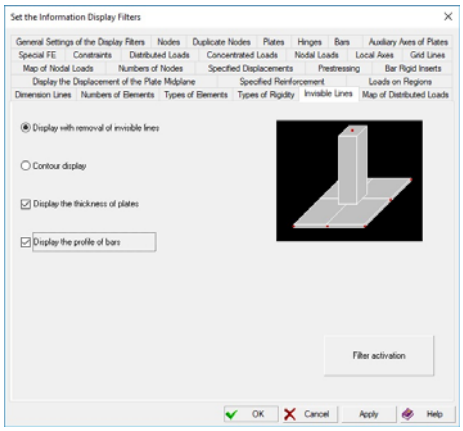
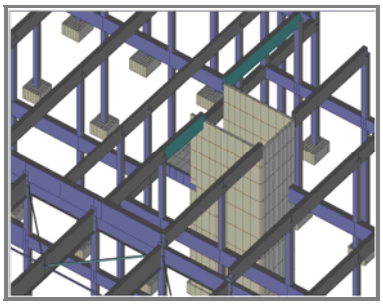
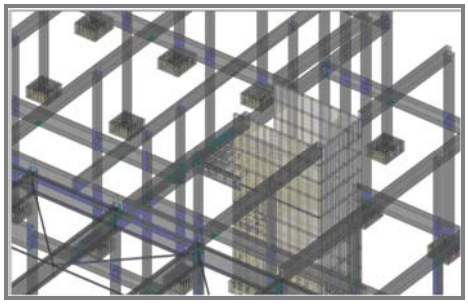


Figure 10.2-56. *The* **Invisible Lines** *tab*

a)



b)



c)

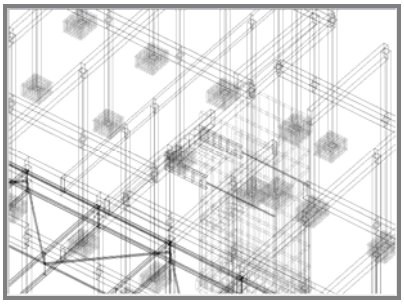


Figure 10.2-57. *The display of profiles of bar elements and plates*
a) with the enabled **Display with removal of invisible lines** *option*;
b) with the enabled **Transparency** *filter; c) with the enabled* **Contour display** *option*

Transparency, Lightning, Perspective

These three buttons of the **Display Filters** toolbar enable to obtain an image with the effects of transparency, lighting and perspective respectively. All three options are available only in the case if the display is performed in the DirectX rendering engine.

Color Display of the Groups of Nodes and Elements

These filters invoke the respective color scale (Fig. 10.2-58), where each group of nodes or elements has a corresponding color and a line with the name of the group. Thus, nodes and elements of the model belonging to these groups are highlighted in a certain color. The following operations are available when working with color scales:

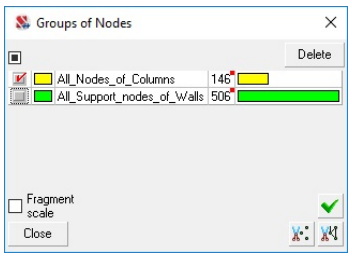
- selecting **active** groups into a fragment (groups of nodes can be included in a fragment without elements or with adjacent elements);
- deleting active groups.

Groups with checked checkboxes are considered to be active. Any operations of correction, specification and visualization can be performed with nodes and elements of groups included in a fragment. These groups can be saved as individual models (button in the **Control** section).

In order to obtain a new fragment consisting of other groups it is necessary to check the respective checkboxes and click the **Fragment** button.

You can return to the display of the full model by performing the fragmentation at the active state of all groups in the color scale or by clicking the **Initial display of the model** button of the visualization filters.

a)



b)

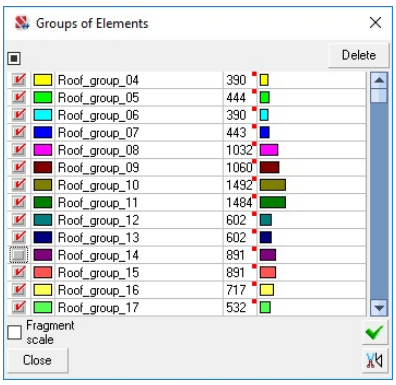


Figure 10.2-58. *Color scales*
a) **Groups of Nodes**; *b)* **Groups of Elements**

It should be noted that the color display of groups will not work simultaneously with other rendering engines where elements/nodes are displayed in color.

When the color indication of the groups elements is performed in the postprocessor, the button enables to digitize elements with the numbers of the groups these elements belong to.

Color Display of the Specified Reinforcement

The filter is used for the color indication of elements with different types of the specified reinforcement. The color scale (Fig. 10.2-59) contains switches , , which enable to obtain color scales for plates or bars respectively.

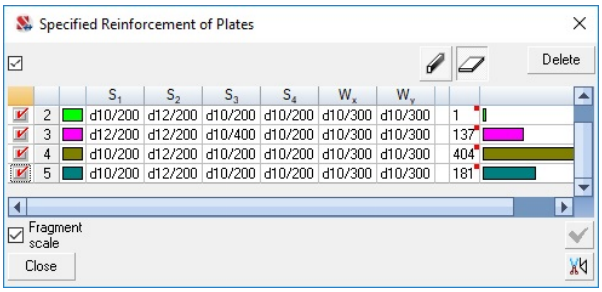


Figure 10.2-59. *The* **Specified Reinforcement of Plates** *color scale*

Display Groups of Structural Members

These filters are used for the color indication of groups of structural members for the check and selection of rolled steel sections — or groups of reinforced concrete elements — . The color scale

(Fig. 10.2-60) includes names of groups, checkboxes, and **Apply** and **Fragment** buttons which perform the same functions as the similar controls of the color indication of the groups of elements.

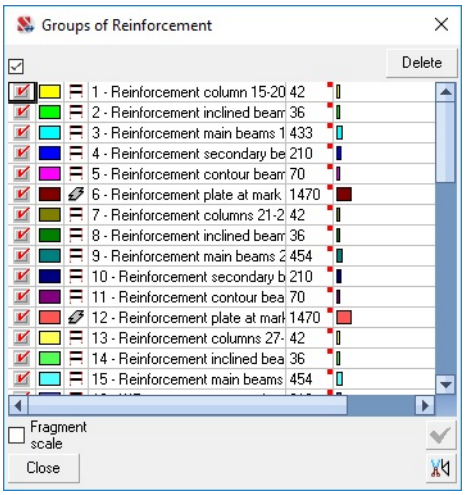


Figure 10.2-60. *The* **Groups of Reinforcement** *color scale*

There are the following icons next to the color markers in structural groups of elements of reinforced concrete structures:

— group of plate structural members;

— group of bar structural members;

— bar structural member.

The following icons are used for structural groups of elements of steel structures:

— group of structural members;

— structural member.

If a structural group is specified as an additional one, the above icons are displayed on a green background. The column marked with a red square indicates the number of elements included in the groups, and the next column provides this information in the form of a diagram.

When the color indication of the groups of structural members is performed in the postprocessor, the button enables to digitize elements with the numbers of the groups these elements belong to.

Cancel the Selection of Nodes and Elements

Technically this operation does not belong to the display filters, but it can be used quite often when working with the model and therefore it is included in the **Display Filters** toolbar. All the selected or marked elements and nodes will be deselected after clicking this button.

Navigator

The navigator enables to determine the position of the visible part of the design model after performing the operations of fragmentation by a marquee and cutting on projections. A window with the image of the design model where its visible part is displayed in red appears after invoking the navigator.

Show Elements Included/Not Included in the Given Erection Stage

It enables to see the elements working in the erection stage (see Chapter 14). The **Erection Stages** tab of the **Set the Information Display Filters** dialog box enables to display the information on the coefficients of variation of the elastic modulus on the model or invoke the color display of erection stages (an element is assigned a color corresponding to the stage where it first appeared).

Prestressing

If the design model has elements with prestressing, this filter enables to obtain a color scale (Fig. 10.2-61) displaying the values of prestressing and/or values of prestressing assigned to the elements. Filter settings (Fig. 10.2-62) enable to select the types of elements (plates, bar/cable stays, or solid elements), specify the force factor you are interested in, and select the style of display: color and/or digitization.

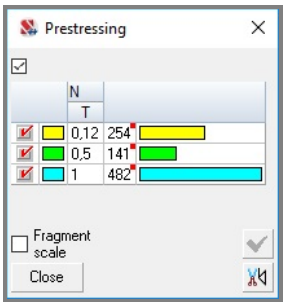


Figure 10.2-61. *The* **Prestressing** *color scale*

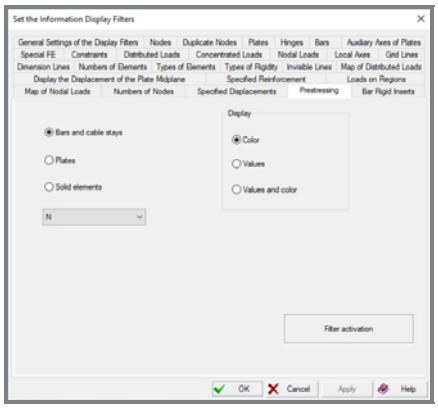


Figure 10.2-62. *The* **Prestressing** *tab*

Color Display of Modules

If you activate this option for the models of **FORUM** type, all elements belonging to a certain module will be displayed in the color assigned to this module.

Types of Rooms

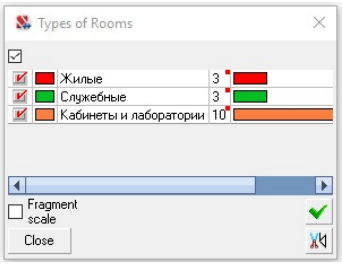


Figure 10.2-63. *The* **Types of Rooms** *color scale*

You can obtain a color scale (Fig. 10.2-63) and the coloring of floors into different colors depending on the type of room assigned to them for the models of the **FORUM** type for which the information on the types of rooms is specified.

Color Display of Fire Resistance Groups

This filter is used for the color indication of fire resistance groups. The color scale contains includes names of groups, checkboxes, and **Apply** and **Fragment** buttons which perform the same functions as the similar controls of the color indication of the groups of elements.

10.3. Displaying the Results of the Calculation

The means for the visualization of the results of the calculation (deformations, forces, stresses) are described in this section.

Visualization of Results Toolbar

The toolbar is available in the mode of the analysis of the results of the calculation and includes the controls for the setting of the image when outputting isofields and isolines, displaying the values of the considered factor on the model, and the visualization of deflections of bar elements (Fig. 10.3-1).

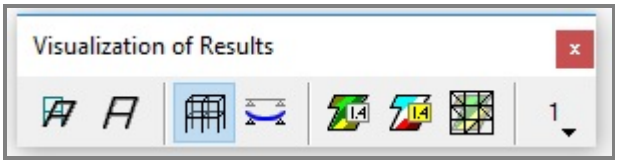


Figure 10.3-1. *The* **Visualization of Results** *toolbar*

Displaying the Deflections in Bars

This filter enables to display the deformed model taking into account the displacements in the intermediate points of the bar in the mode of the graphical analysis of the results of the calculation. The filter can be activated in the mode of the animation or generation of video clips (AVI-files).

It should be taken into account that if the mode of the display of deflections is activated, the time necessary to generate a deformed model slightly increases.

Finite Element Mesh on Isofields

This option should be activated if you have to display a finite element mesh simultaneously with isofields/isolines (see Fig. 10.3-2).

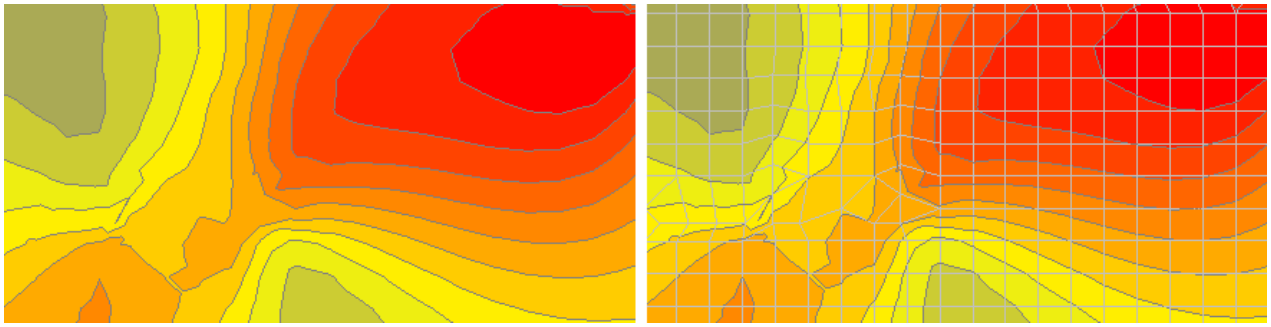


Figure 10.3-2. *Isofields without a finite element mesh and with a mesh*

Displaying the Deformed Model

Filter enables to obtain an image of the deformed model. If the filter is used, both deformed and initial models will appear on the screen. Displacement scale factor can be used when generating a deformed model. In order to do it, click the **Displacement scale factor** button and select the necessary scale factor in the menu that will appear.

Output the Values on Isolines, Isofields and Diagrams

This filter is available in the mode of the graphical analysis of the results and works only “above” the color display of the considered factor (isolines, isofields, etc.) or diagrams. It enables to display the values obtained in the result of the calculation in the centers of elements and/or nodes.

In order to display only the numerical values of the factor on the model, after outputting, for example, the isolines it is necessary to disable all the active positions in the respective color scale (using the **Off** button).

If the **Output the values of stress in the plate nodes and solids** checkbox is checked in the **Results** tab of the **Environment Settings** dialog box, the respective values will be displayed on the model together with the values in the centers of elements.

If some of the intervals are disabled in the dialog box of the color scale, the digitization (or its absence) on the disabled intervals depends on whether the **Do not digitize the disabled intervals of the color scale** checkbox is checked in the **General Settings of the Display Filters** tab of the **Set the Information Display Filters** dialog box (see Fig. 10.3-3).

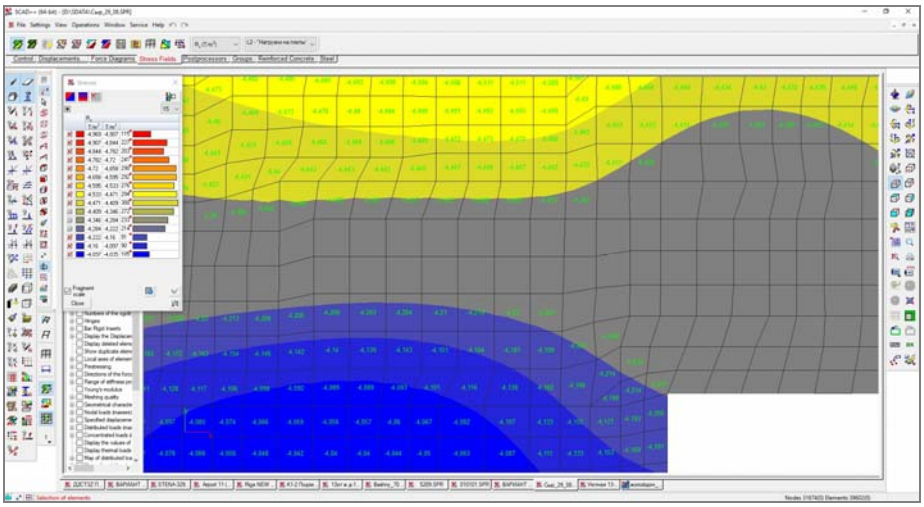


Figure 10.3-3. *Example of digitization of a part of isofields*

Selective Output of Values on Isolines, Isofields, and Diagrams

The numerical values of different factors can be displayed not for all but only for the selected nodes/elements (Fig. 10.3-4). Since the selection of nodes and elements usually deals with the performance of certain operations, a special selection is used for the selective digitization, which is enabled by the **Select objects for displaying information** marker button — . This button is located in the **Control the Selection** toolbar and is duplicated in the status bar.

The following procedure is used to perform the selection when outputting the results of the calculation:

- enable the filter **Display information on the model only for the selected elements and nodes** — ;
- enable the filter of the output of numerical values — ;
- enable the marker button — ;
- select the necessary objects in the model.

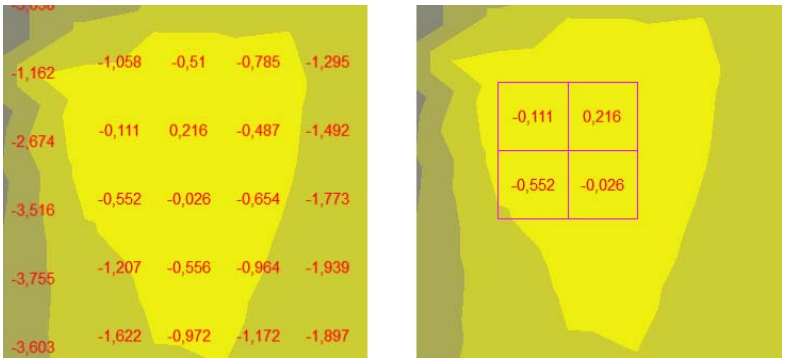


Figure 10.3-4. *Example of a complete and selective digitization*

It should be noted that by using the combination of buttons and in the postprocessor you can localize the output of other information on nodes and elements in the model as well, for example, the numbers of nodes or the names of the types of rigidity

Control of the Output of Isofields

The program provides two forms of representation of isofields — with a contrast transition between adjacent colors (Fig. 10.3-6 *a*) and a gradient one which provides a smooth transition between the colors of adjacent ranges (Fig. 10.3-6 *b*). In order to switch to the gradient drawing check the respective checkbox in the **Digitization** tab of the **Set the Information Display Filters** dialog box (Fig. 10.3-5).

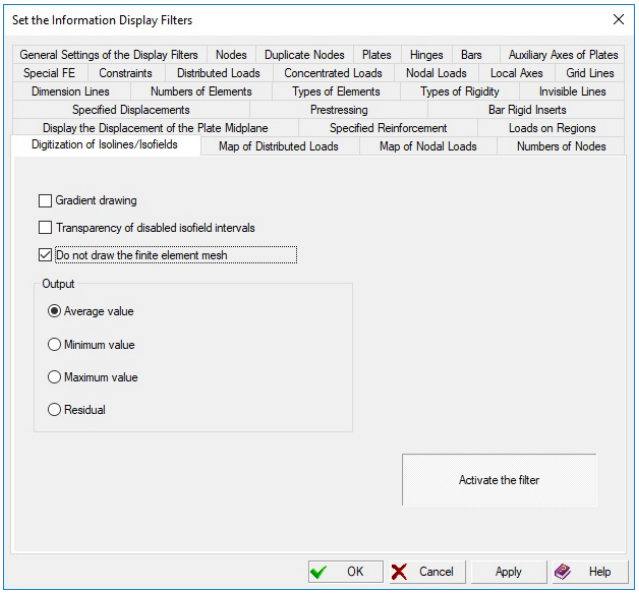


Figure 10.3-5. *The* **Digitization** *tab*

a) *b*)

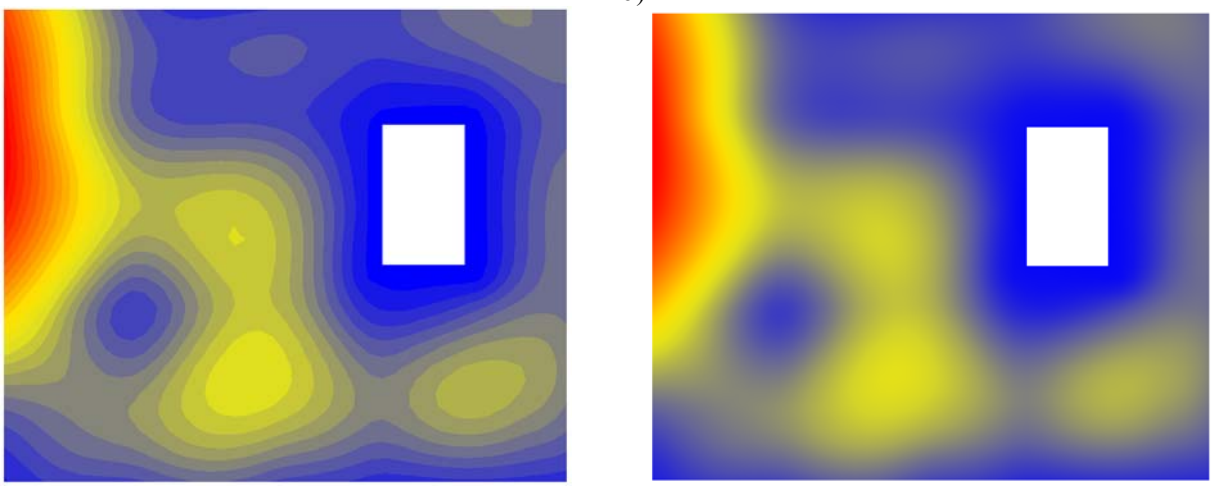


Figure 10.3-6. *Contrast a) and gradient b) forms of representation of isofields*

If some ranges of the color scale are disabled, areas of isofields with active ranges might be invisible in the spatial model (Fig. 10.3-7). You can "Open" the invisible areas by checking the *Transparency of disabled isofield intervals* checkbox (Fig. 10.3-8).

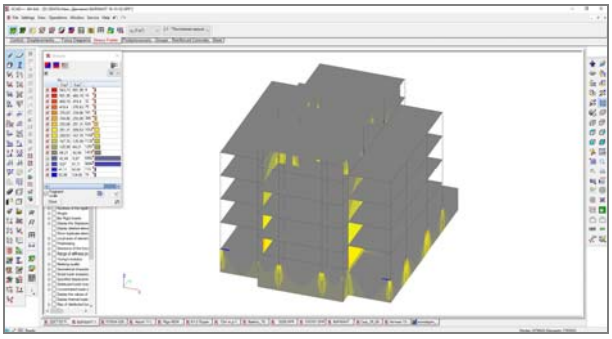


Figure 10.3-7. *Isofields of active ranges are "hidden"*

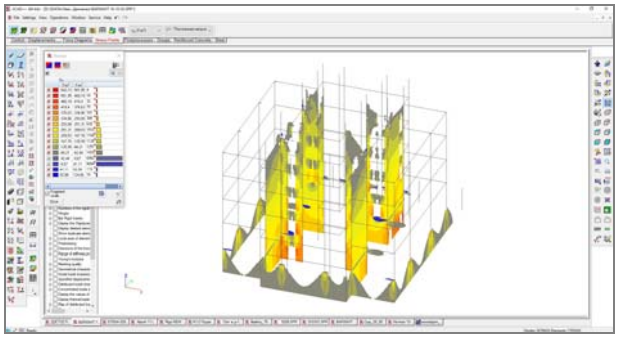


Figure 10.3-8. *Isofields of active ranges are visible*

10.4. Environment Settings

Environment settings can be specified by invoking a multi-tab dialog box from the **Settings** menu. This dialog box contains three tabs: **Design Model**, **Results** and **Animation Control**.

The **Design Model** tab (Fig. 10.4-1) contains the following controls of the display:

- an option of the separate display of plates, which enables to display the plate elements (Fig. 10.4-2) with an interval dividing the adjoining elements (this option enables to control the position of bar elements located between the plates);
- an option of "filling" the distributed loads on bar elements, which enables to visually separate the loads on bars and plates (Fig. 10.4-3);
- slider for selecting the displayed size of nodes;
- slider for selecting the size of color markers;
- slider for selecting the backlight level when the filters button **Enable/disable lighting** is used;
- slider for selecting the displayed size of hinges;
- checkbox which enables to save the results of work automatically and a line for specifying the time interval between the operations of saving;
- buttons for selecting the value of the offset from nodes at the three-dimensional display of bar elements which enables to avoid inaccuracies at the visualization in the case when several bar elements come into one node (if the sizes of the offsets exceed the length of the bar, the profile is not displayed);

- checkbox which enables to restore last state of the environment settings while next opening of the file with project;
- limitation on the number of lines in the log file of changes and on the number of possible undo operations.

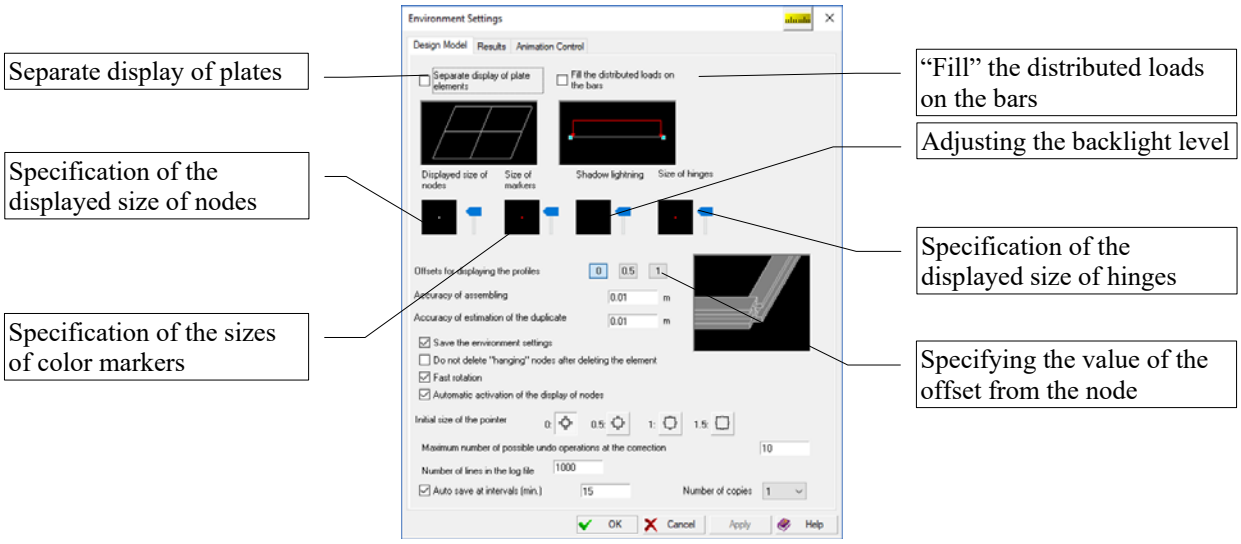


Figure 10.4-1. *The* **Design Model** *tab of the* **Environment Settings** *dialog box*

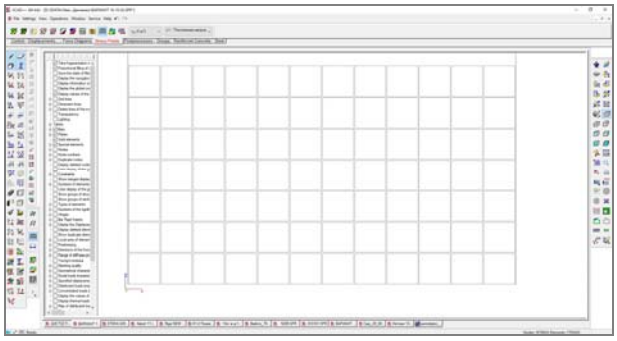


Figure 10.4-2. *Design model with a separate display of plates*

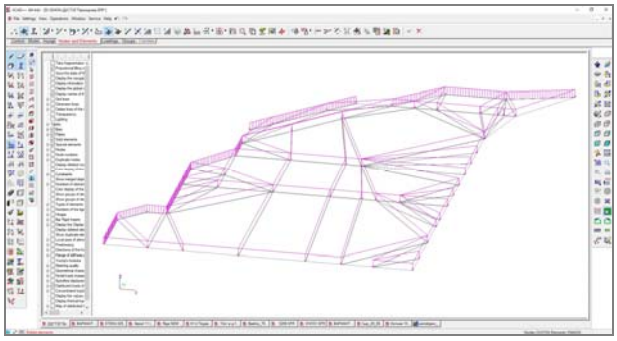


Figure 10.4-3. *Uniformly distributed load on bar and plate elements*

Moreover, this tab also enables to specify the values of the accuracy of estimation of the duplicate nodes necessary when performing the assembly of the model from submodels and other operations of synthesis of the design model including deleting of duplicate nodes, creating bar elements from a group of nodes lying on a straight line etc.

If the **Do not delete ""hanging"" nodes after deleting the element** checkbox is checked, deleting an element does not automatically delete nodes which are not adjacent to any other element.

The **Fast rotation** option invokes the mode of drawing the model at its rotation when only the "wire" design model is drawn. After the rotation the screen displays the model in a new position with all the information.

If the **Automatic activation of the display of nodes** checkbox is checked, the display filter of nodes will be enabled automatically (this filter is not disabled automatically) at the beginning of work of any operation where the selection of nodes is required (for example, deleting nodes, creating a bar, ...).

The auto-save operation is performed according to the following rules

- each model can have its time for autosaving;
- autosaving of the model is performed in the case if the model was modified during the time interval between the last saving and the current time;
- the user can define number of autosaving files;
- if the autosaving is performed the respective message appears in the log.

The **Results** tab (Fig. 10.4-4) contains the following controls of display:

- checkbox which enables to output the values of force factors in the nodes of plate elements;
- specification of the shell shallowness criterion (cosine of the angle between the normals to the adjacent areas of the design model), which enables to determine the relationship between the force factors of the adjacent areas of the design model (see below);
- the checkbox for hatching the diagrams and selection of the hatch spacing. If the spacing between the hatching lines (**Hatch spacing of the force diagrams**) is set as one, it corresponds to the maximum density of hatching. If the spacing values are equal to 1.5 or 2 the density of hatching decreases in 1,5 or 2 times respectively. An example of the display of diagrams on the design model at the checked **Hatch the force diagrams** checkbox is shown in Fig. 10.4-5;
- the option of saving the reactions in nodes from the modes of dynamic loadings;
- the **Two-color diagrams** of internal forces and moments checkbox. If this checkbox is checked, different colors will be used for the parts of diagrams with positive and negative values of the selected internal force or moment;
- When digitizing diagrams, a number corresponding to the maximum absolute value of the internal force or moment is displayed for each bar finite element. If the **Display minimum and maximum values** checkbox is checked, the minimum and maximum values of the internal force or moment separated by a colon will be displayed in the diagram for each element (if these values are different).
- If checkbox **Take into account the structural members** is checked, the force diagram digitization rules change for the previously created structural members (steel or reinforced concrete). Only the extreme values for the entire structural member are displayed, but not the minimum/maximum values for every finite element included in it. This rule does not apply to additional structural members.

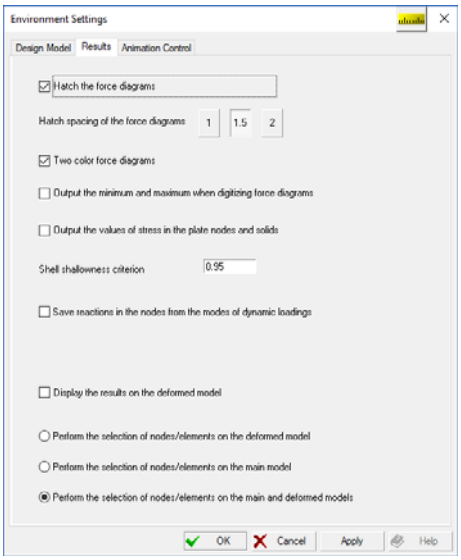


Figure 10.4-4. *The* **Results** *tab*

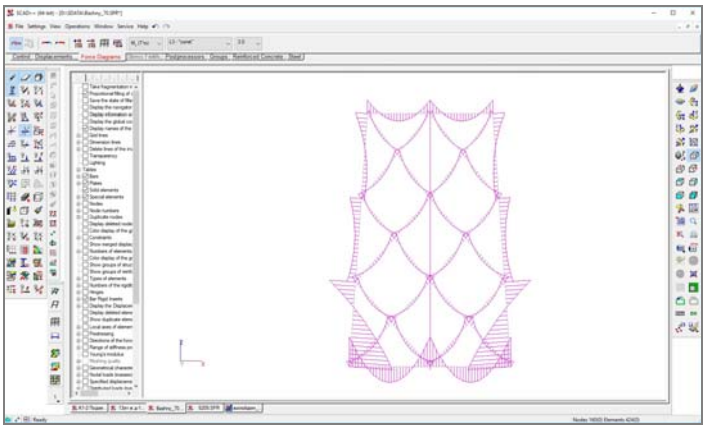


Figure 10.4-5. *Displaying diagrams on the design model*

The **Shell shallowness criterion** parameter enables to distinguish between two cases schematically shown in Figures 10.4-6, 10.4-7. In the first of them (Fig. 10.4-6) the dihedral angles between finite elements which formed after replacing a smooth surface of a shell by a faceted structure from flat finite elements are small. When determining the values of stresses in the nodes it is allowed to use the averaged data taken from all finite elements adjacent to the nodes (assuming that the directions of force output in these elements are specified correctly). In the second case (Fig. 10.4-7), when the angle of a building is considered, the dihedral angle between the walls is large. In this case, the stress fields should be generated separately for each wall. Then the averaging in nodes will be performed for each wall separately.

Thus, the **Shell shallowness criterion** defines the maximum value of the cosine of the angle between the normals to the surface of adjacent elements depending on which the rule for generating stress fields will be selected:

- if on the design model the angle between the normals to the surfaces of adjacent elements is less than the angle specified by the value of the parameter **Shell shallowness criterion**, the values of stresses in the vertex of an angle are averaged for all elements adjacent to the node (Fig. 10.4-6);
- if on the design model the angle between the normals to the surfaces of adjacent elements is greater than the angle specified by the value of the parameter **Shell shallowness criterion**, the values of stresses in the vertex of an angle are averaged for two elements, for example, as shown in Fig. 10.4-7.

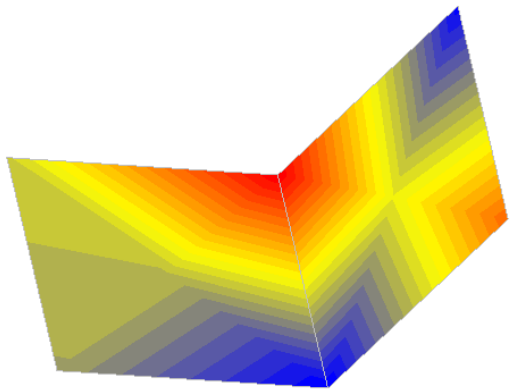


Figure 10.4-6. *Isofields on shallow shells*

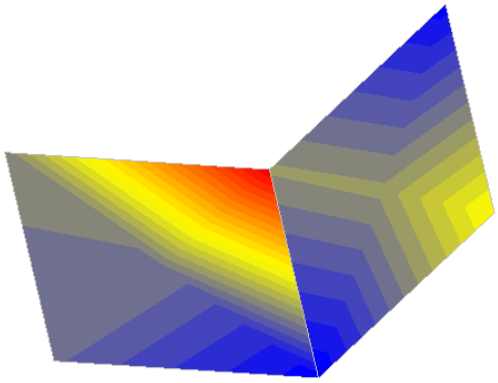


Figure 10.4-7. *Isofields on the corner line*

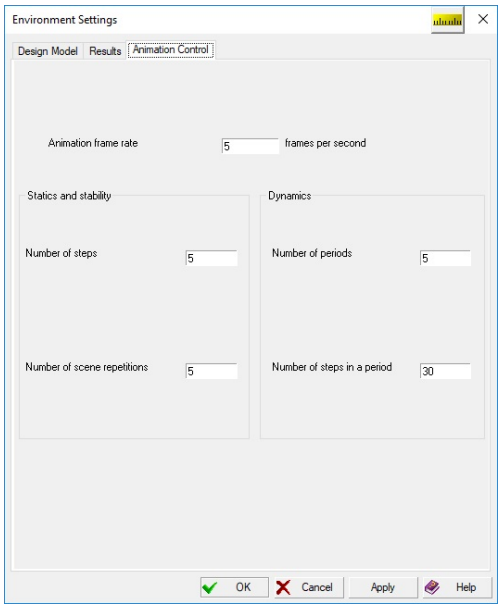


Figure 10.4-8. *The* **Animation Control** *tab*

By default, the software assumes the value of this parameter as 0.95, which roughly corresponds to the angle of 18°. If you specify the value of the parameter as 1, the averaging is not performed, and each element will have its "own" system of isofields.

The graphical display of the results (for example, isofields of stresses) can be performed on a deformed or on a non-deformed model. The respective checkbox in the **Results** tab enables to select the necessary variant.

When working with a postprocessor it is often necessary to select nodes/elements (for example, in order to create groups). A special group of buttons enables to specify the rules for working in a situation when there is a deformed model on the screen. The selection can be performed only on a non-deformed model, only on a deformed model or on both. If one of the models is not displayed on the screen, the selection mode will be automatically provided for the model displayed on the screen.

The **Animation Control** tab (Fig. 10.4-8) enables to specify the parameters of the animation control in the modes of the analysis of displacements and buckling modes for static and dynamic loadings. The number of steps (frames) in one scene of the "movie" and the number of scene repetitions are specified as

the parameters for static loadings and buckling modes. The number of periods and the number of steps in a period are specified for the modes of dynamic loadings.

The frame rate is a general parameter.

Like other parameters the environment settings are saved in the file of the project (for example, spr-file). If the dialog box for setting these parameters is invoked from the *Settings* menu when there are no open models, the saved values will be used for all newly created projects.

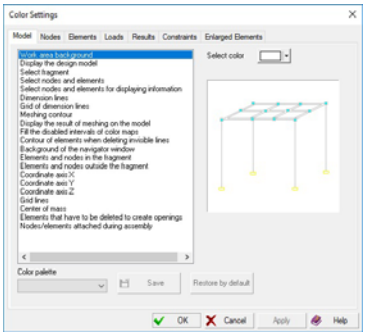
Color setting which are used for the newly created projects are saved in the following files:

- custom.color;
- presentation.color;
- soft.color;
- standard.color;
- white.color.

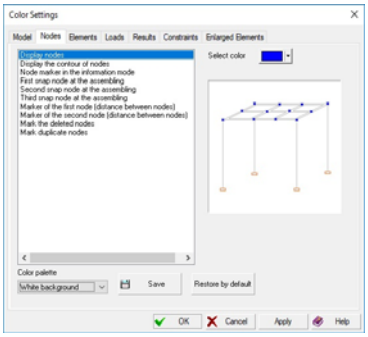
These files are in the following directory "C:\Users\All Users\SCAD Soft\Data\". If you save these files before installing a new version and copy them after the installation into the above directory, you will be able to use (for new projects) the colors set earlier.

10.5. Color Palette Settings

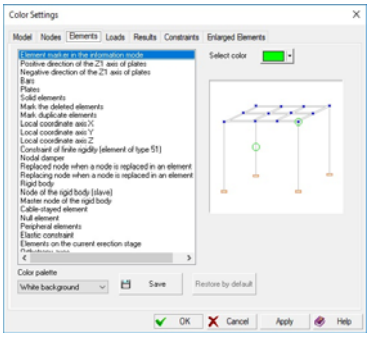
The operation of setting the colors of all elements of the information display on the design mode is provided for convenience of working with the software. The color palette setting is invoked from the ***Settings*** menu and it is available both in the mode for preparing the initial data, and when analyzing the results of the calculation. The settings are performed in the multi-tab **Color Settings** dialog box (Fig. 10.5-1, *a–g*), which includes the following tabs: **Model**, **Nodes**, **Elements**, **Loads**, **Results**, **Constraints**, **Enlarged Elements**.



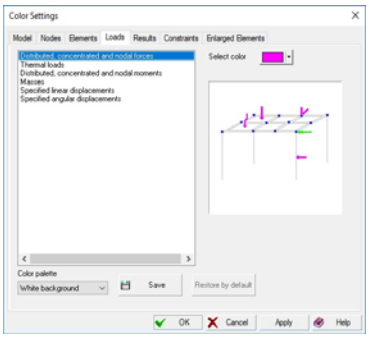
a)



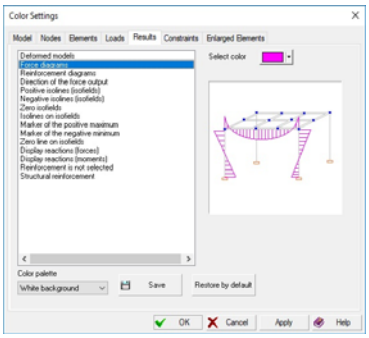
b)



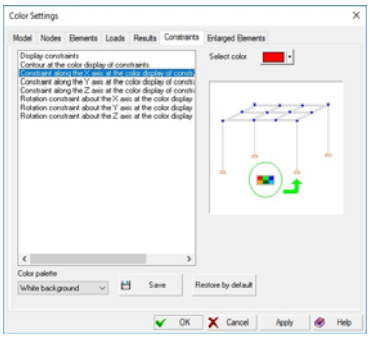
c)



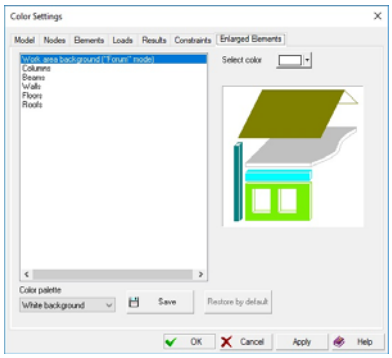
d)



e)



f)



g)

Figure 10.5-1. *The* **Color Settings** *dialog box tabs: a)* **Model**, *b)* **Nodes**, *c)* **Elements**, *d)* **Loads**, *e)* **Results,** *f)* **Constraints,** *g)* **Enlarged Elements**

In order to set the color for any of the possible display elements it is necessary to select its name from the list of elements and set the necessary color using the **Select color** button. The results of the selection will be displayed in the preview window with all the earlier performed settings in the background. You can switch to the set color by clicking the **Apply** button.

The software provides several “standard” color palettes **Standard**, **White background**, **Soft,** ..., which can be used by selecting the necessary variant from the **Color palette** drop-down list. Respective colors will be loaded which can be changed later. The changed color setting can be saved by clicking the respective button. The **Restore by default** button enables to restore a standard color palette, i.e. the color palette after the initial installation of the software.

Color settings are saved in the file of the project (for example, spr-file). If the dialog box for setting these parameters is invoked from the *Settings* menu when there are no open models, the saved colors will be used for all newly created projects.

11. Control of the Calculation

The calculation can be performed only if the initial data of the current project contain the obligatory minimum of information, i.e. geometry of the design model, description of stiffness properties of all elements, and at least one loading.

The calculation can be invoked from the respective section of the **Project tree**. A multi-tab **Calculation Parameters** dialog box appears after this (Fig. 11.1-1).

It is necessary to specify a working mode of the processor (**Complete analysis** or **Continued analysis**) in the **General Parameters** tab of this dialog box and select a solution method (Gaussian, Multifrontal, Iterative or PARFES). You also have to specify the necessary settings of the calculation process in this and other tabs.

11.1. General Parameters

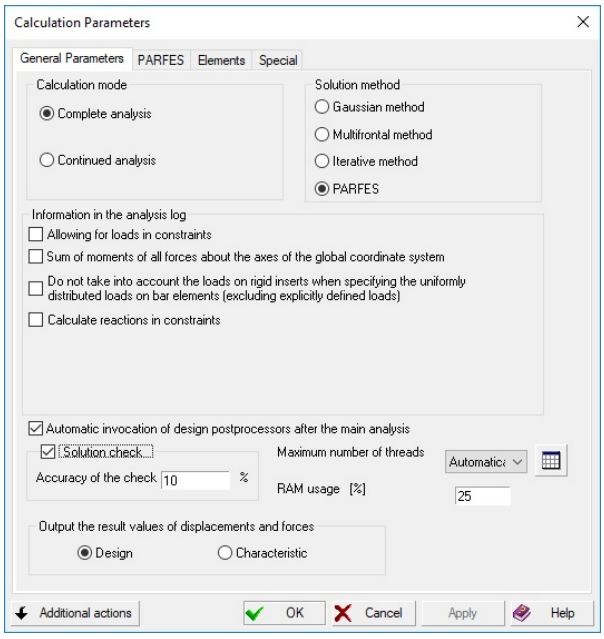


Figure 11.1-1. *The* **General Parameters** *tab of the* **Calculation Parameters** *dialog box*

It is recommended to use the **Continued analysis** mode after modifying the initial data not dealing with nodes, constraints and elements (for example after correcting the loads).

Before describing the settings let's consider the **Additional actions** button which can be found in each tab and which defines the rules for working with the specified parameters. Clicking this button invokes a menu where you can select one of the following three actions:

- **restore the values by default** — restores the initial settings (defined by the software);
- **save the values as the template** — enables to save the settings in a file of **SCAD** settings and to subsequently load them into other tasks;
- **restore the values from the template** — restores the settings which were saved by the previous operation.

Like other parameters the calculation settings are saved in the file of the project (for example, spr-file). If the *Calculation Parameters* dialog box is invoked from the *Settings* menu when there are no open models, the saved values will be used for all newly created projects. It is recommended to perform the analysis of efficiency of multi-threaded calculations (see below), and set the optimum number of threads for all newly created models.

The following settings are provided in the **General Parameters** tab:

- **Solution check** — this checkbox is used to enable or disable the procedure of checking the accuracy of the solution;
- **Accuracy of the check** — it is specified if the **Solution check** checkbox is checked and defines the error in the mode of checking the solution of a system of equations (10% by default); if it is exceeded a message indicating the obtained value of the error is generated and the calculation is terminated;
- **Allowing for loads in constraints** — when this option is activated the log will include the values of total loads in nodes taking into account the loads in nodes with imposed constraints;
- **Sum of moments of all forces about the axes of the global coordinate system** — when this option is activated the log will include the values of components of the principal moment of external forces;
- **Automatic invocation of design postprocessors after the main analysis** — if this option is activated, calculations of combinations of loadings, design combinations of forces and some other problems for which the initial data are prepared will be performed after performing static and dynamic calculations;
- **Do not take into account the loads on rigid inserts when specifying the uniformly distributed loads on bar elements** — this option enables to exclude a part of the external load which falls on the inserts from the calculation;
- **Calculate reactions in constraints** — if this option is activated, reactions in constraints will be determined in the result of the calculation;
- **Maximum number of threads** — enables to limit the number of threads the software can run in the calculation; if you select the option *automatically*, the software will select the optimal number of threads; if you want to run other applications during the calculation, you can limit the number of threads, thus leaving some computing power for other programs (the button enables to analyze the efficiency for different number of threads);
- **RAM usage** — percentage of the RAM usage for storing data not dealing with the solution of systems of equations;
- the software also enables to select which values of displacements/forces/stresses will be output in the postprocessor — **design** or **characteristic** (if EN 1990 is selected for calculating load combinations, these controls are not available and the program always calculates the characteristic values of displacements/forces/stresses).

Not only the computer processor (CPU), but also the processors of a video card are often used for the parallelization of computing operations. SCAD performs parallel calculations using only the possibilities of multi-threaded CPU. Processors of a video card (GPU) are used only for the graphical operations. This is due to the fact that integral losses for the operations of data exchange are too large in the regular graphics cards (which are used by the vast majority of users). This leads to a situation when the acceleration of actual calculations is offset by the exchange operations, which usually leads to the increase of the total operation time.

The usage of GPU can probably be effective for nonlinear problems of small dimension which require many iterations. Moreover, the necessary operating speed can be provided by special quite expensive video cards of Tesla type. However they are not used in the current version of the software, because the increase of the efficiency compared to CPU is observed only for very large stiffness matrices with very low density (see [21], [22]).

The second tab changes depending on the selected solution method: **Parameters of the Gaussian Method** (Fig. 11.2-1), **Multifrontal Method** (Fig. 11.3-1), **Parameters of the Iterative Method** (Fig. 11.4-1) or **PARFES** (Fig. 11.5-1).

11.2. Gaussian Method

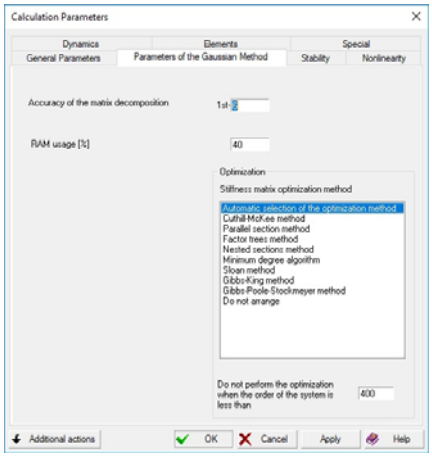


Figure 11.2-1. *The* **Parameters of the Gaussian Method** *tab*

If the Gaussian method is used, the **Accuracy of the matrix decomposition** is specified in the respective tab (Fig. 11.2-1); it defines the minimum value of the element of a stiffness matrix which is considered to be different from zero (the value of 10^{-6} is assumed by default). If a number less than the specified accuracy appears on the main diagonal in the process of the matrix decomposition, such a situation is interpreted as the dimensional instability of the system.

RAM usage (%) — indicates the percentage of RAM used for storing the information in the process of solving a system of equations (40% by default).

Moreover, you can also select the stiffness matrix optimization method, and indicate the minimum value of the order of the system of equations at which the optimization is not performed.

11.3. Multifrontal Method

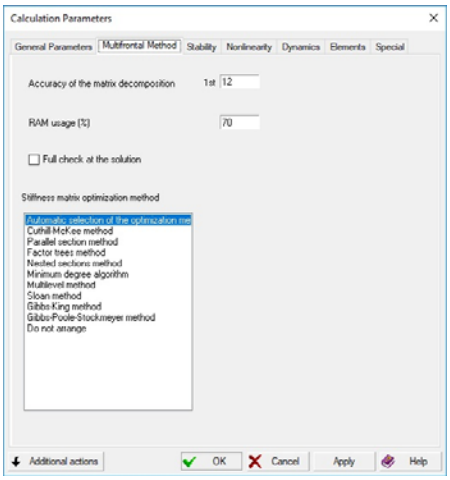


Figure 11.3-1. *The* **Multifrontal Method** *tab*

The **Accuracy of the matrix decomposition** is specified in the **Multifrontal Method** tab (Fig. 11.3-1); it defines the minimum value of the element of a stiffness matrix which is considered to be different from zero (the value of 10^{-12} is assumed by default). If a number less than the specified accuracy appears on the main diagonal in the process of the matrix decomposition, such a situation is interpreted as the dimensional instability of the system.

RAM usage (%) — indicates the percentage of RAM used for storing the information in the process of solving a system of equations (70% by default).

Moreover, you can also select the stiffness matrix optimization method.

If the **Full check at the solution** checkbox is checked, the control of the accuracy of the solution is performed for all loadings.

11.4. Iterative Method

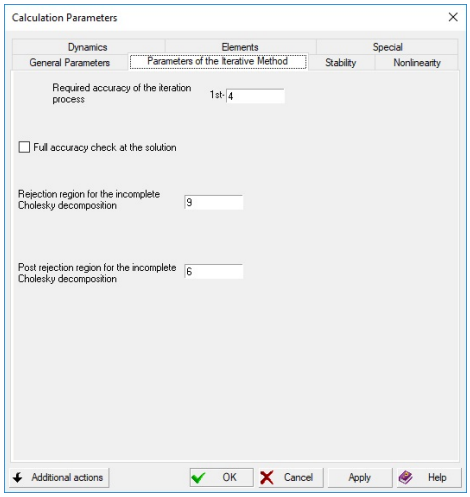


Figure 11.4-1. *The* **Parameters of the Iterative Method** *tab*

The **Required accuracy of the iteration process** and the **Full accuracy check at the solution** option are specified in the **Parameters of the Iterative Method** tab (Fig. 11.4-1). The latter enables to obtain the information on the error of the solution for each loading.

It is recommended to change other parameters only on the instruction of the developers.

11.5. PARFES

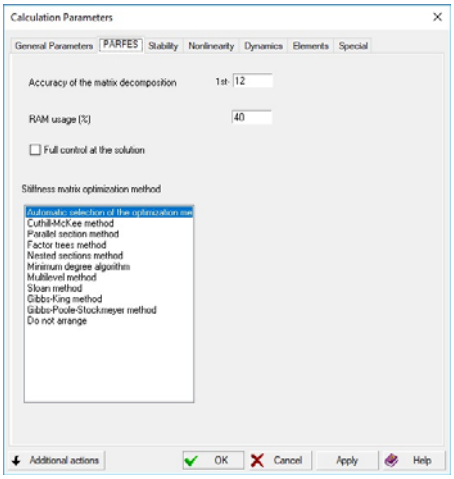


Figure 11.5-1. *The* **PARFES** *tab*

The **Accuracy of the matrix decomposition** is specified in the **PARFES** (Parallel Sparse Direct Solver) tab (Fig. 11.5-1); it defines the minimum value of the element of a stiffness matrix which is considered to be different from zero (the value of 10^{-12} is assumed by default). If a number less than the specified accuracy appears on the main diagonal in the process of the matrix decomposition, such a situation is interpreted as the dimensional instability of the system.

RAM usage (%) — indicates the percentage of RAM used for storing the information in the process of solving a system of equations (40% by default).

Moreover, you can also select the stiffness matrix optimization method.

11.6. Stability

The following parameters are specified in the **Stability** tab (Fig. 11.6-1):

- **Upper search interval limit** — the maximum allowable increase of loads which defines the search range of the value of the stability factor of safety (2 by default);
- **Accuracy of the calculation of stability factors of safety** — accuracy of determining the stability factor of safety (0.01 by default);
- **Accuracy of the determination of the buckling modes** — allowable error at the determination of the buckling mode (10^{-6} by default for linear displacements);
- **Maximum number of iterations at the determination of the buckling modes** — number of iterations after which a poorly convergent process is interrupted (100 by default);
- **Number of buckling modes** — parameter defines the default value for the number of buckling modes of the problems in which the data on stability are not yet specified;

- **Allowing for tangential displacements in the geometric stiffness matrix of shells** — parameter enables not to take into account the tangential displacements in the analysis of the stability of shells;
- **Allowing for possible buckling at the bending of bars**[1] — parameter enables not to take into account the possible buckling at the bending of bars (if this checkbox is not checked, the results of the stability analysis will correspond to the results of the previous versions of **SCAD**, where this possibility was not implemented).

The tab with the settings of the stability analysis mode given in Fig. 11.6-1 is available at the initial customization of the software (all projects must be closed). Information on the upper search limit of the stability factor of safety, accuracy of its calculation, and the number of buckling modes are specified for each particular problem. The values of the above parameters are loaded by default into the respective fields of the **General Stability Check of the System** dialog box (see Section 24) and can be changed.

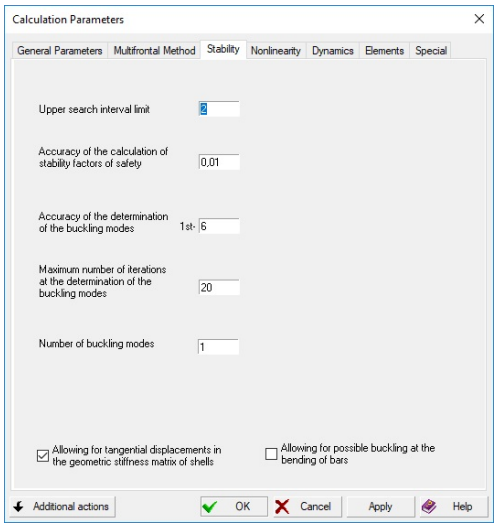


Figure 11.6-1. *The* **Stability** *tab*

11.7. Dynamics

The **Dynamics** tab (Fig. 11.7-1) enables to select the method of the solution of the eigenvalue problem (subspace iterations, method of steepest descent or Lanczos method). A "preferred" variant for all loadings is specified here, it can be changed later when specifying the initial data to the dynamic loading. Additional parameters of the mode are also specified at that time.

Lanczos method of the solution of the eigenvalue problem is specified in the **Dynamics** tab and has the highest priority.

Checkboxes in this tab enable to specify the information which will be output into the analysis log:

- dynamic data;
- plots of the dynamic amplification factor;
- coefficients for the calculation of inertial loads (for example, the dynamic amplification factors β and coefficients η depending on the deformation mode of the building or structure will be derived for the seismic actions).

You also have to specify here the accuracy of the solution of the eigenvalue problem for the Lanczos method, and for the method of subspace iteration and the method of steepest descent.

The software implements the block Lanczos method with spectral transformations [16] – [17], which provides four calculation modes: modal, interval, seismic, and the mode of verification of the design model. Two options can be selected in the **Dynamics** tab (the analysis mode or the mode of verification of the design model). If the first option is selected, the program will automatically (depending on the data of a

[1] The analysis of in-plane bending is considered for bars with a narrow rectangular section in a classic work by Timoshenko S.P. Course of the Theory of Elasticity. — Kiev: Naukova Dumka, 1972. 508 p.

certain dynamic loading) select the mode (modal, seismic, or interval). If the mode of verification of the design model is selected, only the oscillation modes will be calculated (errors in the design model can be found on their basis); the calculation of displacements, forces, ... will not be performed in this case.

Modal analysis — determines the number of frequencies and oscillation modes specified in the **Modal Analysis** tab of the **Parameters of Dynamic Actions** dialog box (see Section 8.7). If a greater number of eigenpairs have converged in the iteration process, the results of the calculation will contain all these pairs.

Interval mode — determines all eigenpairs (frequencies and modes) in the frequency range specified in the **Dynamics** tab of the **Calculation Parameters** dialog box when setting the operation of the Lanczos method.

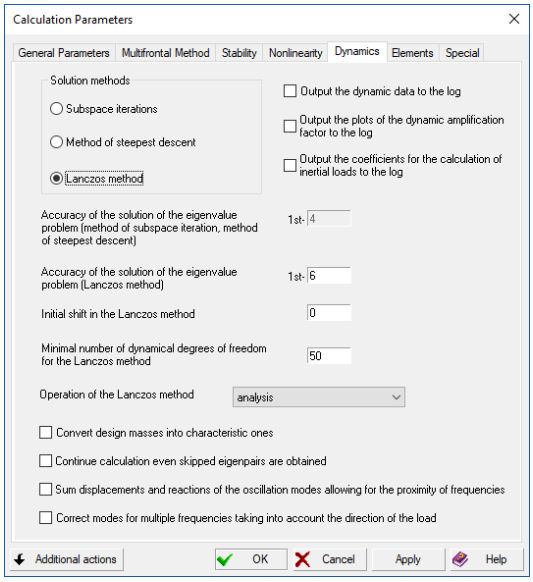


Figure 11.7-1. *The* **Dynamics** *tab of the* **Calculation Parameters** *dialog box*

Seismic mode determines frequencies and natural oscillation modes until the sum of modal masses for each direction X, Y and Z exceeds the values specified in the **Parameters of Dynamic Actions** dialog box. Like in the case of interval mode it is not known beforehand what number of eigenpairs has to be determined.

Verification of the design model mode — enables to detect the dimensional instability of the design model. The number of zero eigenvalues or those very close to zero will show the degree of degeneration of the stiffness matrix, and their respective vectors – modes of the motion of the mechanism. Visualization of the motion modes in the mode of the analysis of displacements enables to obtain information suggesting how to correct the design model. The *initial shift in the Lanczos method* is used in the initial data of the verification mode. It is set to zero by default and is changed automatically. In some (quite rare) cases this parameter can be specified according to the recommendations of the developers.

Experiments show that quite often the Lanczos method is numerically unstable at small problems. Therefore it is possible to specify **the minimum number of dynamic degrees of freedom**. If in the problem this number is less than the selected one, the program will automatically use a different method.

One of the ways of specifying masses in the dynamic calculation is the transformation of a static loading. This tab contains the **Convert design masses into characteristic ones** checkbox which defines the rule for such a transformation, which is used for all dynamic loadings (if EN 1990 is selected for calculating load combinations, this control is not available).

If the code contains instructions on how to take into account the proximity of frequencies and the user has set the respective option (for example, in the calculations according to SP 14.13330.2014), then forces are summed according to the corresponding formula given in the code. If you want to sum displacements and reactions according to the same rules, check the **Sum displacements and reactions of the oscillation modes allowing for the proximity of frequencies** checkbox.

The **Correct modes for multiple frequencies taking into account the direction of the load** checkbox enables to use a special procedure for working with modes corresponding to multiple frequencies. If this checkbox is checked, then the sum vector is rotated so as to obtain its maximum contribution to the overall dynamic response of the structure.

Taking into account the correlations between all modal pairs according to formula (5.13) from SP 14.13330.2018 with the modification No. 1 and other codes leads to very cumbersome calculations. However, the accuracy of the calculation procedures should correspond to the accuracy of the initial data. And since this accuracy is not high in seismic analysis, many guidelines recommend ignoring the correlation between modes with a correlation factor less than 0.2 (see, for example, Birbraer A.N. *Seismic Analysis of Structures* — St. Petersburg: Nauka, 1988). The default rejection threshold for

correlation factors in **SCAD** is 0.1. You can increase the accuracy in the **Accuracy of the correlation matrix coefficients** field.

11.8. Nonlinearity

The following parameters are specified in the **Nonlinearity** tab (Fig. 11.8-1):

- **Accuracy of the iteration process;**
- **Do not automatically add a static part of the dynamic wind loading to the nonlinear loading;**
- **Number of steps added to the nonlinear loading for the static part of the dynamic wind;**
- **False unload bypass.**

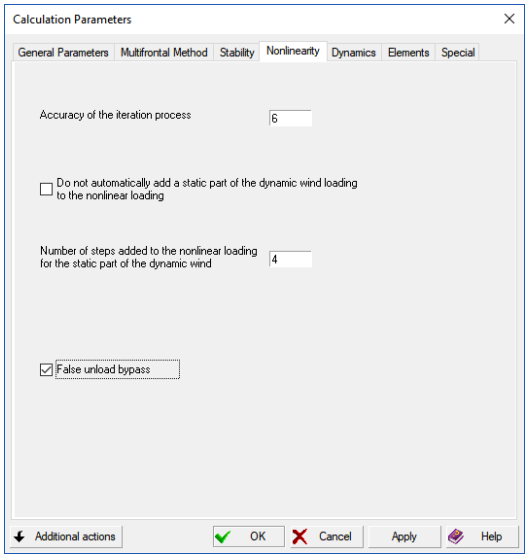


Figure 11.8-1. *The* **Nonlinearity** *tab*

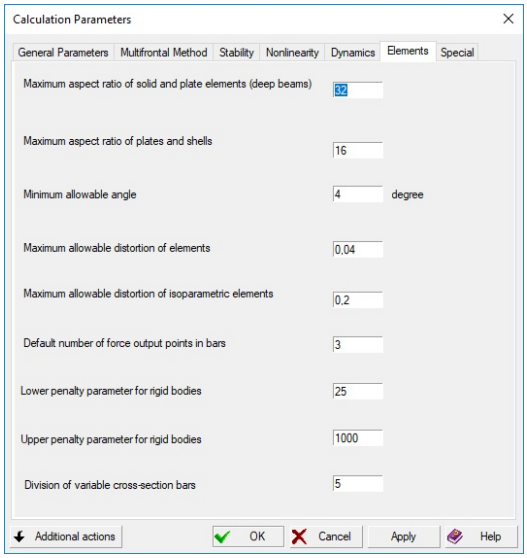


Figure 11.9-1. *The* **Elements** *tab*

When unloading is taken into account (plasticity, but not nonlinear elasticity), false unloading occurs when the iterative process has not yet converged, and in order to achieve this convergence it is necessary to reduce the stress level, but this decrease does not take place in real life. This can sometimes lead to convergence blocking. If the "False unload bypass" checkbox is checked, each iteration step is based not on the previous iteration step, but on the initial stress-strain state for the current load increment. We recommend using this option only for homogeneous materials and only if the problems with convergence cannot be solved otherwise (than by decreasing the load step). This cannot be done for reinforced concrete with a descending branch of the σ – ε diagram, because the convergence of the iterative process will be blocked.

The user can generate a nonlinear loading by including a static part of the dynamic wind into it.

11.9. Elements

The **Elements** tab (Fig. 11.9-1) enables to specify the **default number of force output points in bars**. This number will be used in the calculation of forces in bars for all bar elements where this parameter is not set explicitly in a specific element (elements).

Limitations set by the following parameters are used in the control of the geometric shape of finite elements:

- **Maximum aspect ratio of solid and plate elements (deep beams);**
- **Maximum aspect ratio of plates and shells;**
- **Minimum allowable angle;**
- **Maximum allowable distortion of elements;**
- **Maximum allowable distortion of isoparametric elements;**
- **Default number of force output points in bars.**

Violation of limitations defined by the above parameters is interpreted by the software as an error, the respective message is generated, and the calculation is not performed.

SCAD models variable cross-section bar elements by a set of constant cross-section bars. The number of such bars is defined by the **Division of variable cross-section bars** parameter. The default value of this parameter is 5, which can provide a fairly accurate result. However, the user can specify a larger value (but not greater than 100).

Finite elements of the rigid body type are implemented using the Lagrange multipliers. The following parameters are used (for a detailed description see [23]):

- **lower penalty parameter**;
- **upper penalty parameter**.

You can change their default values only if it is recommended by the developers.

11.10. Special Settings

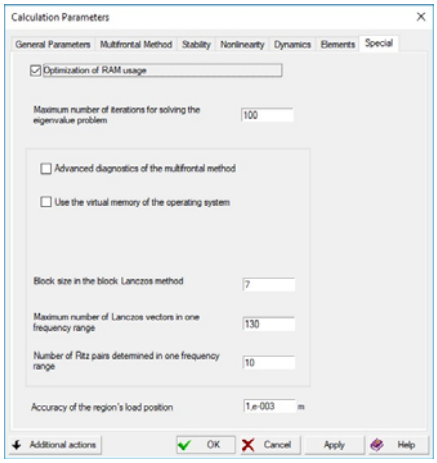


Figure 11.10-1. *The* **Special** *tab*

The following parameters are specified in the **Special** tab (Fig. 11.10-1):

- **Maximum number of iterations for solving the eigenvalue problem**;
- **Optimization of RAM usage** — if this checkbox is checked, the calculation is performed in a separate process, otherwise the main process of **SCAD** is used;
- **Use the virtual memory of the operating system** — if you have to solve very large problems and the amount of RAM is limited, you can check this checkbox (it might significantly slow down the calculation, but it increases the dimension of the problems that can be solved).

Other parameters in this tab can be changed only if it is recommended by the developers.

11.11. RAM Usage

The speed of calculations, especially for large problems, depends on the amount of RAM available for the problem. The rules of the memory distribution on different stages of the calculation process can be defined by the user. Special settings of the "percentage of memory" are provided in two tabs of the **Calculation Parameters** dialog box — in the **General Parameters** tab, where the parameters not depending on the selected method of calculation are specified, and in the tab of settings depending on the selected method of calculation.

The specified "percentage of memory" is applied to obtain the maximum possible segment of continuous RAM available to the process. It should be noted that if a 32-bit version of the program is used, not more than 3.2 GB is available for each process, regardless of the amount of RAM installed on your computer.

It is necessary to distinguish between three different situations that arise in the process of calculation.

1. Matrix decomposition (for example, using the PARFES method), when the memory is required only for the "pieces" of the stiffness matrix. In this case, if the **RAM usage** is specified as 70% in the **PARFES** tab, the software will try to take up to 70% of the free memory available for the process.
2. Calculation of displacements, when the memory is required for the right parts (vectors of loads), calculated displacements and "pieces" of the decomposed stiffness matrix. Let's assume that it is specified in the **General Parameters** tab that 40% of memory can be used. The software will take 40% of the free memory available for the process and will use them for the right parts and displacements. The remaining memory (but not more than 70%) will be used for the "pieces" of the decomposed stiffness matrix. If there is less than 70%, as much as possible memory will be used.
3. Calculation of forces, DCF. The memory is requested according to the settings in the **General Parameters** tab.

The checked **Use the virtual memory of the operating system** checkbox in the **Special** tab enables the software to use not only RAM in the calculation but also the virtual memory. This checkbox should not be used without the recommendations of the developers, since it can significantly slow down the calculation process.

11.12. Optimization of the Stiffness Matrix

The **Parameters of the Gaussian Method**, **Multifrontal Method**, and **PARFES** tabs enable to select the stiffness matrix optimization method.

The following optimization methods are provided in the software:

- Cuthill-McKee method
- Parallel section method
- Factor trees method
- Nested sections method
- Minimum degree algorithm
- Multilevel method
- Sloan method
- Gibbs-Poole-Stockmeyer method
- Gibbs-King method

Moreover, the list of methods also contains the **Automatic selection of the optimization method** item. If it is selected, the software will try to perform the optimization by several methods and will select the best one. Depending on the used solution method the automatic selection will be performed from the methods listed in the table.

	Gaussian method	Multifrontal method	PARFES
Cuthill-McKee method	+		
Minimum degree algorithm		+	+
Multilevel method		+	+
Sloan method	+	+	+

As already mentioned in Chapter 2, the problem of determining the optimal renumbering of the degrees of freedom belongs to “difficult problems”, therefore all the optimization algorithms are heuristic. If the automatic selection does not significantly reduce the parameters of the stiffness matrix (for example, the profile of the matrix), it is advisable to try other methods implemented in the software by explicitly selecting them from the list.

Optimization of matrices for problems with a small number of degrees of freedom is unreasonable, since the time spent on the optimization may exceed the time for solving an unoptimized problem. Therefore you can specify the limit value of the number of the degrees of freedom in the **Do not perform the optimization when the order of the system is less than**... field; the matrix optimization will be performed when they are reached.

11.13. Multi-threaded Calculations

Most modern CPU (Central Processor Unit) are multi-core, i.e. they include several independent physical cores which can work simultaneously. Moreover, each core can alternately implement multiple threads (“hyper-threading” technology), thereby creating multiple logical processors on the basis of each physical core.

When performing the calculation operations (decomposition of the stiffness matrix, calculation of displacements, forces etc.) almost all calculations are performed using a multi-threaded mode, i.e. with the parallel processing of data. The user can specify (select from the list) the maximum number of threads that can be used for the current problem.

When the number of threads increases, the efficiency of the program usually increases as well and the calculation time reduces. This growth will be observed in cases when the number of threads specified by the user does not exceed the number of physical cores.

If the *Automatically* item is selected in the **Maximum number of threads** list, the “optimal” number of threads is determined by the program and it will be taken as half the sum of the number of physical and

the number of logical cores. If the specified **maximum number of threads** is greater than the number of physical cores, then depending on a particular model of CPU the efficiency can increase insignificantly, and in some cases it can even decrease (threads begin to "interfere" with each other).

The button in the **General Parameters** tab of the **Calculation Parameters** dialog box enables to analyze the efficiency of a particular CPU. Clicking on it invokes the operation of determining the optimal number of threads. The results of performing the operation are displayed in the respective dialog box (Fig. 11.13-1).

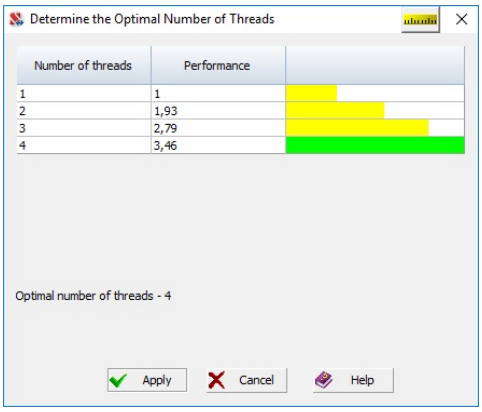


Figure 11.13-1. *The* **Determine the Optimal Number of Threads** *dialog box*

It is recommended to analyze the dependence of the efficiency on the number of threads before specifying the **Maximum number of threads**. This is due to the fact that for many CPU using the maximum number of logical threads at the minimum increase of the computing efficiency of **SCAD** may lead to a significant reduction of the operating speed of other programs working parallel with **SCAD**.

11.14. Description of Solvers

Linear and Nonlinear Statics

A system of linear algebraic equations is solved in linear static problems

$$\mathbf{Kx} = \mathbf{b}, \tag{1}$$

where **K** – symmetric (usually sparse) stiffness matrix which appears when the finite element method is used for solving the problems of structural mechanics and solid mechanics; **x**, **b** – vectors of unknowns and right-hand sides (loadings). The Newton-Raphson method or the modified Newton-Raphson method is used in the nonlinear static problems. A system of linearized algebraic equations (1) is solved at each iteration, where **K** – symmetric tangential stiffness matrix, **x** – vector of the increment of displacements and rotation angles at the current iteration, **b** – residual vector.

Both the direct methods, and the iterative one – conjugate gradient method with preconditioning, are used in **SCAD** for solving the system of linear algebraic equations (1).

Direct methods – multifrontal and PARFES – use the generalized Cholesky decomposition

$$\mathbf{K} = \mathbf{L}\cdot\mathbf{S}\cdot\mathbf{L}^T, \tag{2}$$

where **L** – sparse lower triangular matrix, **S** – diagonal of coefficients which generalizes the Cholesky decomposition in case of indefinite matrices typical when solving the problems of the modal analysis and stability. Then the software performs the forward substitution

$$\mathbf{Ly} = \mathbf{b} \rightarrow \mathbf{y}, \tag{3}$$

diagonal scaling

$$\mathbf{Sz} = \mathbf{y} \rightarrow \mathbf{z}, \tag{4}$$

and back substitution

$$\mathbf{L}^T\mathbf{x} = \mathbf{z} \rightarrow \mathbf{x}. \tag{5}$$

Moreover, the Gaussian method uses the decomposition:

$$\mathbf{K} = \mathbf{L}\cdot\mathbf{U}, \tag{6}$$

where **L**, **U** – lower and upper triangular matrices respectively, and then the forward (3) and back substitutions are performed

$$\mathbf{U}\mathbf{x} = \mathbf{y} \rightarrow \mathbf{x}. \tag{7}$$

When the iterative method is used the original problem (1) comes down to

$$\mathbf{B}^{-1}\mathbf{K}\mathbf{x}_i = \mathbf{B}^{-1}\mathbf{b}_i,\ i \in [1, N_r]\ , \tag{8}$$

where i – number of the right-hand side, N_r – number of the right-hand sides, **B** – preconditioning operator constructed in such a way so that the system of equations $\mathbf{B}\mathbf{z}_i^k = \mathbf{r}_i^k \rightarrow \mathbf{z}_i^k$, having the same order as (1) was solved much faster than (1), and the system of equations (6) had a better conditioning than (1): $Cond(\mathbf{B}^{-1}\mathbf{K}) < Cond(\mathbf{K})$, where $Cond(...)$ – conditioning number. Then the convergence of (6) will be faster. Here $\mathbf{r}_i^k = \mathbf{b}_i - \mathbf{K}\mathbf{x}_i^k$ – residual vector at the current iteration step k. The solution procedure consists of two stages: creating preconditioning and the iterative process of the conjugate gradient method. Conditions at which the iterative process is completed are described below.

Multifrontal Solver

This is a direct method for solving systems of linear algebraic equations (1). In **SCAD** this method implements the idea of the superelement method with an automatic division of the initial design model into substructures without the user intervention and is given in the publications [1-5] as the block substructure multifrontal method BSMFM. Decomposition of a large dimension sparse matrix comes down to a sequence of decompositions of dense matrices of relatively small dimensions (frontal matrices), which enables to use effectively almost all methods for achieving high performance available on multi-core desktops. The decomposed matrix is stored in dense blocks. If the dimension of the problem exceeds the RAM capabilities, the blocks of the decomposed matrix are saved in a file on the disk.

The accuracy of the matrix decomposition is defined by the parameter ε_t, which enables to identify the matrix singularity or poor conditioning at the factorization (it is taken as 10^{-12} by default – Fig. 11.3-1).

A diagonal of the stiffness matrix **K** is compiled before the decomposition. If at any stage of the decomposition of the frontal matrix it turns out that $|\mathbf{L}_{jj}/\mathbf{K}_{jj}| < \varepsilon_t$, where $\mathbf{L}_{jj}$ – diagonal element of the factorized matrix, then the stiffness matrix is either singular or poorly conditioned. In order to be able to continue the decomposition of the matrix, the element $\mathbf{L}_{jj}$ is assigned a large value (10^{50}). A list of all the degrees of freedom where it occurred is output at the end of the decomposition. Parameter ε_t can be reduced to 10^{-14}. However, if the messages on the dimensional instability appear again, the calculation should be aborted. It is likely that this design model is dimensionally unstable. It is necessary to determine the cause of the dimensional instability and eliminate it (block Lanczos method can be used in the mode of the verification of the design model).

Moreover, it can turn out that the design model leads to a poorly conditioned system of linear algebraic equations. Such systems are highly sensitive to small perturbations of the right-hand side and accumulate large errors in the decomposition of the matrix. Therefore, it is necessary to determine the cause of the poor conditioning and eliminate it. *Reducing the parameter* ε_t *further* (*for example, to* 10^{-50}) – *is like ignoring the thermometer readings when having the flu or riveting a pressure relief valve in the boiler.*

RAM usage is a parameter that regulates the maximum amount of RAM for the solver. If you want to solve the problem as quickly as possible, set 80 – 90%. Then the solver will not use the slow disk memory until all free RAM is exhausted. If the free RAM is enough to solve this problem, matrix decomposition

will be performed as quickly as possible. If it is not enough, the solver will still use the disk memory. If the "RAM usage" exceeds the amount of RAM required for the decomposition of the matrix without using the disk, the solver uses only the required amount of RAM, and not the specified value. If you want to solve simultaneously several problems on one computer, then the "RAM usage" should be specified for each problem in such a way so that all the simultaneously solved problems would not exceed the amount of the free RAM. You should be careful when assigning this parameter a value of more than 90%, because at a certain computer configuration it can cause an intense exchange with the page file of the operating system, significantly slow down the calculation process and temporarily block the computer. Further the solution of the problem will continue.

The practice of calculations shows that the control at the matrix decomposition based on the analysis $|\mathbf{L}_{jj}/\mathbf{K}_{jj}| < \varepsilon_t$, does not work in all cases, because when specifying the parameter ε_t we do not have any information on the conditioning of the matrix **K**. If the value of ε_t is too small, the dimensional instability can be missed, while a large value can cause a "false alarm". Therefore, it is recommended to use the "full check at the solution" option, which enables to perform the analysis of the following values for each loading $\varepsilon_i = \|\mathbf{b}_i - \mathbf{K}\mathbf{x}_i\|_2/\|\mathbf{b}_i\|_2$, $i \in [1, N_r]$, where i – number of the loading, N_r – number of loadings, $\mathbf{b}_i$ – load vector, $\mathbf{x}_i$ – vector of the solution, $\|\ldots\|_2$ – Euclidean vector norm, ε_i – measure of error. For well-conditioned systems of equations $\varepsilon_i < 10^{-8}$. If $\varepsilon_i > 10^{-4}$, we consider that the solution is unacceptable because the error is too big. Value $10^{-8} < \varepsilon_i < 10^{-4}$ indicates that the system of equations is "poorly conditioned" – the design model is correct, but small modifications can make it incorrect. The calculated values ε_i are placed in a file with extension p04 in the SWORK directory. The calculation of ε_i is performed only for the linear static analysis and it is blocked at the analysis of frequencies and natural oscillation modes, stability and nonlinear analysis. If the **Full check at the solution** checkbox is not checked, ε_i is not calculated.

The task of ordering the matrix is the reduction of the number of non-zero elements in the factorized stiffness matrix, which enables to make the matrix decomposition dozens of times faster, and the forward and back substitutions – a few times faster [4]. Unfortunately, it is not known in advance which method of ordering is the best. Therefore, if the problem is solved for the first time, it is advisable to leave the choice of the method of ordering to the solver. In order to do it select the "Automatic selection of the optimization method" item in the calculation parameters. If the topology of the design model has not changed and the recalculation is performed, and the method of ordering works slowly, you can specify a method of ordering which was selected by the software in the previous calculation in the mode of the automatic selection. Then the solver will take the specified method and will not waste time searching for the best method. If the stage of the optimization of the stiffness matrix is performed quickly, you should use the automatic selection of the method of ordering at the further recalculations as well.

PARFES

PARFES [6-11] – Parallel Finite Element Solver – direct method for sparse matrices, designed specifically for multi-core computers and workstations with shared memory (shared memory computers). PARFES belongs to the class of supernode solvers and implements a block version of the Cholesky decomposition. Almost in all cases this method shows higher performance and accelerance at the increase of the number of processors on the shared memory computers than the multifrontal solver. In order to achieve peak performance the whole problem must fit in RAM. Otherwise the solver automatically selects the OOC mode (out of core) which uses the external disk. The performance of the method at the stage of the matrix decomposition decreases insignificantly in this mode in comparison with the RAM operation mode, but the duration of the forward/back substitutions increases significantly. In the case of a large problem and small RAM the OOC mode becomes unavailable, and PARFES selects the OOC1 mode. In this mode the solver uses a much smaller amount of RAM than in the OOC mode, but it performs a much greater number of exchanges with the disk, which results in a significant decrease of the performance at the stage of the matrix decomposition. The duration of the forward and back substitutions remains the same as in the OOC mode.

PARFES works with any method of ordering. It uses all methods of achieving the high performance available in the shared memory computers.

The accuracy of the matrix decomposition, RAM usage, full check at the solution and the selection of the ordering method completely correspond to the options of the multifrontal method given above.

Iterative Solver

It implements the conjugate gradient method with preconditioning of the incomplete Cholesky decomposition $\mathbf{B} = \mathbf{H}\cdot\mathbf{H}^T$, where $\mathbf{H}$ – lower triangular matrix. The incomplete Cholesky factorization method is developed in **SCAD**. It is based on the sparse matrix technology, which enables to accelerate the convergence considerably at a moderate increase of the size of the incomplete Cholesky factor $\mathbf{H}$ [12-13]. The ordering is performed by the minimum degree algorithm [14].

Rejection and post rejection parameters ψ, ψ_1 ($0 < \psi < \psi_1 < 1$) (Fig. 11.4-1) control the ability of preconditioning to accelerate the convergence. Only "small elements" remain – $\mathbf{H}_{ij}^2 \geq \psi\ \mathbf{H}_{ii}\mathbf{H}_{jj}$, at the matrix decomposition. The smaller ψ, the closer the incomplete Cholesky factor $\mathbf{H}$ to the complete factor $\mathbf{L}$ and the faster the solution converges, but the greater the amount of RAM is required to store $\mathbf{H}$ and the more time is required to perform the incomplete Cholesky decomposition, and to perform the iterations. There might not be enough amount of RAM for very small values ψ. Then the solver increases ψ 10 times and tries to perform the incomplete Cholesky decomposition again. It continues until the incomplete Cholesky factor $\mathbf{H}$ is saved in RAM. Therefore when performing the recalculations on the same computer it is advisable to use the experience of the first calculation and take the value of the parameter ψ at which the incomplete factorization was completed. This way you will avoid unnecessary attempts at decomposition.

On the other hand, if the value ψ is too small, the duration of the incomplete factorization can increase significantly on the computers with large amount of RAM. Actually there are no methods of determining the optimal value of the parameter ψ. It depends on the conditioning of the system of linear algebraic equations, amount of RAM, dimension of the problem, number of right-hand parts, and the need to perform modal/seismic analysis. Therefore, we give the following recommendations. On a 32-bit platform the value of the parameter ψ is taken as 10^{-9} by default, and you should begin with this value. On the 64-bit platform, the majority of computers today have a large amount of RAM available for the operating system. Therefore, we recommend taking the value ψ within $10^{-12} – 10^{-14}$. If for the current problem it turns out that the size of $\mathbf{H}$ exceeds the RAM capabilities, the solver will "correct" you and reduce ψ. If the incomplete Cholesky decomposition takes unreasonably long time, you should abort the calculations and increase ψ.

After completing the factorization the secondary rejection is performed – only the elements for which $\mathbf{H}_{ij}^2 \geq \psi_1\ \mathbf{H}_{ii}\mathbf{H}_{jj}$ remain. It enables to speed up the procedure of forward/back substitutions in the iteration process and reduce the amount of required RAM. The value ψ_1 is usually taken three orders of magnitude greater than the value ψ.

The procedure of incomplete factorization is parallelized. The parallelization of the iterative procedure is performed by iterating each right-hand side independently of the others on a separate thread. Therefore, time for solving the problem containing a large number of loadings depends greatly on the number of cores (processors) on your computer.

The iterative process continues until $\varepsilon_i > \varepsilon_t$, where $\varepsilon_i = \|\mathbf{b}_i - \mathbf{K}\mathbf{x}_i\|_2/\|\mathbf{b}_i\|_2$, $i \in [1, N_r]$, ε_t – required accuracy.

The iterative solver can be efficient for solving very large problems (several millions of equations), and also in those cases when the direct solvers work in the modes which require the use of the disk space.

The iterative solver is available for the following types of analysis: linear and nonlinear statics, modal and seismic analysis (spectral method).

Gaussian Method

Gaussian method (see [15]) is the direct method for symmetric sparse matrices. It implements the LU decomposition (6). The selection of the pivot element is not performed, because we deal with diagonally dominant symmetric matrices in the finite element design models of structural mechanics and solid mechanics. The "accuracy of the matrix decomposition", "RAM usage", and the "stiffness matrix optimization method" have the same meaning as for PARFES and multifrontal solver.

11.15. Description of the Dynamic Solvers

The problem of determining the lower frequencies and the natural oscillation modes of the buildings and structures when solving by the finite element method comes down to solving an algebraic partial generalized eigenvalue problem

$$\left(\mathbf{K}-\lambda_j\mathbf{M}\right)\mathbf{V}=0$$

For the convenience of the calculations this problem can be given in the following form

$$\left(\mathbf{K}_\sigma-\lambda_j\mathbf{M}\right)\mathbf{V}=0, \qquad (11.15.1)$$

where $\mathbf{K}_\sigma = \mathbf{K} - \sigma\cdot\mathbf{M}$, $\mathbf{K}$ – stiffness matrix, $\mathbf{M}$ – diagonal or consistent mass matrix, σ – shift parameter[1], $\mathbf{V} = \{\mathbf{v}_1, \mathbf{v}_2, \ldots, \mathbf{v}_n\}$, $\mathbf{v}_i$, $i \in [1, n]$ – first n eigenvectors which have to be determined, λ_j – eigenvalues $\lambda_1 \le \lambda_2 \le \ldots \le \lambda_n$, and $\lambda_i = \omega_i^2 = (2\cdot\pi\cdot f_i)^2$, where f_i [Hz] – frequency of the natural oscillations. The dimension of the problem is equal to $N \ge n$, where N — number of the degrees of freedom.

Three methods of the modal analysis are implemented in **SCAD**. The subspace iterations method and the method of the steepest descent are classic and are described in many books on computational linear algebra (see, for example, [18]). The implementation of the block Lanczos method has some peculiarities, therefore this method is described in more detail below.

Subspace Iterations Method

It is a classic iterative method described, for example, in Chapter 14 of [18].

The following criterion for the estimation of the eigenvalue (λ_j) calculation error is used in the algorithm:

$$\left|\frac{\lambda_j^{(k+1)}-\lambda_j^{(k)}}{\lambda_j^{(k)}}\right|<\varepsilon,$$

where $\lambda_j^{(k+1)}$, $\lambda_j^{(k)}$ — two successive eigenvalues on the k+1, k iteration steps, and ε denotes the accuracy specified in the **Dynamics** tab of the **Calculation Parameters** dialog box (Fig. 11.7-1) for the subspace iterations method. The iterations stop if the calculation error is less than the specified value ε or the number of iterations exceeds the limitation specified in the **Special** tab of the **Calculation Parameters** dialog box (Fig. 11.10-1).

Method of Steepest Descent

This classic method is based on the minimization of the generalized Rayleigh quotient [18, § 1.7]. The search of the minimum is performed by the iterative gradient descent method.

The following criterion for the estimation of the eigenvalue (λ_j) calculation error is used in the algorithm:

$$\left|\frac{\lambda_j^{(k+1)}-\lambda_j^{(k)}}{\lambda_j^{(k)}}\right|<\varepsilon,$$

where $\lambda_j^{(k+1)}$, $\lambda_j^{(k)}$ — two successive eigenvalues on the k+1, k iteration steps, and ε denotes the accuracy specified in the **Dynamics** tab of the **Calculation Parameters** dialog box (Fig. 11.7-1) for the method of steepest descent.

[1] A non-zero shift value is used in the analysis of the dimensionally unstable structures.

The iterations stop if the calculation error is less than the specified value ε or the number of iterations exceeds the limitation specified in the **Special** tab of the **Calculation Parameters** dialog box (Fig. 11.10-1).

Lanczos Method

The description of the algorithm of the block Lanczos method is published in the paper [16]. The implementation of the Lanczos method used in **SCAD** provides four calculation modes: *modal*, *interval*, *seismic* and *the verification of the design model mode*, which are described below.

Modal mode. The frequencies and the natural oscillation modes the number of which is specified in the **Modal Analysis** tab of the **Parameters of Dynamic Actions** dialog box (Fig. 8.7-1) is determined. The initial shift value is specified in the **Dynamics** tab of the **Calculation Parameters** dialog box (Fig. 11.7-1). The default shift is σ = 0. The Lanczos method has to be selected in the **Parameters of Dynamic Actions** dialog box (Fig. 8.7-1), and the **Analysis in the specified frequency range** checkbox has to be unchecked.

Interval mode. The interval mode determines all eigenpairs in the interval [*a*,*b*]. After selecting the Lanczos method in the **Parameters of Dynamic Actions** dialog box (Fig. 8.7-1), it is necessary to check the **Analysis in the specified frequency range** checkbox and specify the start *a* and end *b* frequencies which define the frequency range [*a*, *b*]. Keep in mind the fact that the spectrum of the natural frequencies of the design models of most of the structures is very dense, and there may be several thousand natural frequencies in the range of a few Hertz.

Seismic mode. It determines the frequencies and the natural oscillation modes until the specified sum of modal masses for each of the principal directions of the seismic action OX, OY, OZ is provided (OXYZ – global coordinate system). Moreover, if the cutoff frequency is specified (in order to do it check the **Maximum frequency limitation** checkbox, a dialog box will appear where you have to enter the value of this frequency), the algorithm will finish its work as soon as one of the two events occurs.

- The sum of the modal masses will be obtained for each of the directions of the seismic action OX, OY, OZ.
- The cutoff frequency is reached.

Verification of the design model mode. It is developed for the identification of the errors in the design models leading to the dimensional instability which are difficult to detect. The main idea of the method is that if the structure is dimensionally unstable, the problem (11.15.1) contains zero eigenvalues the number of which is equal to the number of the missing constraints the imposition of which would eliminate the dimensional instability, and the natural oscillation modes corresponding to these frequencies represent the mechanism motion modes. The analysis of these modes and their animation should help the user to identify where the constraints have to be imposed to make the design model dimensionally stable. The justification of the approach and the examples of its application are given in[18], [19].

In order to enable the verification mode it is necessary to create a dynamic loading – modal analysis, select the Lanczos method and specify the required number of the eigenpairs slightly higher than the expected number of the degrees of freedom of the mechanism (number of the missing constraints). Select the Lanczos method, verification mode and specify a small initial *negative* shift, for example -0.001 Hz, in the **Dynamics** tab of the **Calculation Parameters** dialog box. If the value of the initial shift is taken by default (as 0 Hz), the algorithm will substitute the value of -1 Hz. You should keep in mind that the closer to zero the value of the shift, the faster the convergence of the zero frequencies. On the other hand, if the value of the shift is too small, the matrix **K** – σ**M** is close to a special one in the case of a dimensionally unstable system, and the Lanczos method becomes numerically unstable.

If the system is dimensionally unstable, the matrix **K** is special and therefore it can not be factored. However, the matrix **K** – σ**M** is not special if the matrix **M** can be decomposed by the Cholesky decomposition. Therefore, small non-zero elements are added in the positions of the zero diagonal

elements of the mass matrix. For this reason, the eigenvalues obtained in the verification of the design model mode can differ from those obtained in other modes.

Example. The design model of a flat truss is given in the Figure 11.15-1. The missing diagonal is shown by a red dashed line. The oscillation mode corresponding to the zero frequency is shown in the Figure 11.15-2. The initial shift is assumed to be -0.001 Hz.

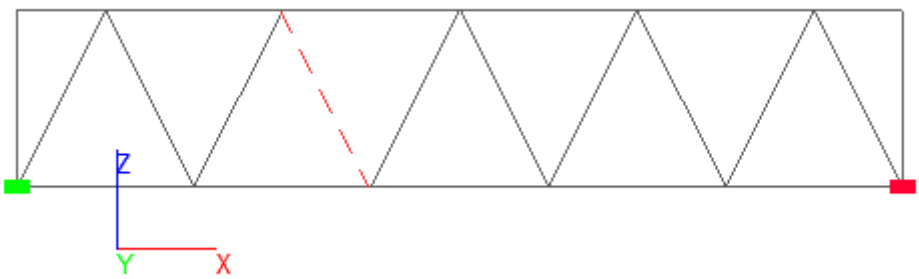


Figure 11.15-1. *Flat truss. The missing bar is shown by a dashed line*

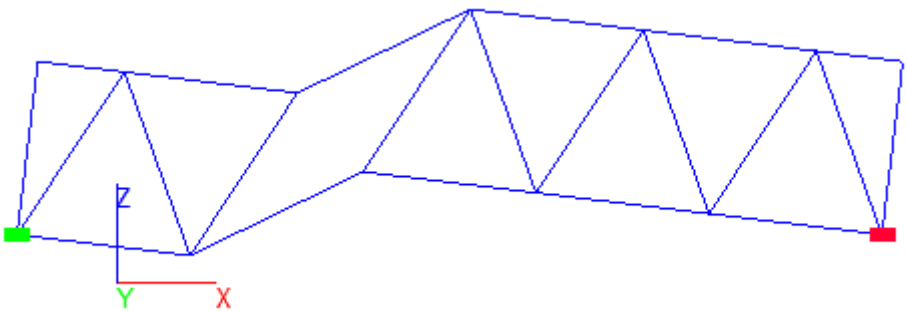


Figure 11.15-2. *Oscillation mode for the "zero" frequency (f* = -1.804e-006 *Hz)*

Information on the calculation. Full information on the calculation can be found in the file with the extension *.p04, which is saved in the SWORK directory. In particular, this file contains the data on the accuracy of the determination of the eigenpairs (see *Prec* column). In the verification of the design model mode the accuracy is calculated on the basis of the Kaniel-Saad criterion [18, § 12.4], in all other cases — it is the value equal to $\frac{\left\|\mathbf{v}_i - \lambda_i \mathbf{K}^{-1}\mathbf{M}\mathbf{v}_i\right\|_2}{\left\|\mathbf{v}_i\right\|_2}, \quad i \in [1, n]$.

It is recommended to view at least the accuracy of the obtained results after performing the calculation. The accuracy of the determination of an eigenpair is considered to be acceptable, if the *Prec* value does not exceed the value of the "accuracy of the solution of the eigenvalue problem (Lanczos method)" parameter (Fig. 11.7-1). Moreover, values of the modal masses m_x, m_y, m_z given in the table, as well as the sums of modal masses $\sum m_x$, $\sum m_y$, $\sum m_z$, calculated as the sums of modal masses from the first to the current natural modes, are very important information for the seismic analysis, and for the calculation for the wind pulsations, because they enable to determine which oscillation modes make a significant contribution to the dynamic response of the structure, and which ones do not.

References to Chapter 11

1. S. Fialko, *Application of the multifrontal method in SCAD for the analysis of large scale design models of buildings and structures,* BST PROJECT, No.9, 2005. – p. 61 – 63.

2. S. Fialko, *Application of the multifrontal method in SCAD for the analysis of large scale design models of buildings and structures,* CADmaster #30/5.2005 (additional) – p. 74 – 78.
3. S. Fialko, *A Sparse Shared-Memory Multifrontal Solver in SCAD Software, Proceedings of the International Multiconference on Computer Science and Information Technology*; October 20-22, 2008, Wisła, Poland, 3 (2008), ISSN 1896-7094, ISBN 978-83-60810-14-9, IEEE Catalog Number CFP0864E-CDR, pages 277–283. (http://www.proceedings2008.imcsit.org/pliks/47.pdf).
4. S. Fialko, *Direct methods for solving systems of linear equations in modern FEM-complexes* – M.: SCAD SOFT, 2009.
5. S. Fialko, *A block sparse shared-memory multifrontal finite element solver for problems of structural mechanics.* Computer Assisted Mechanics and Engineering Sciences, 16, 2009. p. 117 – 131.
6. S. Fialko, PARFES: *A method for solving finite element linear equations on multi-core computers.* Advances in Engineering software. v 40, 12, 2010, p. 1256 – 1265.
7. S. Fialko, *Direct Parallel FEM Solver PARFES for Multi-Core Computers.* III International Scientific and Practical Conference "Theory and Practice of Analysis of Buildings, Structures and Structural Elements. Analytical and Numerical Methods." November 17, 2010, Russia, Moscow, MGSU, p. 496 – 508.
8. S. Fialko, *Parallel Finite Element Solver for Multi-Core Computers.* Federated Conference on Computer Science and Information Systems, September 9–12, 2012. Wrocław, Poland. IEEE Xplore Digital Library, 978-83-60810-51-4, IEEE Catalog Number CFP1285N-USB. p. 525 – 532. URL: http://fedcsis.org/proceedings/fedcsis2012/pliks/101.pdf .
9. S. Fialko, *Application of AVX (Advanced Vector Extensions) for Improved Performance of the PARFES – Finite Element Parallel Direct Solver*. Federated Conference on Computer Science and Information Systems, September 8–11, 2013. Kraków, Poland. IEEE Xplore Digital Library, p. 447 – 454. https://fedcsis.org/proceedings/2013/pliks/98.pdf .
10. S. Fialko, *Parallel solvers in finite element software for multi-core shared memory computers.* Strength of Materials and Theory of Structures. 2015. No. 94, p. 155 – 171. URL: http://opir.knuba.edu.ua/files/zbirnyk-94/13-94.pdf.
11. S. Fialko, *Parallel direct solver for solving systems of linear equations resulting from finite element method on multi-core desktops and workstations*. Computers and Mathematics with Applications 70 (2015) 2968–2987. doi:10.1016/j.camwa.2015.10.009.
12. S. Fialko, *Methods for solving large-scale problems of structural mechanics using multi-core computers*. Civil Engineering Journal CEJ/2013 #5(40), p. 116 - 124. URL: http://www.engstroy.spb.ru/index_2013_05/fialko.html .
13. S. Fialko, *Iterative methods for solving large-scale problems of structural mechanics using multi-core computers*. Archives of civil and mechanical engineering (ACME), 14, 2014, p. 190 – 203.
14. A. George, J.W.H. Liu, *The evolution of the minimum degree ordering algorithm*, SIAM Rev. 31 (March) (1989) 1–19.
15. S. Pissanetski, *Sparse Matrix Technology*. — M.: Mir, 1988. —410 p.
16. S. Fialko, *Realization of block Lanczos method with shifts in SCAD software applying to seismic analysis of structures*. CADmaster #40/5.2007. (additional). – p. 102 – 105.
17. S. Fialko, *Efficient methods for solving the generalized algebraic eigenvalue problem. In the book* Perelmuter A.V., Slivker V.I. Design models *of* structures *and a* possibility of their analysis. – *Second edition, Kiev, 2002, p. 570-597.*
18. B. Parlett, *Symmetric eigenvalue problem. Numerical methods, M. Mir, 1983.*
19. S. Yu. Fialko, E. Z. Kriksunov and V. S. Karpilovskyy, *A block Lanczos method with spectral transformations for natural vibrations and seismic analysis of large structures in SCAD software,* in Proc. CMM-2003 – Computer Methods in Mechanics, Gliwice, Poland, 2003, pp. 129 —130.
20. S. Yu. Fialko, *Modular multi-level method for solving finite element problems of structural mechanics.* Habilitation Thesis. Kiev, KNUCA, 2004.
21. M. Naumov, *"Parallel Solution of Sparse Triangular Linear Systems in the Preconditioned Iterative Methods on the GPU",* https://research.nvidia.com/publication/parallelsolution-sparse-triangular-linear-systems-preconditioned-iterativemethods-gpu

22. S.Yu. Fialko, F.Zeglen*, Preconditioned Conjugate Gradient Method for Solution of Large Finite Element Problems on CPU and GPU,* Journal of Telecommunications and Information Technology, 2016, No. 2, 26-33.
23. S.Yu. Fialko, Application of rigid links in structural design models, International Journal for Computational Civil and Structural Engineering , 2017, 13 (3), 119-137.

12. Graphical Analysis of the Stress Strain State

Graphical analysis of the results of the calculation is performed in the same environment as the preparation of the initial data. Only the toolbar changes (Fig. 12-1) and new display filters appear.

The following types of graphical representation of results are provided in this mode: a deformed state of the structure, color markers, and isofields (for plate elements) of displacements, force diagrams in bar elements, stress fields for plate and solid elements. Moreover, the results of work of different postprocessors are displayed, groups of nodes and elements are created, etc.

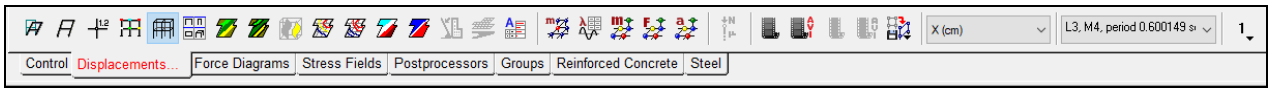


Figure 12-1. *Toolbar of the mode of the graphical analysis of the calculation results*

Different toolbars are provided for performing these operations. You can switch to them using the following tabs:

- **Control** — access to the data, printing, etc.;
- **Displacements** — analysis of the deformed state;
- **Force Diagrams** — analysis of forces in bar elements;
- **Stress Fields** — analysis of stresses and forces in plate and solid elements;
- **Postprocessors** — analysis of the results of postprocessing;
- **Groups** — generation of the groups of nodes and elements, in particular for transferring the data to postprocessors;
- **Reinforced Concrete** — check and selection of reinforcement in elements of reinforced concrete structures;
- **Steel** — check and selection of sections in elements of steel structures.

12.1. General Principles of Controlling the Display of Results

Before considering the operations of displaying the results of the calculation, let's first consider the rules of their implementation and the display controls.

The following procedure is recommended for the graphical analysis of the results:

- select the respective loading from the list:
  - for static loadings – number of the considered loading or combination of loadings;
  - for dynamic loadings – number of the oscillation mode;
  - for integration in time – time of the analysis;
- select the type of the analyzed information in the list of factors;
- click the button displaying the type of representation of the information.

Results obtained in the process of solving the problem can be analyzed in several windows with different fragments of the design model. The windows can simultaneously display the results of the calculation for different factors and different fragments of the model. Color scales of one problem belonging to different windows are displayed simultaneously, which enables to compare the information they contain. Initialization of a new window is performed with the help of the **New Window** operation in the ***Window*** menu (see Sec. 4.1 for more details).

A description of some display controls common for all operations of the analysis of results is given below.

Selecting Loading

Number of loading, combination of loads or other results of the calculation is selected from the **Select loading** list (Fig. 12.1-1) in the **Displacements**, **Force Diagrams**, **Stress Fields** toolbars. The following designations are used for the identification of the output information:

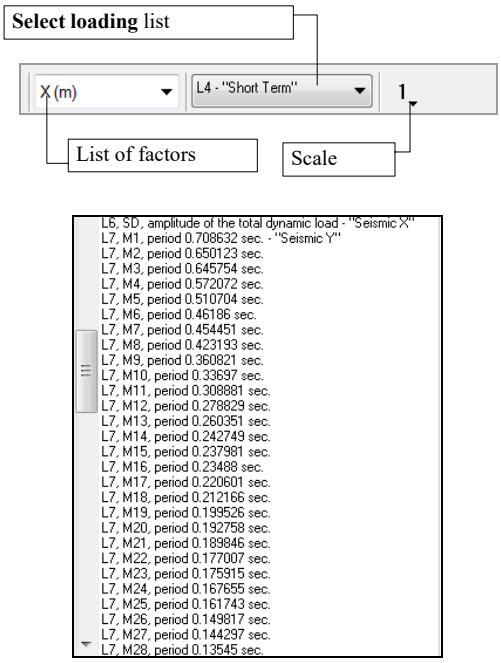


Figure 12.1-1. *The* **Select loading** *list*

- **L** –index of the loading followed by the number of the loading (Fig. 12.1-1). Number of the mode and the value of the period are additionally output for dynamic loadings at the expansion in oscillation modes. For the residual modes the following notations are used **+MX,+MY, +MZ**. Name of the loading is specified only in the line with the value of the period of oscillations of the first mode;
- **LS** – amplitude from the static load component (for example, wind);
- **SD** – amplitude from the total dynamic load;
- **LS+SD** – amplitude from the static (for example, wind) and total dynamic loads;
- **RD** – real component of each action at the calculation for harmonic oscillations;
- **RI** – complex component of each action at the calculation for harmonic oscillations;
- **S1**, **S2**…– total amplitude from the real and complex components depending on the number of actions;
- **LS+S1, LS+S2...** – amplitude from the static load component and *i*-th total amplitude from the real and complex components depending on the number of actions;
- **T** – savepoint number and the time from the beginning of the process at the direct integration of the equations of motion (see Chapter 15);
- **C** – index of the combination of loadings followed by the number of the combination and its formula;
- **M** – index of natural oscillation modes followed by the number of the mode and period of oscillations;
- **MX, MY, ...** – results for a 6-component action at the accelerogram based analysis;
- **S** – index of a buckling mode for loadings;
- **SC** – index of a buckling mode for combinations of loadings.

Peculiarities of the identification of loadings for the nonlinear analysis, variations of models and erection are described in the respective sections.

Right-clicking on the **Select Loadings** list invokes a dialog box where you can set and hide certain types of data in the list, for example, hide oscillation modes, combinations of loadings etc. If the list has been thus modified, an additional indicator will appear on the drop-down list arrow showing that the standard settings have been changed (similarly to the display filters).

Selecting the Considered Factor

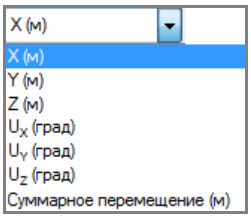


Figure 12.1-2. *List for selecting the directions of displacements*

The considered factor is selected from the list in the **Displacements**, **Force Diagrams, Stress Fields**, and **Postprocessors** toolbars (Fig. 12.1-2).

Changing the number of the loading or the type of the considered information can take some time, which largely depends on the dimension of the problem.

It should be noted that the operating conditions of a structure can lead to zero values of some factors.

For example, there is no displacement in direction of the Y axis for a truss system in the XOZ plane.

If the value of a factor in the whole design model is small, it is necessary to increase the number of significant digits using the **Units of Measurement of Results** tab of the **Parameters** dialog box.

Scale of Display

When displaying a deformed state of the model and force diagrams you can change the scale of the display of displacements or diagrams. The scale can be selected in the **Displacement scale factor** and **Diagram scale factor** lists.

Displaying Markers, Isolines, Isofields, and Isosurfaces

Displacements and force factors for plate and solid elements can be output as isolines or isofields, and also as color markers in nodes (for displacements) and in the centers of elements (for forces and stresses). Moreover, isosurfaces can be displayed for solid elements. The type of display can be selected with the help of buttons in the respective toolbars:

— display isofields;

— display isofields and isolines;

— display isolines;

— display isolines with a constant step;

— display isosurfaces;

— color display of stresses;

— color indication of the displacement value in nodes;

— display isofields with a constant step;

— display isofields with the selected value.

If the values of the considered factor have to be displayed on the diagrams, isolines or isofields, use the operation —**Digitization** (Sec. 10). The following values of the factor are displayed on the model in the result of performing this operation: for force diagrams — maximum absolute value of the force in the element, for displacements — value in the nodes of the model, and for stresses — values in the nodes and in the centers of finite elements (depending on the settings performed in the **Environment Settings** section of the ***Settings*** menu). The **Font Setting** tab of the mulit-tab **Parameters** dialog box enables to change the font, color and size of numbers.

Numerical values of factors can be displayed only for the selected nodes and elements (Fig. 12.1-3). The **Select objects for displaying information** operation in the **Control the Selection** toolbar is used to select such objects. In order to display the values only for the selected nodes and elements check the **Display information on the model only for the selected elements and nodes** checkbox in the **General Settings of the Display Filters** tab of the **Set the Information Display Filters** dialog box or activate an

equivalent operation in the **Visualization of Results** toolbar. If you simultaneously click the buttons , in the **Visualization of Results** toolbar and the button in the **Control the Selection** toolbar, every time you place a cursor over the model with the display of the results of the calculation, the numerical value of the considered factor of the selected object will appear.

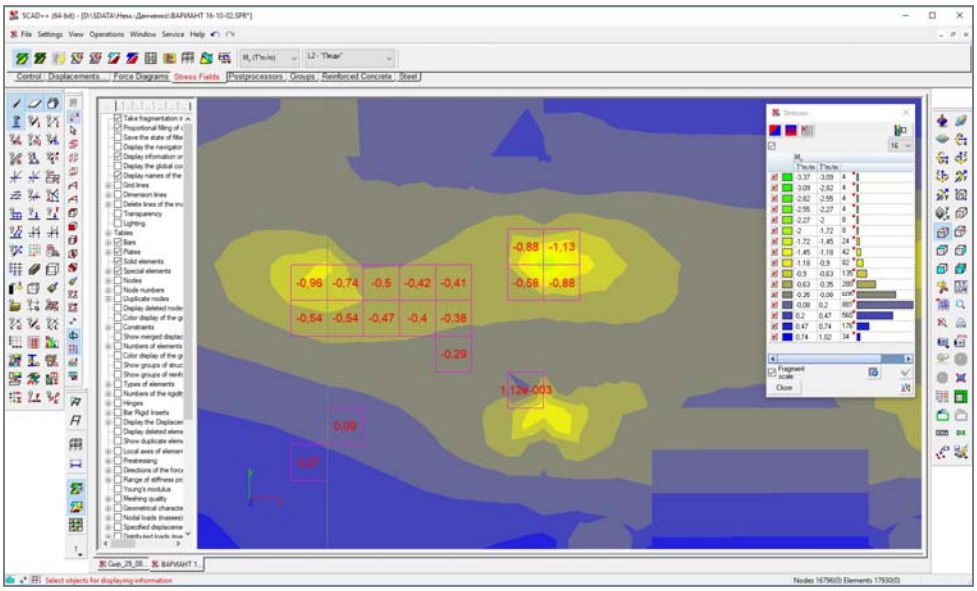


Figure 12.1-3. *Selective digitization of isofields*

Output of Isolines and Isofields with a Constant Step of the Considered Factor

These operations enable to output the isofields and isolines, and the number of isolines is determined depending on the specified step.

Positive and negative values of the factor are displayed on isolines and isofields in two different colors (Fig. 12.1-4), and the range of values of the same sign from minimum to maximum is given in the form of fields (lines), which differ by the intensity of the selected color (gradient scale). Ranges of the values of the factor are displayed in the color scale shown in Fig. 12.1-5.

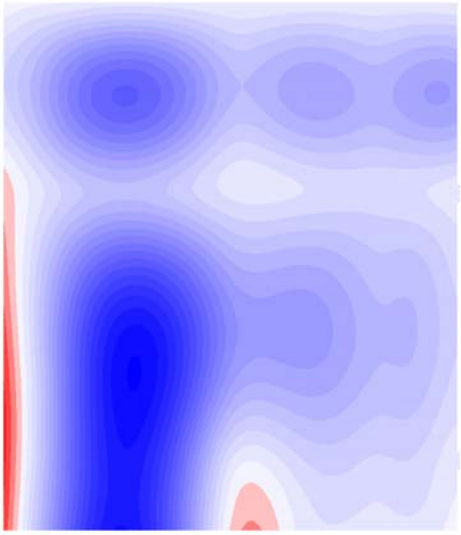


Figure 12.1-4. *Display of isofields on the design model*

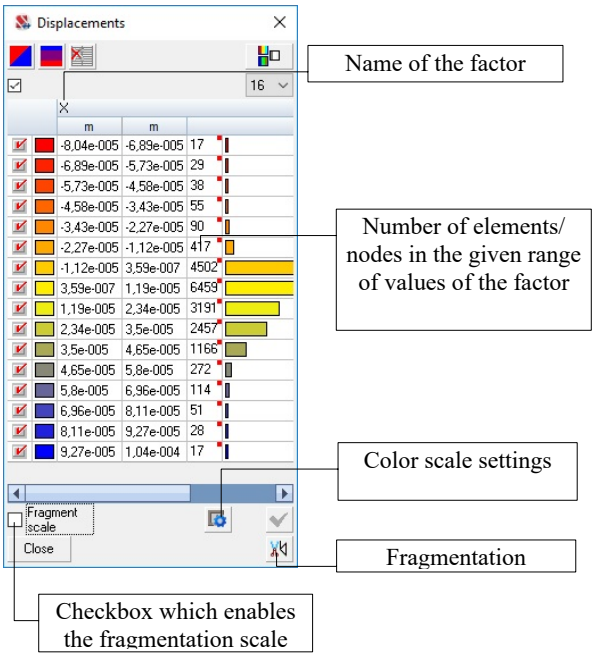


Figure 12.1-5. *The* **Displacements** *color scale*

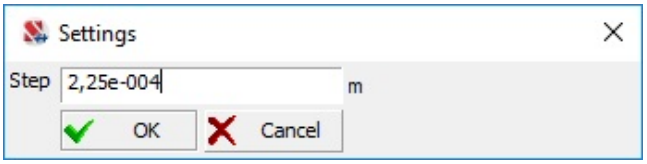


Figure 12.1-6. *The* **Settings** *dialog box*

The step of isolines is specified in the **Setting** dialog box (Fig. 12.1-6). The step equal to 1/10 of the maximum value of the factor is selected by default. Isolines and isofields with a new step are output after closing the dialog box by clicking the **OK** button.

Units of Measurement

When analyzing the displacements and force factors you can change the units of measurement, for example, display the results of displacements not in meters, but in inches or in millimeters.

The **Settings** operation — enables to do it. It is invoked from the respective menu or from the **Control** toolbar. The units of measurement of the results of calculation are specified in the respective tab of the **Parameters** dialog box (see Sec. 34.1) or with the help of the **Express Setting of the Units of Measurement** tool (see Sec. 34.2).

Displaying the Design Model

The **Display design model** button in the toolbars of the **Graphical analysis of the calculation results** mode enables to "erase" the graphical representation of the results from the current model. Only the design model or its fragment is displayed in the result. All active operations of the visualization of results, for example, color scales, will be disabled.

12.2. Analysis of Displacements

Analysis of the deformed state of a model is performed with the help of controls in the **Displacements** toolbar (Fig. 12.2-1).

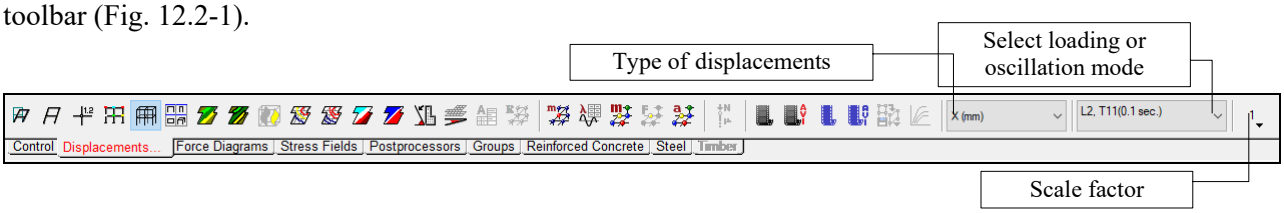


Figure 12.2-1. *Toolbar of the mode of the analysis of displacements*

The **Displacements** toolbar contains buttons for invoking the following operations:

— display the initial model together with the deformed one (Fig. 12.2-2. a);

— display the deformed model (Fig. 12.2-2, b);

— output values of displacements in nodes;

— color indication of the displacement value in nodes;

— display design model;

— display deformations on projections;

— display isofields of displacements (Fig. 12.2-2. c);

— display isofields and isolines of displacements;

— display isosurfaces of displacements in solid elements (Fig. 12.2-2, d);

— display isolines of displacements with a constant step;

— display isolines of displacements;

— display isofields of displacements with a constant step;

— display isofields of displacements with the selected value;

— determine the tilt of the building;

— determine the tilt of the foundation slab;

— potential energy and the total load;

— centers of rigidity of the storyes;

— determine the center of mass;

— frequencies and periods of oscillations;

— weights of nodal masses;

— dynamic loads by oscillation modes;

— accelerations;

— unsupported lengths of bar elements;

— animation of displacements;

— save the video with the animation;

— animation of system state changes;

— save the video with the animation of system state changes;

— save the deformed model;

— time history (see section 15.4).

The toolbar also contains the following lists: **Type of displacements**, **Select loading** and **Displacement scale factor**.

The results of the calculation of displacements are graphically displayed as isofields, isolines, isosurfaces, and markers of linear displacements and rotations about the axes of the global coordinate system, as well as the total displacements calculated as $\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$.

You can change the scale of displaying the displacements when outputting the deformed model, generating videos and animations of displacements. The scale factor (Fig. 12.2-1) is selected from the suggested list of values.

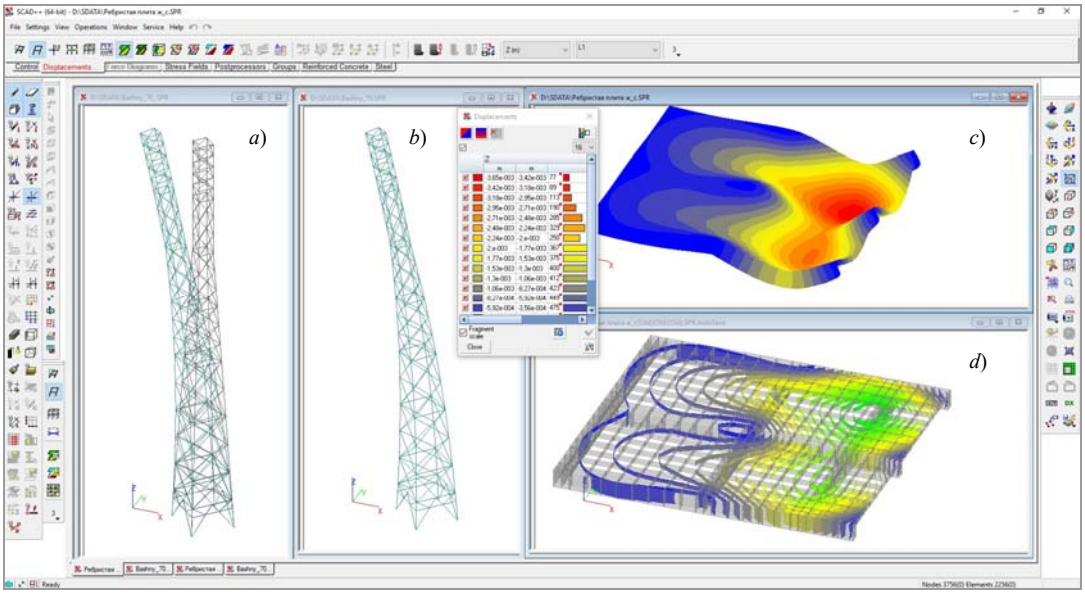


Figure 12.2-2. *Visualization of the results of the calculation of displacements*
a) deformed model together with the initial one *b) deformed model*
c) isofields of displacements on the deformed model *d) isosurfaces of displacements*

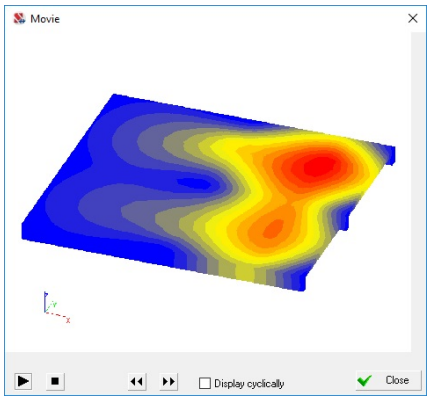


Figure 12.2-3. *The* **Movie** *dialog box*

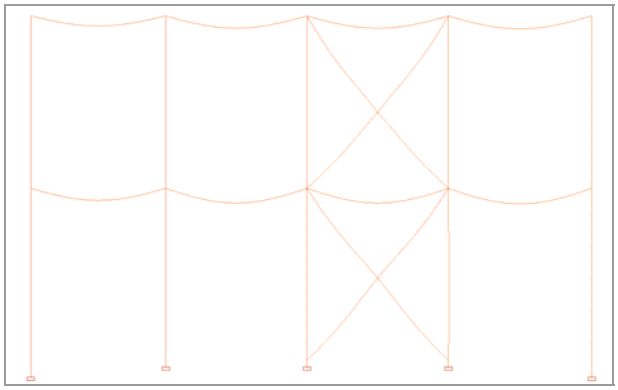


Figure 12.2-4. *Deformed model with the enabled filter of the display of deflections*

Animation and generation of videos can be performed for the whole design model or for its fragment. Videos are saved in a file with the **.avi** extension. These files are placed in the directory with the files of the results or any other directory and can be viewed with the help of standard Windows tools for the visualization of videos.

A standard **Save As** dialog box appears after finishing the function of generating a video, where you have to specify the file name. The **Movie** dialog box (Fig. 12.2-3) opens after saving the file, where you can view the created video. The **Environment Settings** operation from the ***Settings*** menu enables to change the default parameters of the animation mode, for example, increase or decrease the number of repetition cycles.

A deformed model accounting for the displacements in intermediate points of the bar can be displayed for bar elements (Fig. 12.2-4). The **Display deflections in bars** filter enables to do it (see Sec. 10.3). If this filter is enabled, the function of animation is also performed with the visualization of deflections. You can use the **Deflection diagrams** operation in the **Information on the Element** dialog box to analyze the deflections in one element (Sec. 10.2).

Animation of Stepwise and Time Processes

These operations enable to animate the processes of changing the model state for the given time interval (direct integration of the equations of motion), for the specified portion of the stepwise process (nonlinear problems) or for the change of erection stages (for accumulated loading). In the case of direct integration, it is assumed that the state changes are analyzed for the entire time interval. In the case of the stepwise process, in addition to the analysis for the entire step interval, it is possible to specify a subset of steps corresponding to the additional loading by the selected linear loadings.

If nonlinear loadings are considered and the **Animation** operation is invoked, the **Animation of the Nonlinear Loading** dialog box appears (Fig. 12.2-5). It contains a list of loadings used when modeling nonlinear loads (specified for the given nonlinear loading in the **Control the Stepwise Process** dialog box). Use the checkboxes on the right from the names of the loadings to select the loadings the set of steps of which will be taken into account in the animation process.

If the animation is performed with the color scale visualization in the background, then when the step of the nonlinear process or time changes (in the case of direct integration), the minimax values will change as well. Therefore, only the number and colors of the ranges, the step (for the scales with a constant step), the threshold, and the excluded range for the scales with the selected value will be inherited for the animation from the color scale settings. The boundaries of the image (the shift position relative to the window) do not change.

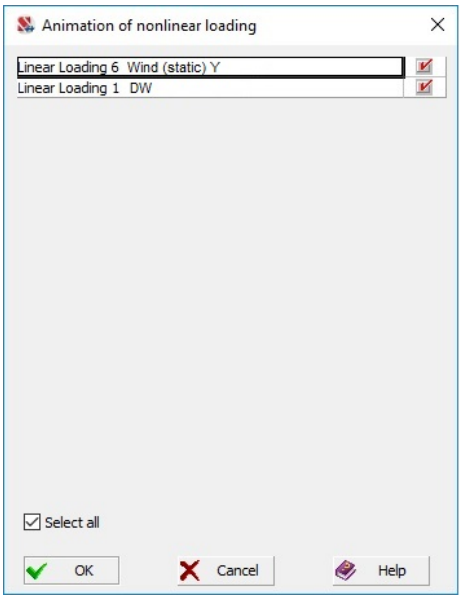


Figure 12.2-5. *The* **Animation of the Nonlinear Loading** *dialog box*

Determination of the Coordinates of the Center of Mass

This operation enables to determine the X, Y and Z coordinates of the center of the specified masses for each dynamic loading. The **Determine the Center of Mass** dialog box (Fig. 12.2-6) appears on the screen after invoking this operation, which provides a table with the list of dynamic loadings and coordinates of the center of mass.

If you select a line with the name of the loading, two lines with the center of mass of the given loading at their intersection will be displayed on the design model (Fig. 12.2-7). Both the specified masses and those obtained by transforming the static loads are taken into account when determining the center of mass.

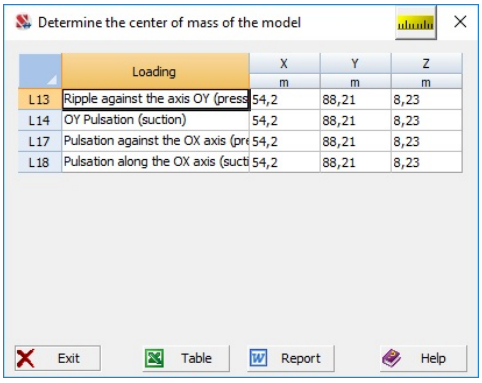


Figure 12.2-6. *The* **Determine the Center of Mass** *dialog box*

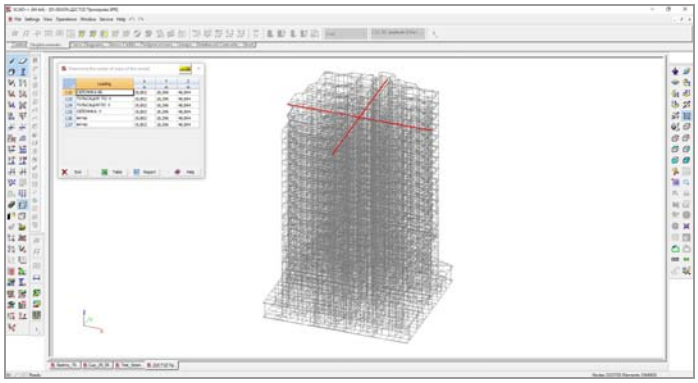


Figure 12.2-7. *Design model with the displayed center of mass*

If there are selected nodes and/or elements, invoking this operation drops down a menu where you can select one of the following options:

- determine the center of mass of the model;
- determine the center of mass of the selected nodes;
- determine the center of mass of the selected elements.

In the latter case, the center of mass of all nodes belonging to the selected elements will be determined.

Determination of the Tilt of the Structure

In order to determine the tilt of the structure , it is necessary to specify a group or groups of nodes the displacements of which define the value of the tilt. The **Determine the Tilt of the Structure** dialog box (Fig. 12.2-8) appears after invoking the operation.

It is necessary to select the name of the group from the **Groups of nodes** list in this dialog box, and the value of the tilt will appear in the table for each loading.

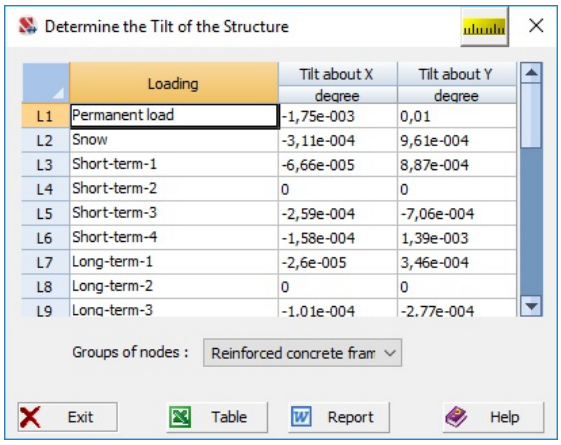


Figure 12.2-8. *The* **Determine the Tilt of the Structure** *dialog box*

A tilt is calculated according to the following algorithm:

- a line which is the best for approximating the position of nodes of the group in their initial coordinates is determined for the selected group of nodes;
- a similar operation is performed for the same group taking into account the displacements of nodes in each loading;
- angles between the projections of the obtained lines on the XOZ, YOZ planes are the tilt along X and tilt along Y.

The **Report** and **Table** buttons enable to obtain a report document in the MS Word or MS Excel format respectively.

Determination of the Tilt of the Foundation Slab

In order to determine the tilt of the foundation, it is necessary to specify a group or groups of nodes the displacements of which define the value of the tilt.

The **Determine the Tilt of the Foundation Slab** dialog box (Fig. 12.2-9) appears after invoking the operation. It is necessary to select the name of the group from the **Groups of nodes** list in this dialog box, and the value of the tilt will appear in the table for each loading.

A tilt is calculated according to the following algorithm:

- a plane which is the best for approximating the position of nodes of the group in their initial coordinates is determined for the selected group of nodes;
- a similar operation is performed for the same group taking into account the displacements of nodes in each loading;
- the angle between the obtained planes is the tilt.

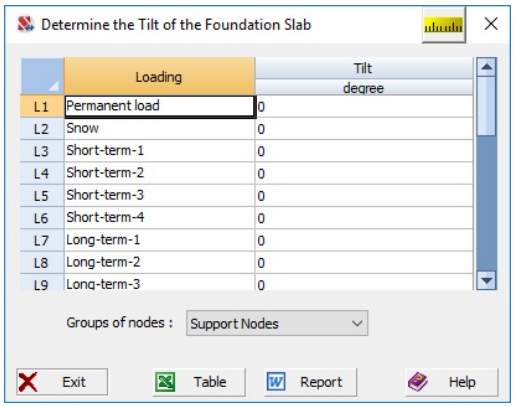
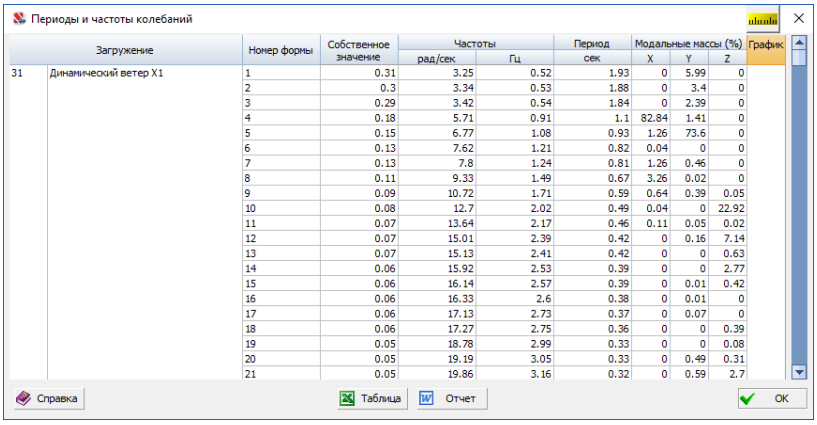


Figure 12.2-9. *The* **Determine the Tilt of the Foundation Slab** *dialog box*

The **Report** and **Table** buttons enable to obtain a report document in the MS Word or MS Excel format respectively.

Periods and Frequencies of Oscillations

This operation enables to obtain a table with the list of characteristics of dynamic loadings. In the result a table with eigenvalues, frequencies, periods of oscillations and percentage of modal masses for all dynamic loadings and for each oscillation mode specified in the initial data is output in the **Periods and Frequencies of Oscillations** dialog box (Fig. 12.2-10).



Загружение		Номер формы	Собственное значение	Частоты рад/сек	Частоты Гц	Период сек	Модальные массы (%) X	Y	Z	График
31	Динамический ветер X1	1	0.31	3.25	0.52	1.93	0	5.99	0	
		2	0.3	3.34	0.53	1.88	0	3.4	0	
		3	0.29	3.42	0.54	1.84	0	2.39	0	
		4	0.18	5.71	0.91	1.1	82.84	1.41	0	
		5	0.15	6.77	1.08	0.93	1.26	73.6	0	
		6	0.13	7.62	1.21	0.82	0.04	0	0	
		7	0.13	7.8	1.24	0.81	1.26	0.46	0	
		8	0.11	9.33	1.49	0.67	3.26	0.02	0	
		9	0.09	10.72	1.71	0.59	0.64	0.39	0.05	
		10	0.08	12.7	2.02	0.49	0.04	0	22.92	
		11	0.07	13.64	2.17	0.46	0.11	0.05	0.02	
		12	0.07	15.01	2.39	0.42	0	0.16	7.14	
		13	0.07	15.13	2.41	0.42	0	0	0.63	
		14	0.06	15.92	2.53	0.39	0	0	2.77	
		15	0.06	16.14	2.57	0.39	0	0.01	0.42	
		16	0.06	16.33	2.6	0.38	0	0.01	0	
		17	0.06	17.13	2.73	0.37	0	0.07	0	
		18	0.06	17.27	2.75	0.36	0	0	0.39	
		19	0.05	18.78	2.99	0.33	0	0	0.08	
		20	0.05	19.19	3.05	0.33	0	0.49	0.31	
		21	0.05	19.86	3.16	0.32	0	0.59	2.7	

Figure 12.2-10. *The* **Periods and Frequencies of Oscillations** *dialog box*

The table also displays modal participation factors for each oscillation mode. In the case of seismic loadings the **Inertia Matrices** button enables to view the matrices of inertial characteristics of the model as a rigid body (with and without constraints), which are used in some soil-structure interaction models[1].

The **Report** and **Table** buttons enable to export the table to MS Excel or to generate an RTF-file by loading it into the associated program.

If more than one oscillation mode is calculated, the **Chart** button appears in the table, which enables to obtain a graph of the accumulation of the percentage of modal masses for different directions (Fig. 12.2-11).

[1] See, for example, A.G.Tyapin, *Seismic Analysis of the Soil-Structure Interaction*, M., ASV, 2013.

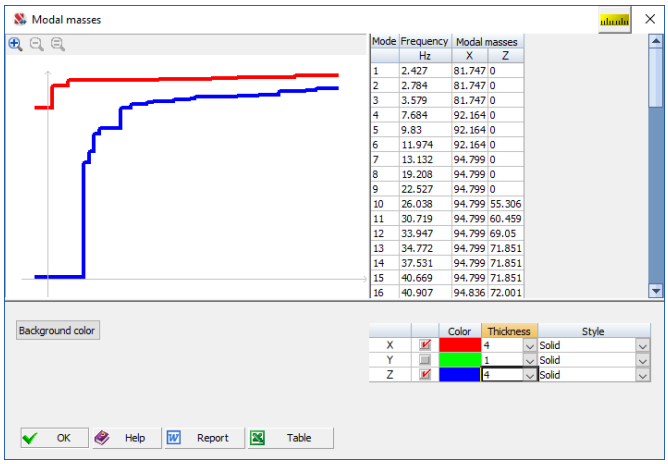


Figure 12.2-11. *The* **Accumulation of Modal Masses** *dialog box*

Weights of Nodal Masses

This operation enables to display the weights of nodal masses on the design model as color markers. A color scale of the weights of nodal masses (Fig. 12.2-12) specified by the user and obtained automatically by transforming the static loads into masses is output in the result.

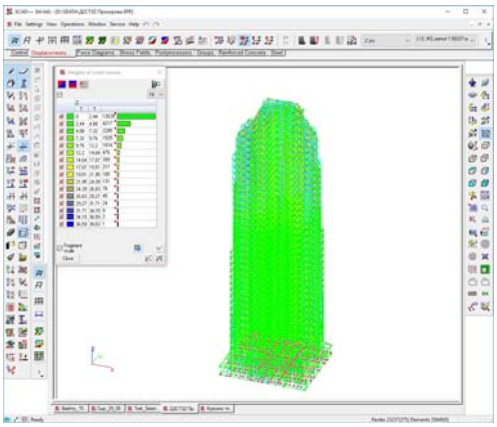


Figure 12.2-12. *Distribution of nodal masses*

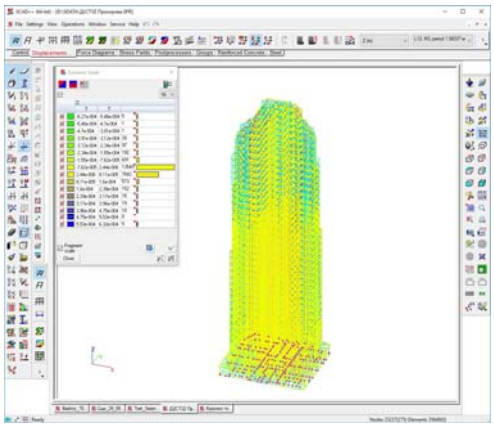


Figure 12.2-13. *Distribution of dynamic loads*

Dynamic loads by oscillation modes

This operation enables to display the distribution of dynamic loads on the design model as color markers. A color scale of dynamic loads for each oscillation mode (Fig. 12.2-13) is output in the result.

Accelerations

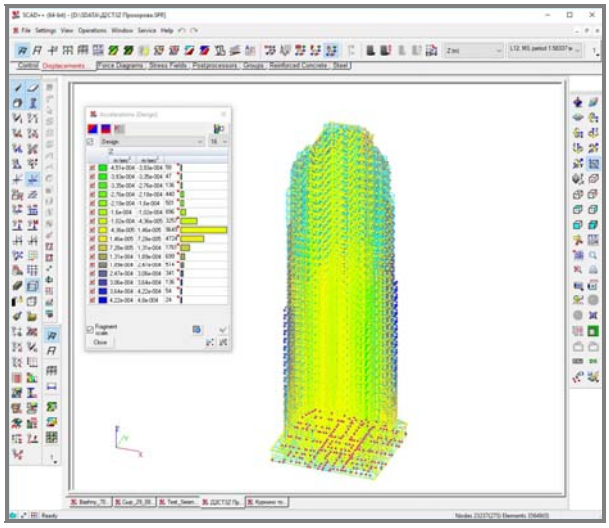


Figure 12.2-14. *Accelerations in nodes*

This operation enables to display as color markers the accelerations obtained in the result of the calculation in the nodes of the design model for each oscillation mode and for the total dynamic load (Fig. 12.2-14).

The drop-down list in the dialog box of the color scale enables to select the type of output values (design or characteristic). When the pulsation component of the wind load is calculated according to SP 20.13330, this list contains the **Dynamic comfort** item, which enables to analyze values equal to 70% of the characteristic accelerations (the limitations of these values are specified in SP 20.13330).

Unsupported Lengths of Bar Elements

This operation enables to display the unsupported lengths of bar elements as a color scale.

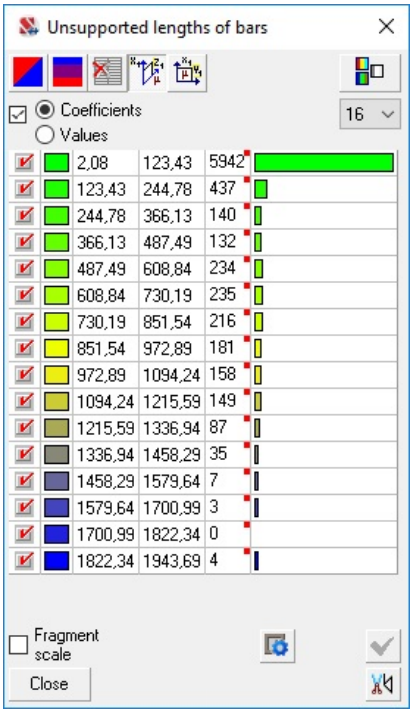


Figure 12.2-15. *Unsupported lengths of bars*

The dialog box of the color scale (Fig. 12.2-15) enables to select the plane of inertia of the bar (X_1OZ_1 or X_1OY_1), and the form of presentation — the values of the unsupported lengths or the unsupported length factors (ratio of the unsupported length to the length of the flexible part).

Saving the Deformed Model

This operation enables to generate a new design model on the basis of the deformed model obtained from the active (considered) loading.

A drop-down menu suggesting to create a deformed model or a deformed model with prestressing appears in the result. Coordinates of the nodes of the new model are generated according to the following rule $X_i + \Delta X_i$, $Y_i + \Delta Y_i$, $Z_i + \Delta Z_i$.

Saving the results of the calculation as a corrected design model of a structure can be performed in the following ways:

a) saving a deformed model in which the coordinates of the nodes are corrected by adding the displacements obtained in the calculation;

b) saving the internal forces and the deformed model;

c) saving the internal forces.

Variant “a” is especially useful when designing suspension systems and determining the geometry of the structure. Then starting from an initial geometry of the unstrained system and saving its deformed model obtained in the nonlinear calculation we can obtain a geometric image the system will have for example under the action of self-weight.

Variant “b” enables to save the deformed model and internal forces which is necessary when designing prestressed structures. In this case the results of the calculation of a system subjected to thermal or other kinematic actions are usually saved.

If an oscillation mode which has corresponding dynamic loads is selected, you can select the third item *Save the dynamic loads as a group of loads* in the menu. This group of loads can be used later to create static loadings and to perform, for example, the stability analysis at the action of the selected oscillation mode (you should keep in mind that the oscillation modes are alternating).

If the design model of the frame building is considered and the first buckling mode is selected, then this operation enables to save the deformed model for taking into account the initial imperfections in accordance with the recommendations of Eurocode 3 (see below).

The last item of the menu *Save weights of nodal masses as groups of loads* allows to create groups of nodal loads (for each dynamic loading) corresponding to the weights of the nodal masses.

Accounting for the Initial Imperfections

In accordance with the recommendations of Eurocode (EN 1993), when choosing the configuration of the design model that takes into account the initial imperfections, a deformed model corresponding to the first buckling mode (at the minimum values of the elastic critical load) can be adopted. The normalizing factor, which indicates the maximum possible displacement of the node, can be specified by the user. If the normalization is carried out in accordance with the recommendations of EN 1993, the value of the maximum displacement (Δ) is calculated by the formula

$$\Delta = \frac{H}{200}\sqrt{0.5\left(1+\frac{1}{m}\right)},$$

where H — height of the building, m — number of compressed columns on the floor.

The generation of the design model taking into account the initial imperfections is invoked in the **Displacements** section of the toolbar. The model can be saved on the disk according to rules similar to those for saving the deformed model with the help of the respective operation. It is necessary to perform the following actions to obtain a design model taking into account the initial imperfections:

- Perform the stability analysis with the determination of the first buckling mode;
- Switch to the graphical analysis of the calculation results;

- Select the line with the results of the calculation of the first buckling mode in the **Select loading** list of the **Displacements** toolbar;
- Invoke the **Save the deformed model** operation, and the **Calculation of the Initial Imperfections by the Buckling Mode** dialog box will appear (Fig. 12.2-16);
- Specify the absolute value of the maximum displacement of the node in the **Maximum displacement of nodes by the buckling mode** text field;
- If the calculation is performed taking into account the recommendations of **Eurocode 3**, the value of the maximum displacement can be calculated automatically according to the above formula (the **Calculate** button with the EU flag icon);
- After invoking the **Calculate** operation in the **Calculation of the Maximum Displacement** dialog box (Fig. 12.2-17), it is necessary to specify the height of the building and the number of compressed columns in the basement or on the first floor (the values from the design model are taken by default);
- Close the dialog box by clicking the **OK** button;
- The **Maximum displacement ...** text field (Fig. 12.2-16) will contain the displacement value, which is considered as the maximum one when the initial imperfections for the given model are taken into account;
- Closing the dialog box (Fig. 12.2-16) by clicking the **OK** button opens the standard **Save As …** dialog box, where you can specify the file name and save the design model in which the coordinates of the nodes are modified taking into account the initial imperfections.

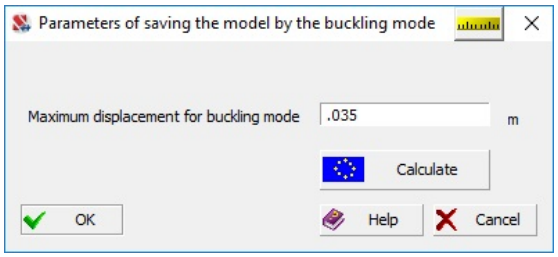


Figure 12.2-16. *The* **Parameters for Saving the Model by the Buckling Mode** *dialog box*

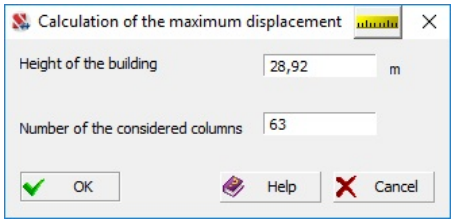


Figure 12.2-17. *The* **Calculation of the Maximum Displacement** *dialog box*

Special Processing of the Output Displacements

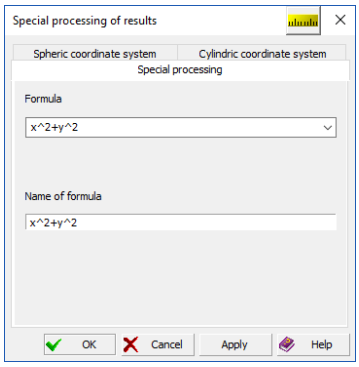


Figure 12.2-18. *The* **Special Processing of Results** *dialog box (displacements as a function)*

The calculated displacements can be output as color markers, isofields and isolines according to special rules, which include the option of obtaining the results in cylindrical or spherical coordinate systems, and as a function. The rules for outputting the displacements are set in the multi-tab **Special Processing of Results** dialog box, which is invoked by selecting the **As a function** option from the list of displacement directions.

The function $R = f\ (x,\ y,\ z)$ which will be used to process the calculated displacements can be specified in the **Special Processing** tab (Fig. 12.2-18) (variables *x, y, z* will be interpreted as displacement in the corresponding

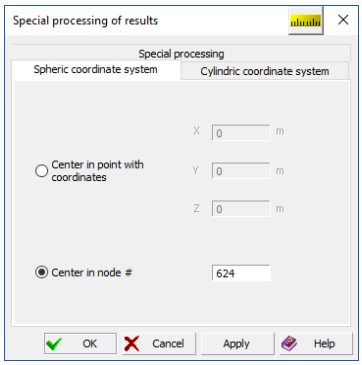


Figure 12.2-19. *The* **Special Processing of Results** *dialog box (displacements in a spherical coordinate system)*

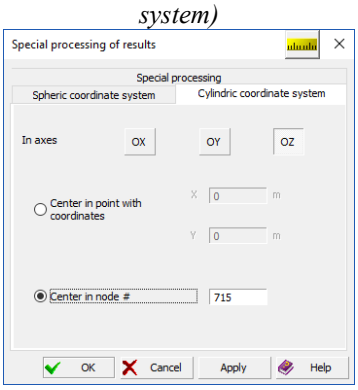


Figure 12.2-20. *The* **Special Processing of Results** *dialog box (displacements in a cylindrical coordinate system)*

directions). This function is saved automatically and can be used again in this and other problems. It can also be given a name. In order to specify a function, enter the expression in the **Function** list box and its name in the **Function name** text field. After closing the dialog box by clicking the **OK** button and selecting the type of display, the results will be displayed on the model. If you want to use this function again, you can simply select it from the list.

In order to display the displacements in a spherical coordinate system, specify the coordinates of its origin (the **Origin at the point with coordinates** radio button) or indicate the number of the node where the origin lies (the **Origin at the node No.** radio button) in the **Spherical Coordinate System** tab (Fig. 12.2-19).

In order to display the displacements in a cylindrical coordinate system, specify the axis of the global coordinate system parallel to the output axis (the **Parallel to the axis** buttons), as well as the point or node the output axis passes through (**Axis passes through the point with coordinates** and **Axis passes through the node** radio buttons) in the respective tab (Fig. 12.2-20).

After closing the dialog box by clicking the **OK** button and selecting the type of display, the results will be displayed on the model.

12.3. Analysis of Forces in Bars

Analysis of forces in bars is performed with the help of controls in the **Force Diagrams** toolbar (Fig. 12.3-1).

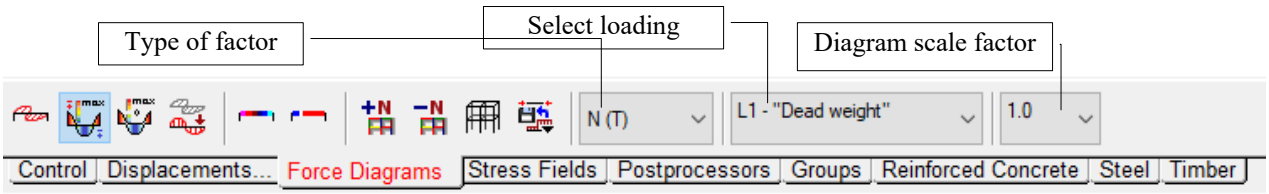


Figure 12.3-1. *Toolbar of the mode of the analysis of forces in bar elements*

This mode provides the following operations for displaying the results:

— force diagrams;

— color display of the diagrams of forces;

— color display of the diagrams of forces with the selected value;

— change the display axis of diagrams N and M_k;

— color display of the values of forces;

— color display of the values of forces with the selected value;

— color indication of the positive values of forces;

— color indication of the negative values of forces;

— display design model;

— save the model with prestressing.

The toolbar also contains the following lists: **Select type of force**, **Select loading**, and **Set the diagram scale factor**.

When outputting the diagrams you can change the scale of their display. The scale changes in accordance with the value of the scale factor selected from the suggested list of values.

The **Force diagrams** button in the **Information on the Element** dialog box (see Sec. 10.2) enables to analyze diagrams of all force factors of one element.

Force diagrams can be hatched. The activation and control of the operation of hatching are performed in the **Results** tab of the **Environment Settings** dialog box (Fig. 12.3-4), which is invoked from the ***Settings*** menu. If the hatch spacing (**Hatch spacing of the force diagrams**) is set equal to one, it corresponds to the maximum density of hatching. If the spacing is equal to 1.5 or 2, the density of hatching decreases by half or twice respectively.

Fig. 12.3-2, a provides an example of the display of diagrams on the design model when the **Hatch the force diagrams** checkbox is checked.

The filters button enables to display the maximum absolute numerical values of the considered factor.

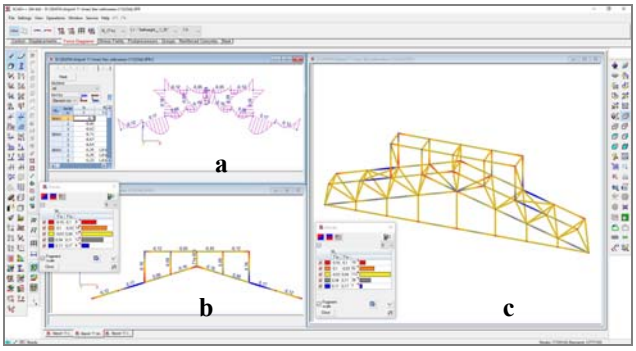


Figure 12.3-2. *Display of forces on the design model*

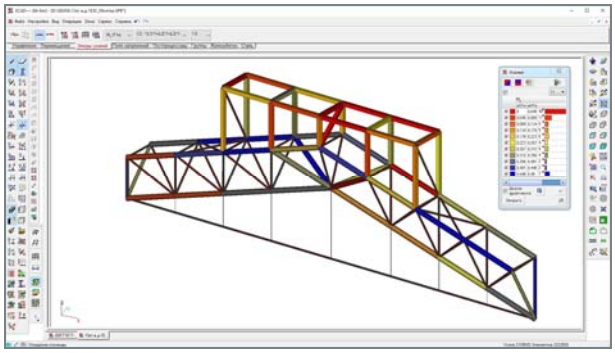


Figure 12.3-3. *Color scale of forces on the model with the visualization of the profiles of bars*

At the color indication of values of force factors (the button — for the indication of positive values and — for negative ones) the elements are displayed in a color corresponding to the maximum value of the factor in the element. Any button can be used for the indication of values of normal forces, since the coloring of elements of the model and the scale will simultaneously include both positive and negative values. If when using the considered operations you activate the filter of displaying the profiles of bar elements taking into account the removal of lines of the invisible contour, the visualization of the profile will be performed taking into account the color scale (Fig. 12.3-3).

The **Color display of the values of forces** operation— provides a gradient coloring of each element (Fig. 12.3-2, b) depending on the changes of the value of the considered factor along the element. This operation does not take into account the fact of activating the filter of displaying the profiles of bar elements taking into account the removal of lines of the invisible contour.

When the **Color display of the values of forces with the selected value** operation— is used, the elements of the model are colored in the shades of two colors. One of the colors is used for the elements or some of them for which the value of the factor is greater than the selected value, and the other color is used for the elements with the value of the factor less than this value. The closer it is to the selected value the less is the intensity of the color. Elements with the value of the force factor within the range of the selected

value are displayed in a third color (Fig. 12.3-2, c). The range of the selected value is specified in the **Settings** dialog box, which is invoked from the dialog box of the scale by clicking the **Color scale settings** button (Fig. 12.3-4).

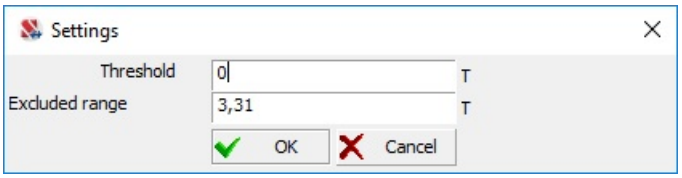


Figure 12.3-4. *The* **Settings** *dialog box*

Two values are specified in the **Settings** dialog box: **Threshold** — the value of the factor defining the middle of the selected range, and **Excluded range** — the value of the factor characterizing the width of the range. For example, if the threshold is set equal to 1, and the selected range is equal to 0.2, the values of the factor within the selected range will be from +1.2 to -1.2.

Operations (**Color display of force diagrams**) and (**Color display of force diagrams with selected value**) are similar to operations , , but they provide gradient coloring of force diagrams:

12.4. Analysis of Forces and Stresses in Plate Elements

Control of the Display of Forces and Stresses

The analysis of force factors in plate and solid elements is performed with the help of the operations in the **Stress Fields** toolbar (Fig. 12.4-1).

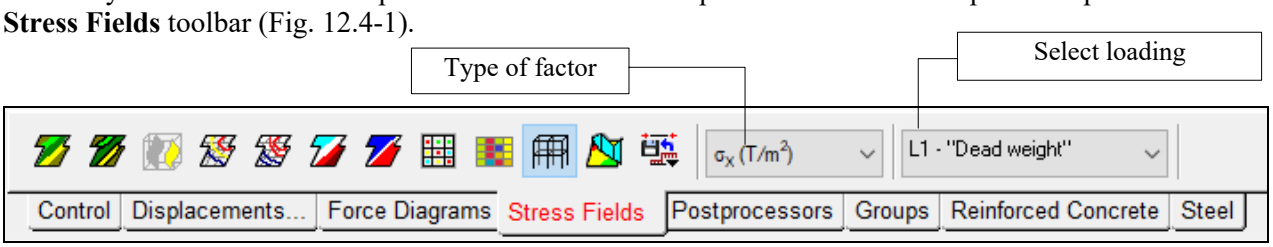


Figure 12.4-1. *Toolbar of the mode of the analysis of stresses in plates*

The toolbar of this mode includes the following operations for displaying the results:

— display isofields of stresses;

— display isofields and isolines of stresses;

— display isosurfaces of stresses in solid elements;

— display isolines of stresses;

— display isolines of stresses with a constant step;

— display isofields of stresses with a constant step;

— display isofields of stresses with the selected value;

— color display of stresses;

— color indication of stresses at the centers of elements;

— display design model;

— plot stress diagrams along a secant line;

— save the model with prestressing.

The toolbar also contains two lists for selecting the name of the force factor and the number of loading (Fig. 12.4-1). If the design model has multilayer finite elements, the toolbar will contain one more drop-down list for selecting the number of the layer the stresses in which have to be visualized.

The filters button enables to display the numerical values of the force factor on isofields and isolines. The values can be displayed in the centers of elements and in nodes. In order to obtain the values in nodes it is necessary to activate the respective option in the **Environment Settings** dialog box (it can be invoked from the ***Settings*** menu).

Isolines and isofields are output according to the same rules as in the analysis of displacements in plate elements (Fig. 12.4-2).

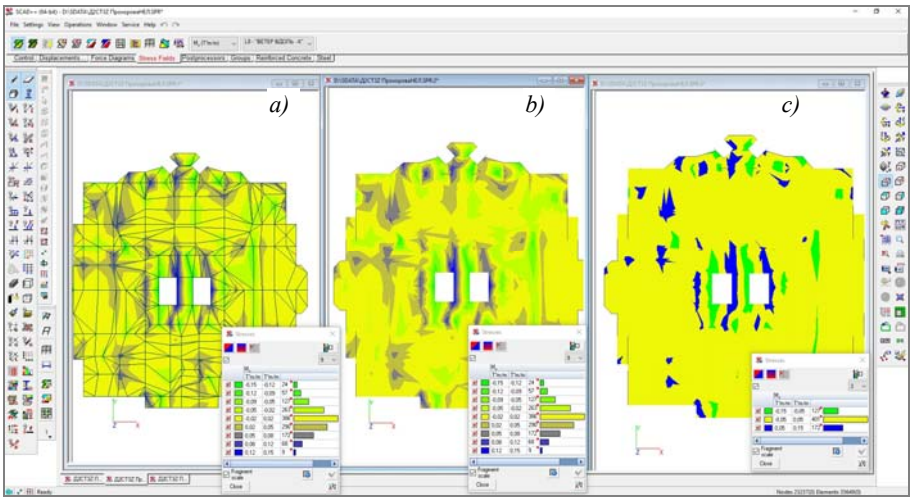


Figure 12.4-2. *Display of isofields and isolines of stresses on the design model a) with the display of the FE mesh; b) without the display of the FE mesh; c) with the selected value*

When analyzing the numerical values of force factors given in the color scale of isofields and isolines, it is necessary to take into account that their default values are given allowing for the averaged values in nodes from all elements coming into this node. Moreover, depending on the settings of the filter of digitization (see Fig. 12.4-8), these values can be based on the minimum or maximum values of the factor in the node, and also show the residual. It should be noted that the values of force factors in the nodes of the considered element are given in the information on the element without averaging and can differ from the values on isofields.

Generation of the design combinations of forces is performed on the basis of the values of factors obtained in the centers of elements. In order to estimate these particular values, it is necessary to use the mode of displaying with the help of color markers — . Moreover, the lines of the invisible contour are not deleted in this mode, which enables to determine the points of the design model with extreme values of the factors more accurately.

The software enables to generate isolines and isofields of stresses for the upper and lower faces of planar elements. These stresses are designated as follows in the list of force factors in the **Stress Fields** toolbar:

$\sigma_{x\downarrow}$, $\sigma_{y\downarrow}$ — stresses on the lower face of the elements in the X_1 and Y_1 directions respectively;

$\sigma_{x\uparrow}$, $\sigma_{y\uparrow}$ — stresses on the upper face of the elements in the X_1 and Y_1 directions respectively.

Selective Marking of Force Factors

When displaying the results of the calculation of the force factors in plate elements they can be selectively digitized (see Sec. 12.1).

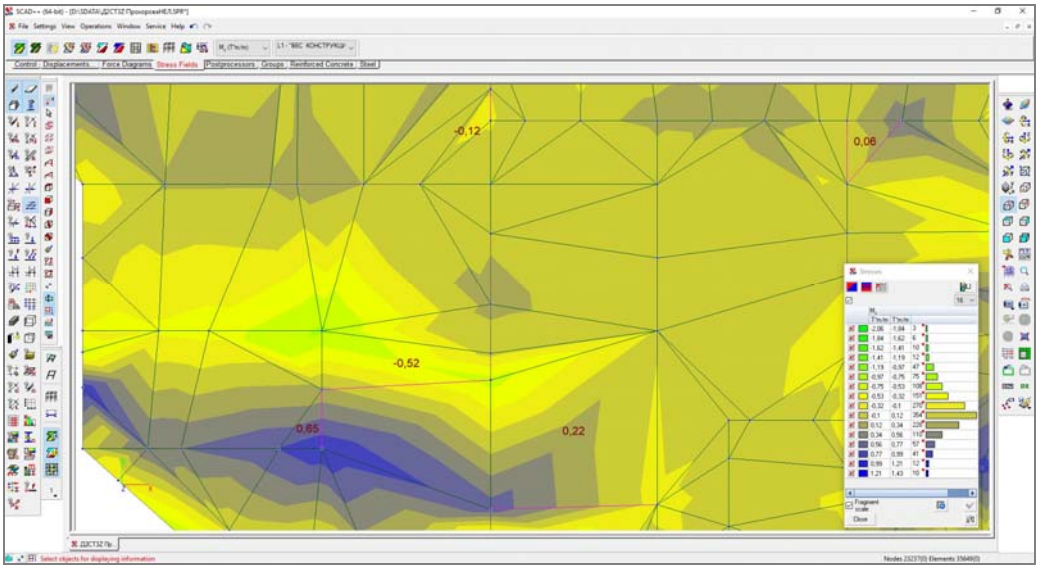


Figure 12.4-3. *Example of the selective digitization of force factors*

Plotting Stress Diagrams

This operation enables to plot diagrams of force factors along the specified line (secant line) in the shell, plate and deep beam elements.

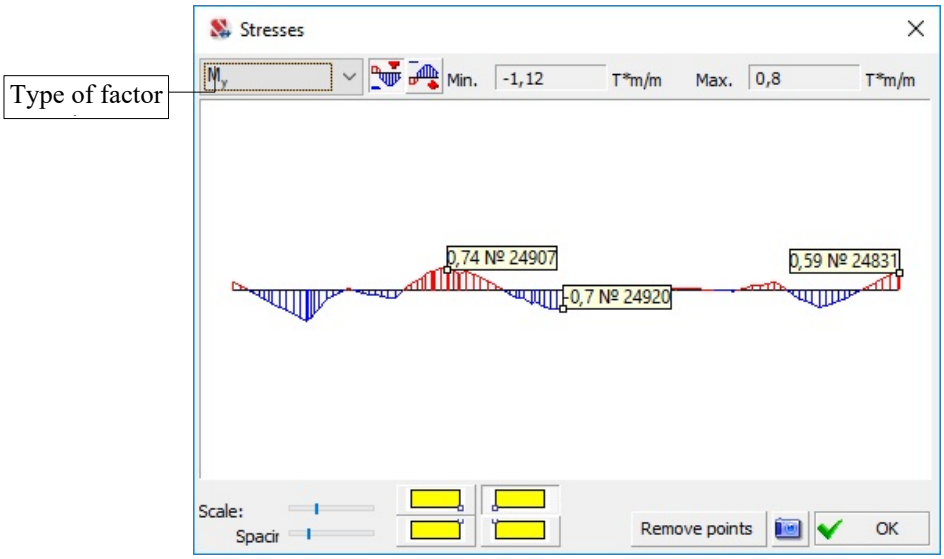


Figure 12.4-4. *The* **Stresses** *dialog box*

After invoking this operation it is necessary to draw a secant line through the part of the model you are interested in and the **Stresses** dialog box will appear (Fig. 12.4-4). This dialog box contains the following controls:

- list of force factors the name of the factor is selected from;

- buttons and for switching the direction of display of diagrams depending on the sign (positive values of factors are shown in the diagrams in red);
- **Scale** — diagram scaling control;
- **Spacing** — diagram hatch spacing control (relative distance between hatching lines);
- **Remove points** — removing leaders with the values of the factor on the diagram;
- the **Save the image as a wmf. file** button — for saving diagrams in the Windows Metafile Format (the **.wmf** extension);
- diagrams will be plotted in the section of the model which is formed by a plane passing along the secant line and perpendicular to the plane of the screen.

The following features should be taken into account when plotting diagrams:

- regardless of the length of the secant line the diagrams will be plotted on all parts of the model (fragment) lying in the secant plane;
- diagrams are not plotted for bars lying in the secant plane.

In order to display the values of the considered factor on the diagram you just have to place the cursor over a point in the diagram and left-click. If you want to delete a leader with the value of the factor, click on the same point again.

Isofields of Stresses in Plates

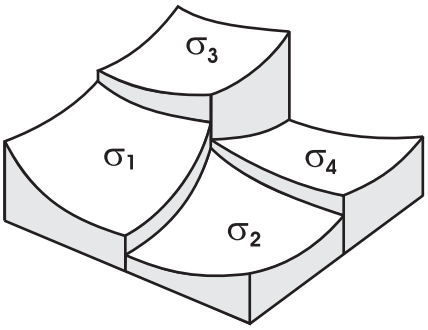


Figure 12.4-5. *Discontinuous stress field*

When a finite element approach in displacements is used, the calculation of stresses involves a number of problems. The local calculation of stresses for each element results in the stress fields with discontinuities at the inter-element boundaries (Fig. 12.4-5). A certain method of averaging the contributions of stresses of individual elements into their common node (averaging over the star of elements of the node) is used in these cases to obtain nodal stresses.

It is provided in the program that for each finite element these forces are calculated in the centers (in the center of gravity) and in the points of junction with the nodes of the design model.

Then the arithmetic mean value of the given forces is calculated for the node for the junction points of all elements coming into the node and defining the considered force provided that such elements are located approximately in one plane. The very concept " approximately in one plane" is regulated by the value of the "shallowness parameter", which is by default taken as 0,95 (it is the cosine of the angle between the normals to adjacent elements approximately equal to 18°), but can be changed by the user.

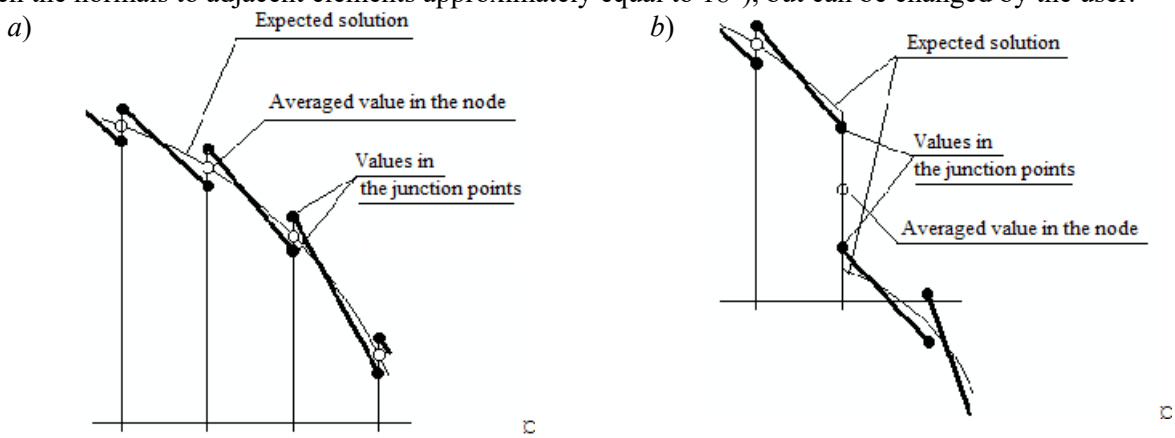


Figure 12.4-6. *Illustration of the behavior of the finite element solution: a — smooth case; b — discontinuous solution*

The isofields or isolines are plotted based on these arithmetic mean values, the same values are output on the screen at the respective setting of the mode of outputting the results.

Concerning the above it should be noted that the averaging makes sense only for the so-called regular nodes where the discontinuous variation of forces should not be expected. Then it results in a value approaching the expected “smooth” solution (Fig. 12.4-6, *a*). In those cases when a jump of the values of internal forces is expected in the node (Fig. 12.4-6,*b*), the averaging of values smoothes such a discontinuity which can lead to ambiguity.

Therefore, in the cases when a concentrated action is applied in the node, or a constraint is imposed there, or finally if a discontinuous variation of rigidity occurs in the area of the node, the output nodal values should be treated with caution. For example, there should not be any moments on ABEF in a rectangular plate clamped along the contour ABCDEF and along a certain internal line BE, which is loaded in the rectangle BCDE. However, a picture shown in Fig. 12.4-7 occurs on the BE line due to the averaging of moments.

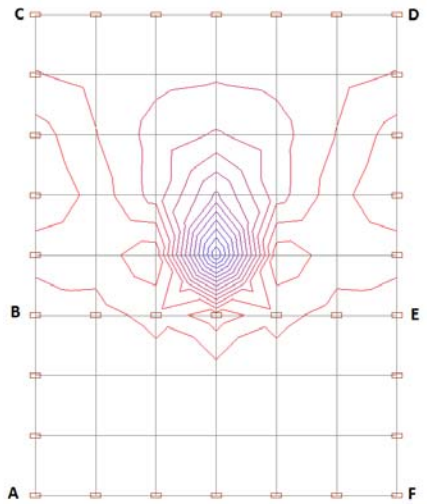


Figure 12.4-7. *Error due to averaging*

You can check yourself in the above cases by outputting the detailed data on the stress-strain state of the elements, i. e. when setting the parameters of the graphical environment you should specify the output of the values of force factors in the nodes of the plate

The above procedure of averaging is used by default. However, the software provides other options as well. If you right-click on the button , the multi-tab **Set the Information Display Filters** dialog box will appear. The following options can be selected in the *Output* group in the **Digitization** tab (see Fig. 12.4-8):

- **Average value**;
- **Minimum value**;
- **Maximum value**;
- **Residual**.

The **Average value** option is described above. When selecting the **Minimum/Maximum value** options when plotting isofields and isolines the minimum/maximum values of the factor selected from the values of the factor in the elements adjacent to the node are used as the value of the factor in the node. The **Residual** option enables to plot the isofields and isolines, for which the value in the node is equal to the difference of the maximum and minimum values of the factor in the adjacent elements. This enables to estimate the error due to averaging.

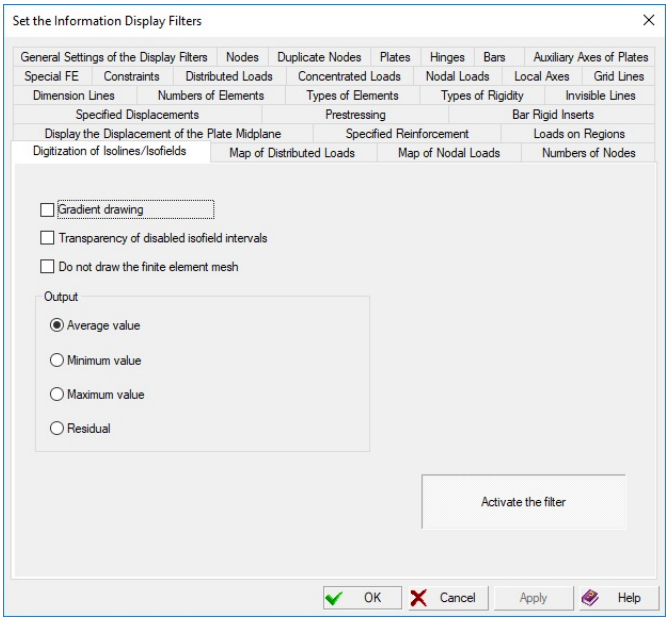


Figure 12.4-8. *The* **Digitization** *tab*

Units of Measurement of Force Factors

A table with the list of all force factors and their dimensions, as well as the indication of classes of finite elements where they can occur is given below (the dimensions are given assuming that the basic system of units is used — tons for measuring forces and meters for measuring linear values).

Force factors and units of measurement

Identifier	Units of measurement	Type of elements	Notes
N	T	Bars	axial force
M_K	Tm	Bars	moment rotating a section about X_1
M (M_Z)	Tm	Bars	bending moment about Z_1
Q (Q_Y)	T	Bars	shear force acting on the section of the bar along the Y_1 axis
M_Y	Tm	Bars	bending moment about Y_1
Q_Z	T	Bars	shear force acting on the section of the bar along the Z_1 axis
R_Y	T/m	Bars	reactive earth pressure in the Y direction
R_Z	T/m	Bars	reactive earth pressure in the Z direction
N_X	T/m^2	Deep beams, solid and axisymmetric elements	normal stress along the X_1 axis, (for axisymmetric elements N_r)
N_Y	T/m^2	Shells, deep beams[1], solid and axisymmetric elements	normal stress along the Y_1 axis, (for axisymmetric elements — N_θ)
N_Z	T/m^2	Deep beams, shells [1], solid and axisymmetric elements	normal stress along the Z_1 axis
T_{XY}	T/m^2	Shells and solid elements	shear of the section in the respective plane, (for axisymmetric elements — T_{rZ})
T_{XZ}	T/m^2	Deep beams, solid and axisymmetric elements	
T_{YZ}	T/m^2	Solid elements	
M_X	Tm/r.m	Shells, plates	moment acting on the section, orthogonal to the respective axis
M_Y	Tm/r.m	Shells, plates	
M_{XY}	Tm/r.m	Shells, plates	torque
Q_X	T/r.m	Shells, plates	shear forces in the section, orthogonal to the respective axis
Q_Y	T/r. m	Shells, plates	
R_Z	T/m^2	Shells, plates with the specified Winkler coefficient	reactive earth pressure
WG	mm	Multilayer finite elements	
R_X	T	Special	reactions
R_Y	T	Special	
R_Z	T	Special (type 53,54)	
R_{UX}	Tm	Special	reaction moments
R_{UY}	Tm	Special	
R_{UZ}	Tm	Special	

[1] The given forces in the given elements are calculated only for the plane strain problems.

12.5. Analysis of the Results of Postprocessing

Figure 12.5-1. *The* **Postprocessors** *toolbar*

The operations of controlling the postprocessing and the analysis of the results of postprocessing can be invoked from the **Postprocessors** toolbar (Fig. 12.5-1) . They include:

— analysis of loads *from the structural fragment* (see Chapter 22);

— *reactions in constraints* (see Sec.12.6);

— forces in *special elements* (see Sec. 12.7);

— analysis of *principal and equivalent stresses* (see Chapter 26);

— energy postprocessor (see Chapter 25);

— *progressive collapse* (see Chapter 23);

— postprocessor of *design combinations* (see Chapter 18).

Most of the descriptions are given in the chapters devoted to the respective postprocessors.

12.6. Graphical Display of the Reactions in Constraints

In order to calculate the reactions in constraints you should check the respective checkbox in the **Calculation Parameters** dialog box (see Chapter 11). In this case the values of reactions in all nodes of the design model in which the constraints are specified will be obtained in the process of the calculation. The control of the graphical display of the results of the calculation of reactions is performed in the **Reactions in constraints** section of the **Postprocessors** toolbar (Fig. 12.6 -1).

Reactions can be displayed by arrows — (similarly to the nodal loads) or as color markers — . Besides the indicated buttons the toolbar also contains two lists. The first one is used for selecting the displayed factor, and the second one is the list of loadings and combinations.

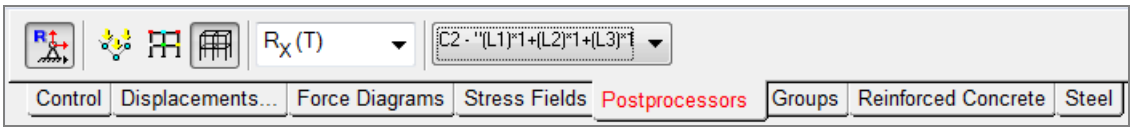


Figure 12.6-1. *The* **Reactions in constraints** *toolbar*

You can return to the initial design model by clicking the button .

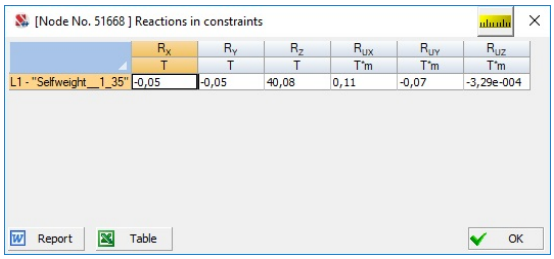


Figure 12.6-2. *The* **Reactions in Constraints** *dialog box*

The **Reactions in constraints** button in the **Information on the Node** dialog box enables to obtain a table with reactions for all loadings/combinations for the selected node (Fig. 12.6-2).

12.7. Graphical Display of the Forces in Special Finite Elements

The postprocessor enables to display the forces in special finite elements on the design model.

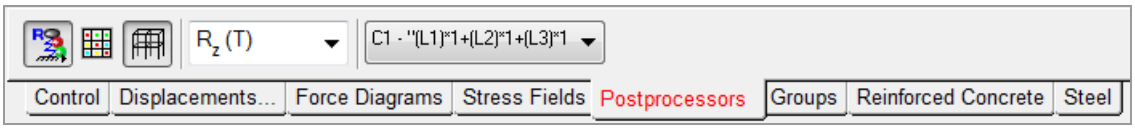


Figure 12.7-1. *The* **Forces is special elements** *toolbar*

The toolbar of this mode (Fig. 12.7-1) contains the following operations:

— display the color map of forces for the selected loading or combination of loadings;

— display design model.

In order to obtain the necessary display you have to select the respective loading (or combination of loadings) and the type of force from the drop-down lists in the toolbar.

You can return to the initial design model by clicking the button .

12.8. Generation of the Groups of Elements

The **Groups** toolbar (Fig. 12.8-1) enables to specify groups of nodes and elements for their postprocessing.

Figure 12.8-1. *Toolbar of the mode of the generation of* ***groups***

The toolbar includes the operations of generating standard groups — , which repeat similar operations of the **Groups** toolbar of the window of the design model generation (Sec. 6.6), and special operations — for generating structural groups for postprocessors of the selection/check of elements of reinforced concrete and steel structures.

The set of operations and the procedure for working with groups of reinforced concrete and steel elements are described in Chapters 19, 20 respectively.

13. Documentation of the Initial Data and Analysis Results

In order to document the initial data and the results of the calculation you can use the generator of tables in a text format (the results of its work are displayed in the text editor specified in the settings of the software) or a special subsystem — **Documenter**, which enables to export the tables into MS Word or MS Excel.

The main differences between the table generator and the **Documenter** are as follows:

- the table generator enables to obtain one table with the specified type of the initial data or results as a text file at a time;
- **Documenter** enables to generate a report document in the form of an MS Word or MS Excel document with the specified set of initial data and/or results.

The principles of control of both subsystems are quite similar and are described below.

Documenting the results of the selection of reinforcement in the elements of reinforced concrete structures, and also of checking and selecting sections of elements of steel structures are described separately in the chapters devoted to these postprocessors.

13.1. Generator of Tables in a Text Format

The general control of the table generation is performed in the **Presentation of the Analysis Results** dialog box (Fig. 13.1-1), which is invoked from the **Print Tables** section of the **RESULTS** group in the **Project tree**. Let's consider the issues dealing with the setting of the mode of outputting and printing the results in more detail.

The **Output parameters** button is used for the specification of the rules of generation and presentation of the results. It invokes a multi-tab dialog box (Fig. 13.1-2) which provides access to the following operations:

- specification of the settings of the table generator;
- specification of the lists of nodes or elements for which the results are output;
- specification of the list of loadings;
- specification of the output units of measurement.

These operations are performed in the respective tabs.

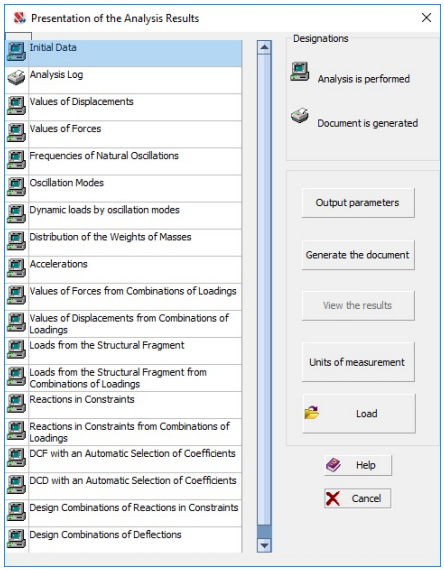


Figure 13.1-1. *The* **Presentation of the Analysis Results** *dialog box*

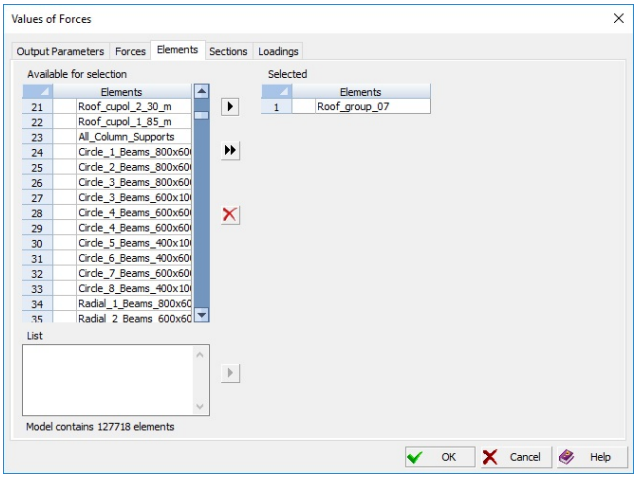


Figure 13.1-2. *The* **Elements** *tab* (*the* **Nodes** *tab* is similar)

Lists of nodes and elements are specified in the **Nodes** and **Elements** tabs (Fig. 13.1-2), respectively, depending on the type of results specified in the **Presentation of the Analysis Results** dialog box. Results for all objects of the model are printed by default (the *Selected* list is empty). Otherwise, the list of objects

can be input into the **List** of nodes/elements line. The list consists from the numbers of objects separated by spaces. If the numbers are consecutive, the intermediate objects are not listed, and the numbers of the first and the last one are indicated. For example, 1 2 4-9 23 56 72-109. After filling the list click the button on the right from the text field.

This tab also contains the list of groups of elements/nodes. After selecting one or several groups in the list and clicking the respective button , you can obtain a report document only for those elements/nodes which are included in the selected groups. The button enables to "transfer" all groups at once. The selected lines can be deleted from the **Selected** list by clicking the button .

The **Loadings** tab enables to select the numbers of loadings (Fig. 13.1-3). Results for all loadings are output by default (the *Selected* list is empty). If only the results of the specified loadings are output, they are selected in the **Available for selection** list.

Moreover, the software also enables to select sections for bar elements for which the forces will be output (for example, select only the first section); you can also specify which combinations should be included in the report document for DCF (for example, only the design ones).

The **Output Parameters** tab (Fig. 13.1-4) enables to specify the number of lines on the page and the width of tables, the accuracy of output of displacements and forces, select the type of table, orientation of pages, etc.

Tables of two types can be generated in the text format.

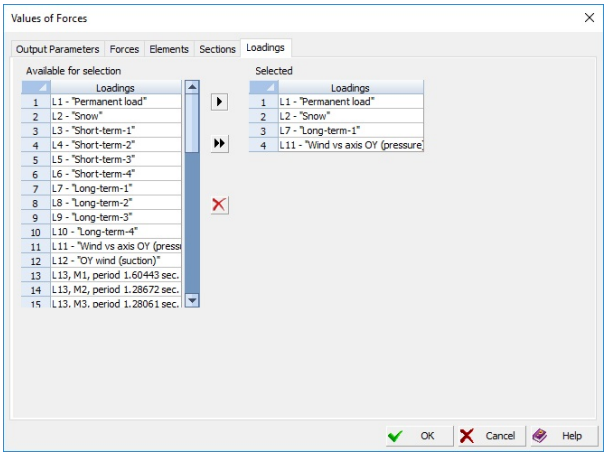


Figure 13.1-3. *The* **Loadings** *tab*

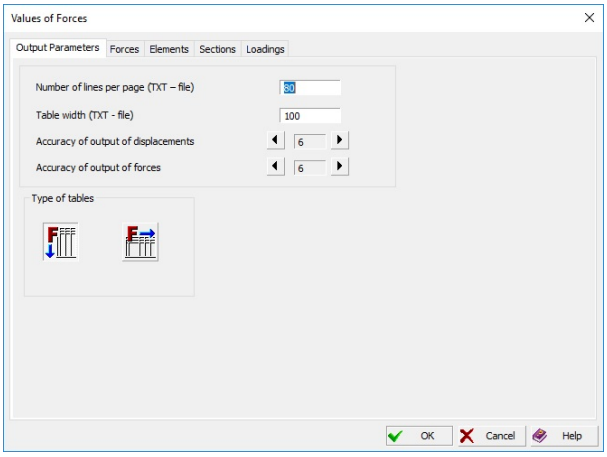


Figure 13.1-4. *The* **Output Parameters** *tab*

In the tables of the first type (icon in the **Type of tables** group) the names of force factors or displacements are written at the beginning of each line, and the numbers of nodes or elements are given in the headings of columns. In the tables of the second type (icon) the headings of columns contain the names of force factors or displacements, and the lines contain the numbers of elements and nodes.

Example of a table with the output "in lines"			
Node displacements			
	30	31	32
1 - Self-weight			
X	–.003771	.000777	.000398
Y	–.001397	–.000824	.000321
Z	–4.50144	–5.11411	–5.41677

Example of a table with the output "in columns"

Node	Loading	X	Y	Z
30	1	–.003771	–.001397	–4.50144
31	1	.000777	–.000824	–5.11411
32	1	.000398	.000321	–5.41677

A respective output mask is used for setting the program to print only certain types of force factors and displacements (Fig. 13.1-5 and 13.1-6).

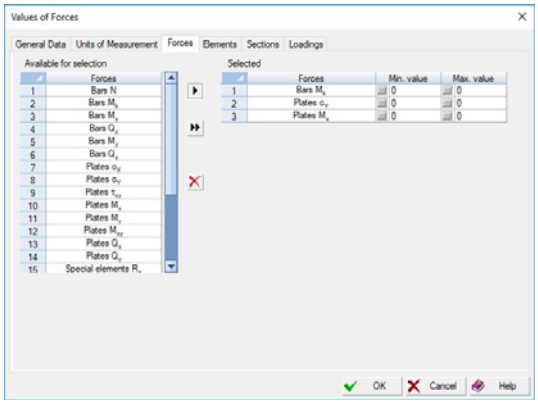


Figure 13.1-5. *The* **Forces** *tab*

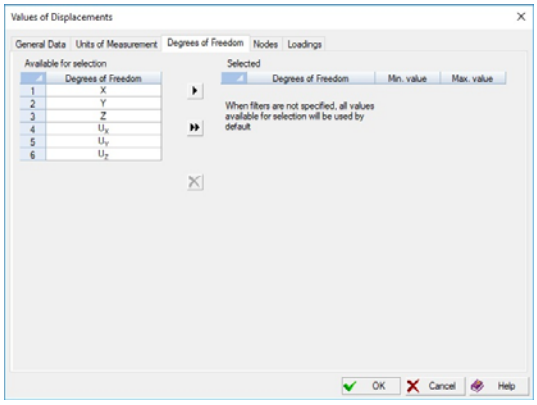


Figure 13.1-6. *The* **Degrees of Freedom** *tab*

It is necessary to perform the following actions to generate the tables with the results of the calculation:

- activate the **Print Tables** section in the **RESULTS** group in the **Project tree**;
- activate the option with the name of the factor (for example, **Displacements** or **Forces and stresses**) in the **Presentation of the Analysis Results** dialog box. If the selected information exists, there is an icon indicating the presence of information next to the name of the option;
- click the **Output parameters** button and specify the necessary parameters (if the parameters are assumed by default, you can skip this step);
- click the **Generate the document** button and wait till an icon indicating that the table is ready appears next to the name of the factor;
- click the **View the results** button, and the window/windows of the text editor with the tables of the results will appear.

Before invoking the function of printing tables it is necessary to perform the customization of the software by specifying the name of the text editor where the tables of the results will be viewed in the **Select a Program** dialog box (the dialog box is invoked with the help of the **Settings of the Result Tables** item in the ***Settings*** menu).

It should be noted that the selected font (Font) is a significant factor affecting the quality of the representation of results in tables. The readability of tables is provided by the Courier (Cyrillic) or Courier New (Cyrillic) fonts, where all letters and numbers have the same width (so-called normal spacing font).

If the table does not fit the width of the page, you can reduce the font size, or reset the parameters of the page in the text editor, for example, reduce the margins or change the format from portrait to landscape, or change the accuracy of the output of values. These specifications can be performed in advance in the **Output Parameters** tab (Fig. 13.1-4).

Generator of tables in the text format creates tables which are difficult to export to Excel spreadsheets or the tables of the Word editor. The software provides a special module — **Documenter**, which enables to obtain fully functional tables.

Names of the files in the text format with the results of the considered problem are created from the names of the files with the data of the problem XXX and extension starting with the letter P (XXX.Pnn). Files with the information on the calculation progress — **Analysis log** (XXX.P01) and a file of the messages about the formal errors in the initial data (XXX.P03), are always generated at the end of the calculation.

The **Appendix** contains a table with the list of extensions of files with the results of the calculation and the results of the table generator.

It is recommended to look through the analysis log (file XXX.P01) even after the successful solution of the problem.

If the operation of saving the structure of the output document was used in the process of working with the **Documenter** (see Sec. 13.2) (a file with the**.SCADDocMaker** extension was generated), you can use the **Load** button in the **Presentation of the Analysis Results** dialog box (Fig. 13.1-1). This will enable to use the same settings as in the **Documenter** for each individual type of the output document when formatting tables.

13.2. Documenter

The initial data and the results of the calculation in the tabular form can be transferred to the MS Word editor or to the MS Excel spreadsheets. The program for generating tables can be invoked from the **Documentation** section of the project tree. As noted above the principles of control of the table generator and the **Documenter** are quite similar. Let's briefly describe the main differences.

The dialog box of the documenter (Fig. 13.2-1) includes a number of lists: **Types of initial data**, **Types of the results**, **Information on the Project, List of report sections**.

Types of initial data – lists prepared when generating a design model, which contain the information on the groups of nodes and elements, control (Fig. 13.2-2) and loads (Fig. 13.2-3);

Types of the results — list of results, obtained in the process of solving the problem;

List of report sections — information selected by the user to be included in the report.

In order for any information to be included in the report it is necessary to transfer its name into the **List of report sections**. The buttons with one ▶ and two ▶▶ arrows are used to do it. The first button provides the selective transfer by one line from the left list to the right one, and the second button enables to transfer the whole list.

The same types of initial data and the results of the calculation can repeat in the sections of the report. This enables to group the information in the report according to certain characteristics, for example, by their belonging to the same group of elements or nodes.

The **Documenter** system provides the **Each group in a separate table** checkbox when specifying the lists of nodes/elements for which the results are output. If you check this checkbox, a separate table will be generated for each selected group of nodes/elements (or a user-specified list of numbers of nodes/elements) instead of a single table with all nodes/elements included in the selected groups.

Taking into account the fact that a design model can be modified during the calculation process, when working with the documenter you can save the once specified structure of the document given as a list of sections of the report in a file with the **.SCADDocMaker** extension and read this structure from the file after performing another variant of calculation. The **Save** and **Load** buttons located in the lower part of the **Results of Analysis** dialog box are used to do it. If necessary, the structure of the report saved in a file can be repeatedly adjusted and supplemented, and also loaded into the table generator in the text format.

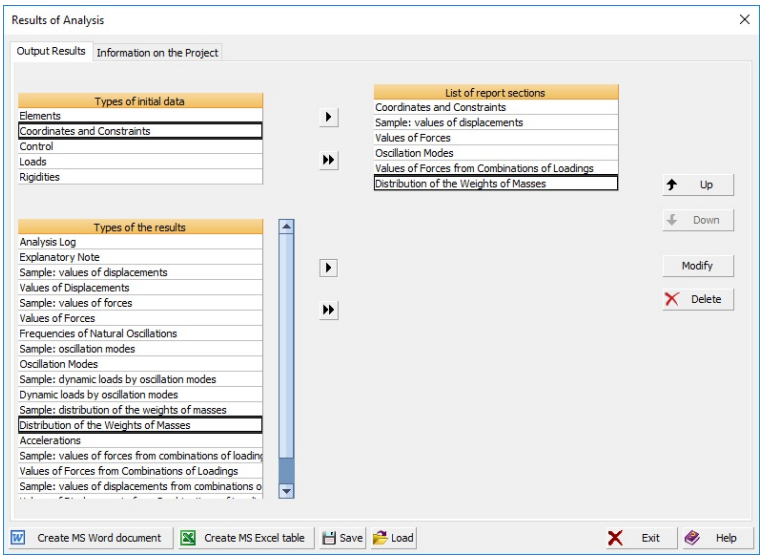


Figure 13.2-1. *The* **Results of Analysis** *dialog box*

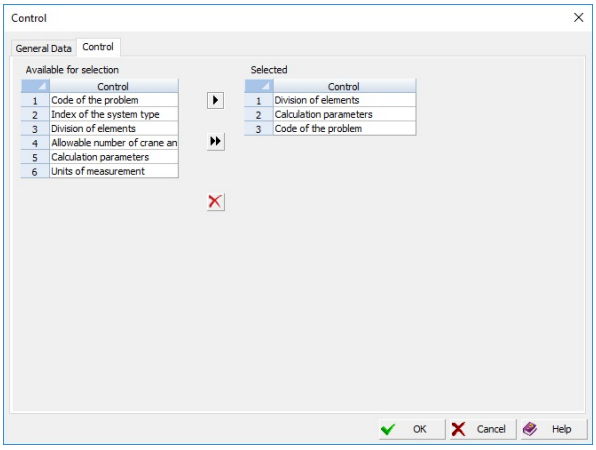


Figure 13.2-2. *The* **Control** *tab*

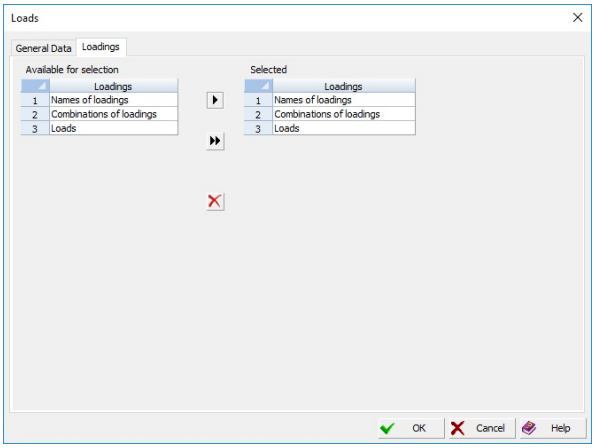


Figure 13.2-3. *The* **Loadings** *tab*

When outputting the results of the calculation clicking the button ▶ invokes the dialog box for setting the contents of the report (Fig. 13.2-4). The structure of this dialog box is standard and does not depend on the type of the results. The dialog box includes several tabs. The name of the report section is input in the first tab, which is included as the name of the section in the **List of report sections** list of the documenter window after closing the dialog box. It also contains the following checkboxes:

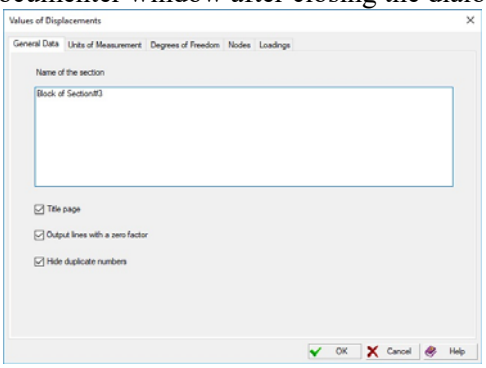


Figure 13.2-4. *The* **Values of Displacements** *dialog box*

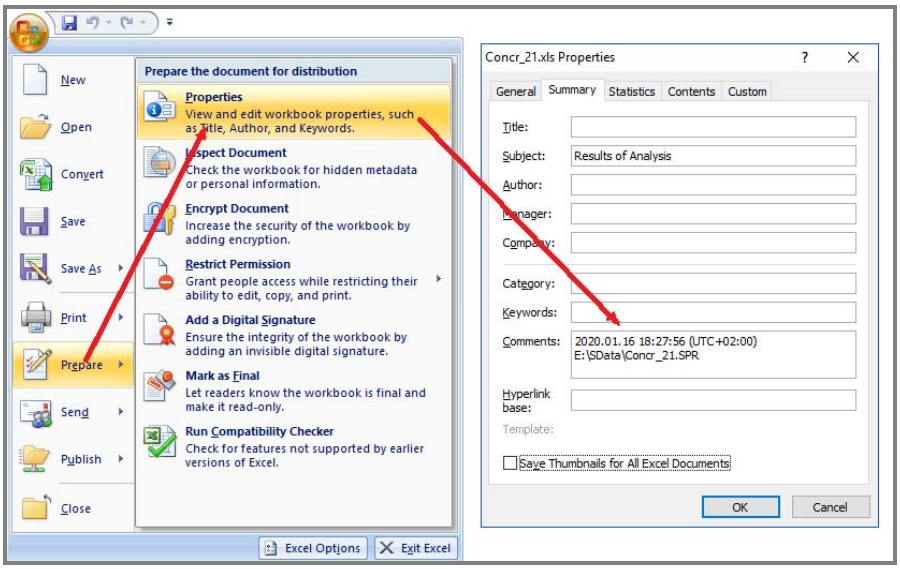
Title page — outputs the title page before each section of the report;

Output lines with a zero factor — generates tables with a full set of lines including the lines where all factors have a zero value (it enables to perform an additional processing of all results in the Excel spreadsheets);

Hide duplicate numbers — excludes the repetition of the number of one node or element in the table (for example, when outputting the forces in different sections of **bars** of one element, the number of the element will be output only once next to the number of the first section).

Export Tables

Tables can be generated in the **.xls** format for exporting to Excel spreadsheets (part of the Microsoft Office System) or in the **.rtf** format — for subsequent visualization, processing and printing in the text editor, for example, Write, Word etc. The process of preparing a report finishes with the generation of tables and their transfer to the specified text editor or spreadsheets. The respective application can be invoked from the **Results of Analysis** dialog box (Fig. 13.2-1). The standard **Save As** dialog box of the Windows environment appears, where you have to specify the name of the file with the tables of results, and then the application will be automatically invoked with the created file. When generating an xls-file, the Documenter enters the path to the corresponding spr-file and the report generation date into the properties of the xls-document.



13.3. Reading the Results

First the software determines the stress-strain state of the structure subjected to static and dynamic actions. These results include the displacements of nodes of the design model, forces (stresses) in its elements, and also, in the case of dynamic actions — frequencies, periods and modes of natural oscillations, and dynamic loads. All these data can be displayed in the graphical or tabular form. Moreover, they are used in different postprocessors included in the software (stability analysis, selection of reinforcement, checks according to different failure theories, determination of design combinations of forces, combinations of loadings etc.), and a number of program-satellites of **SCAD Office** (such as **ARBAT**, **Kristall**).

Information with the results of the selection and check of reinforcement in the elements of reinforced concrete structures, and also of the check and selection of sections in the elements of steel structures are described in the respective sections of this book.

Window of the Processor

This window (Fig. 13.3-1) appears when performing the calculation and includes four areas:

1. Matrix profile of the considered problem with the indicated number of the degrees of freedom[1];
2. Analysis log;
3. Design model;
4. Status bar with the information about the progress of another calculation stage and the progress indicator.

The design model of the structure is displayed at the beginning of the work of the processor. The image can be rotated with the help of the mouse. Double left-clicking in the field of the design model returns the image to its initial position.

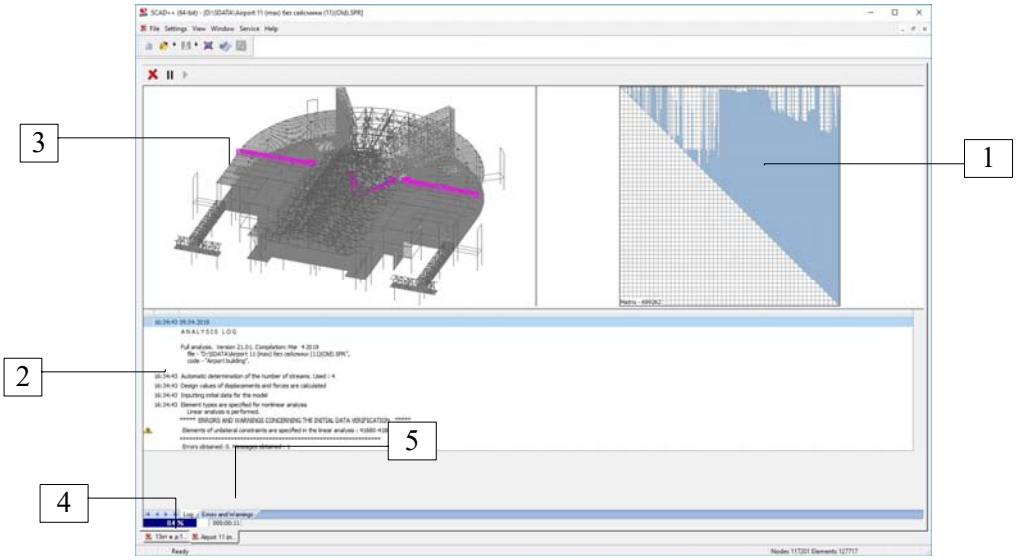


Figure 13.3-1. *The* **SCAD** *window in the calculation mode*

Analysis Log

The analysis log is generated during the calculation process and includes: information on the progress of the main stages of the structural analysis and the detected errors and inaccuracies; data on the design model, necessary resources for performing the calculation; total loads (in the directions allowed in the problem) for each loading, etc.

The log ***must be viewed*** either in the process of performing the calculation or after completing it. Error messages in the log (Fig. 13.3-2), as well as the warnings on the deviation from the normal operation modes of the software, or the inaccuracies made in the design model require special attention.

[1] When solving a problem by the multifrontal method the state of the matrix profile is not coordinated with the speed of solving the problem.

The beginning of the line with the error messages is marked with a sign in the red box in the log which is output in the calculation window, and that with the warning messages is marked with a sign in the yellow box. If the cursor is located in the field of the log and the right mouse button is pressed, a menu will appear, two items of which enable to save the selected line in the clipboard — the **Copy line** operation, if any line in the log has been selected, or to save all lines of the messages — the **Copy all lines** operation. Moreover, if there are errors and warnings, the **Errors and Warnings** tab (Fig. 13.3-1, item 5) appears after finishing the calculation. It opens a list of all warnings included in the analysis log. The main messages can be also viewed in the log of messages of the preprocessor and postprocessor which are described in Sec. 4.1.

If the syntactic and semantic checks are finished successfully, they are followed by several working stages, the first one being the optimization of the matrix profile when the direct methods of the stiffness matrix decomposition are used (Fig. 13.3-3). The successful completion of this stage can significantly speed up the solution of the problem. Since the software suggests several optimization methods, in order to obtain the best result you can stop the process of solving the problem and try to select a different method. Moreover, the program can automatically find the optimal optimization method.

The information on the design model and the resources necessary for solving the problem are included in the log (Fig. 13.3-3). Pay attention to the ratio of the required and free memory on the disk. If there is not enough free memory on the disk, you should stop the calculation (click the **Stop the calculation** button) and try to free the disk space, and continue the calculation by clicking the **Continue the calculation** button . If there are dynamic loadings in the problem and the mode where the number of oscillation modes has to be determined on the basis of the required percentage of modal masses is selected, the prediction of the volume on the hard disk can be significantly reduced.

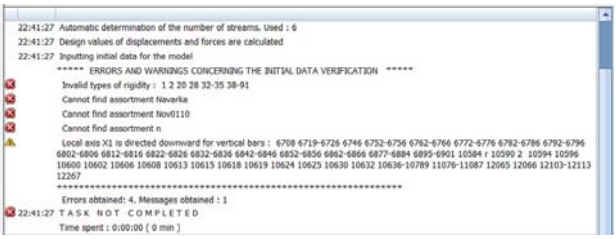


Figure 13.3-2. *Error messages*

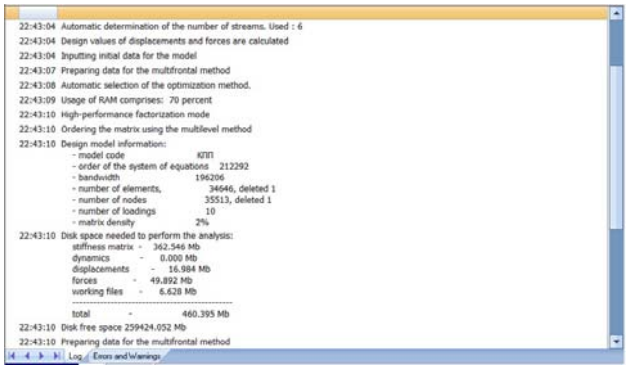


Figure 13.3-3. *Messages on the optimization of the stiffness matrix, information about the design model and the required disk space*

If it is impossible to free the required amount of memory on the disk with the directory with working files and files of results, it is necessary to perform the following actions:

- click the **Close the calculation window** button and exit the calculation mode;
- use the **Close Project** section in the ***File*** menu and close the problem;
- redirect the working directory to another disk in the **Specify Work Directories** section of the ***Settings*** menu;
- open the project and perform the calculation from the beginning.

After the stiffness matrix decomposition a message on the ***dimensional instability of the system*** (Fig. 13.3-4) can appear indicating the numbers of nodes and directions of displacements for which the constraints are automatically imposed (the displacements are restricted).

```
                forces          -     0.098 Mb
                working files   -     0.132 Mb
                ---------------------------------------------
                total           -             4.965 Mb
15:01:50  Disk free space 258136.448 Mb
15:01:50  Preparing data for the multifrontal method
15:01:50  Decomposing the stiffness matrix using the multifrontal method.
15:01:51  Geometrically unstable system in the direction 1 in nodes : 681
15:01:51  Geometrically unstable system in the direction 2 in nodes : 681
15:01:51  Geometrically unstable system in the direction 6 in nodes : 681
15:01:51  Constraints are imposed in the directions of the dimensional instability
15:01:51  Accumulating loads.
```

Figure 13.3-4. *Messages on the dimensional instability of the system*

```
15:34:57  Accumulating masses
15:35:03  Converting static loadings into masses for loading 6 :
          (L1)*0.9+(L2)*0.9+(L3)*0.8+(L4)*0.5+(L5)*0.5
15:35:03  Distributing masses for loading: 6, active masses: 180341
            X       Y       Z       UX      UY      UZ      (T, Tm)
          463.351 463.351 463.357   0       0       0
15:35:09  Converting static loadings into masses for loading 7 :
          (L1)*0.9+(L2)*0.9+(L3)*0.8+(L4)*0.5+(L5)*0.5
15:35:09  Distributing masses for loading: 7, active masses: 180341
            X       Y       Z       UX      UY      UZ      (T, Tm)
          463.351 463.351 463.357   0       0       0
15:35:10  Determination of oscillation modes for loading 6. Block Lanczos method
15:35:10  Seismic mode - to reach the specified sum of modal masses in each direction
15:39:16  Sum of modal masses in percent: Mx = 97.41    My = 98.40    Mz = 85.27
15:39:17  Loading 7 has been analyzed using results of the iteration
            process for loading 6.
15:39:18  Calculating dynamic forces. Loading 6
          Number of the dynamic module : 53, number of modes : 10
          Converting static loadings into masses :
            (L1)*0.9+(L2)*0.9+(L3)*0.8+(L4)*0.5+(L5)*0.5
          Seismic analysis according to DBN B.1.1-12:2006 of Ukraine.
15:39:18  Calculating dynamic forces. Loading 7
```

Figure 13.3-5. *Messages when there are dynamic actions*

Dimensional instability can be caused for example by the errors in the specification of boundary conditions of structures (constraints) or by the insufficient value of the accuracy of the matrix decomposition for a particular problem (minimum allowable number on the diagonal of the matrix of equations at its decomposition).

The automatically specified constraints have to be carefully analyzed. They often do not meet specific boundary conditions of the structure, and in these cases it is necessary to specify the boundary conditions correctly and to solve the problem again.

In those cases when the dimensional instability is caused by the peculiarities of the design model, it might be necessary to change the value of the accuracy of the matrix decomposition, which is performed in the **Calculation Parameters** dialog box (see Chapter 11).

The message on the presence of zero lines in the stiffness matrix is informational and appears if there are elements of different dimensions in the design model (for example, shells and solid elements). If the calculation for dynamic actions is provided in the problem, besides the information on the stages of calculation the log will also contain the data on the number and distribution of active masses (Fig. 13.3-5).

```
18:25:06  Accumulating loads.
            Total external loads (T, Tм)
18:25:06        X        Y        Z       UX       UY       UZ
18:25:06  1-    0.000133255 -0.000161299  6055.36  0.178694 -3.29212    0
18:25:06  2-          0        0  4533.05  -1.5608  2.87694    0
18:25:06  3-          0        0  8418.53 -2.89863  5.34288    0
18:25:06  4-   -4.33237  3.69957   -18549       0        0    0
18:25:06  5-          0        0        0       0        0    0
18:25:06  6-          0        0    23.22       0        0    0
18:25:06  7-          0        0    58.16       0        0    0
18:25:06  8-          0        0    28.76       0        0    0
18:25:06  9-          0        0  417.223 0.0498155 0.0376769    0
18:25:06  10-         0        0 -1058.96 -4.40341  2.16011    0
18:25:06  11-         0        0  1.78171 -5.07432 0.103657    0
18:25:06  12-         0        0  -706.01  6.45833 0.0152046    0
18:25:06  13-         0        0  97.3488 -6.40935 0.106836    0
18:25:06  14-         0        0 -421.829 0.171553 -6.79221    0
18:25:06  15-         0        0  8087.54 -2.85485  5.29788    0
18:25:06  16-         0        0  7367.97 -15.3409  4.80016    0
18:25:06  17-         0        0  7399.09  9.54196  5.82819    0
18:25:06  18-         0        0  7408.59 -1.41626 -9.84064    0
18:25:06  19-         0        0  7539.32 -2.52406  14.072    0
```

Figure 13.3-6. *Table of distribution of total loads*

```
15:34:57  Accumulating masses
15:35:03  Converting static loadings into masses for loading 6 :
          (L1)*0.9+(L2)*0.9+(L3)*0.8+(L4)*0.5+(L5)*0.5
15:35:03  Distributing masses for loading: 6, active masses: 180341
            X       Y       Z       UX      UY      UZ      (T, Tm)
          463.351 463.351 463.357   0       0       0
15:35:09  Converting static loadings into masses for loading 7 :
          (L1)*0.9+(L2)*0.9+(L3)*0.8+(L4)*0.5+(L5)*0.5
15:35:09  Distributing masses for loading: 7, active masses: 180341
            X       Y       Z       UX      UY      UZ      (T, Tm)
          463.351 463.351 463.357   0       0       0
15:35:10  Determination of oscillation modes for loading 6. Block Lanczos method
15:35:10  Seismic mode - to reach the specified sum of modal masses in each direction
15:39:16  Sum of modal masses in percent: Mx = 97.41    My = 98.40    Mz = 85.27
15:39:17  Loading 7 has been analyzed using results of the iteration
            process for loading 6.
15:39:18  Calculating dynamic forces. Loading 6
          Number of the dynamic module : 53, number of modes : 10
          Converting static loadings into masses :
            (L1)*0.9+(L2)*0.9+(L3)*0.8+(L4)*0.5+(L5)*0.5
          Seismic analysis according to DBN B.1.1-12:2006 of Ukraine.
15:39:18  Calculating dynamic forces. Loading 7
```

Figure 13.3-7. *Message on the normal completion of the calculation*

Values of the total nodal load are indicated in the log (Fig. 13.3-6). Depending on the option selected in the parameters of the control of the calculation (see Chapter 11), the log will include values either taking into account the loads "taken by constraints", or not taking them into account. Total moments with respect to the origin can be calculated.

The user usually knows the approximate total loads on the structure in the linear directions. If the boundary conditions of the problem are not very difficult, you can determine whether the calculated total load corresponds to the specified one. If the load in the loading is less than the specified value, it is necessary to find the missing loads, and if it is greater – you have to find the excessive loads.

At the stage of checking the solution a message may appear indicating the value of the error which occurred when solving the system of equations for controlling the loading. Moreover, the global number of the equation, number of the node, and number of the local equation locating the place of the possible source of the error are indicated as well.

An error can be caused by an “almost dimensionally unstable system” or by a large difference between stiffnesses of adjacent elements. In any case the possible causes of this message have to be carefully analyzed.

The user can, on his own responsibility, uncheck the **Solution check** checkbox in the **Calculation Parameters** dialog box (see Chapter 11).

Documentation of the Initial Data

SCAD enables to graphically display the design model of the structure and most of its attributes (loads, constraints, hinges, etc.). In those cases when it is necessary to obtain information about certain characteristics of the design model in the tabular form, you can use the **Documenter** or the **Generator of tables** in the text format. It is obvious that the full tabular description of the model consisting of thousands of nodes and elements is unreasonable because of the large amounts of data. However, the description of the stiffness properties of elements (Fig. 13.3-8) and some other tables can be useful in the process of working with the model and for archiving the “hard copies” of project materials.

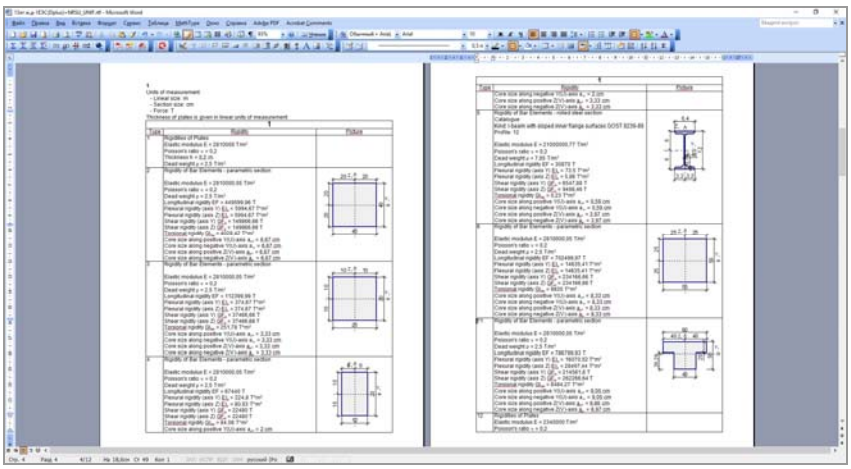


Figure 13.3-8. *Example of the display of the table of stiffnesses in the* MS Word *editor*

13.4. Displacements of Nodes

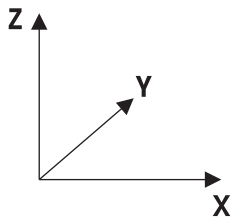


Figure 13.4-1. *Global coordinate system*

One of the main characteristics of the stress-strain state are the displacements of nodes of the design model of the considered structure. They are always calculated in the main (global) Cartesian coordinate system XYZ (Fig. 13.4-1). The number of displacements calculated for each node is defined by the used index of the model type (see Chapter 3).

There are six displacements in a node of spatial structures for which the index of the model type *General type system* “5” is specified:

X — linear displacement along the X axis;
Y — linear displacement along the Y axis;
Z — linear displacement along the Z axis;
$\mathbf{U_X}$ — rotation angle about the X axis;
$\mathbf{U_Y}$ — rotation angle about the Y axis;
$\mathbf{U_Z}$ — rotation angle about the Z axis.

For models with index “1” (plane hinged bar system) only the linear displacements **X** and **Z** are calculated.

For some indexes of the model type additional degrees of freedom (additional unknowns) are assigned to the node of the design model. These degrees of freedom are no longer identified with the components of linear or angular displacements of the node as an infinitesimally small rigid body in a point coinciding with the center of the node.

For example second mixed derivative (torsion) of the deformed surface of a plate or the components of deformations of transverse shear and compression in laminated piecewise inhomogeneous shallow shells can be such an additional unknown. These values are usually auxiliary and are used to improve the accuracy of the calculation of the main unknowns of the model (linear displacements and rotation angles), and their calculated values do not have to be analyzed.

Positive Directions of Displacements

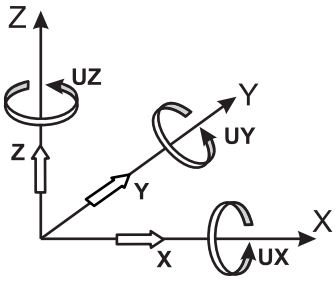


Figure 13.4-2. *Positive directions of linear and angular displacements*

Positive directions of linear displacements coincide with the directions of the respective axes **X**, **Y**, **Z** (Fig. 13.4-2). Counterclockwise rotation about the respective axis when viewed from the end is considered as positive for angular displacements $\mathbf{U_X}$, $\mathbf{U_Y}$, $\mathbf{U_Z}$.

Tables of Displacements

Displacements are output in millimeters by default, and the angular values are output in thousandths of a radian. The specified units of measurement are output in the heading of each table.

Names of loadings	
Number	Name
1	Permanent
2	Live
3	Snow
4	Wind along X
5	Wind against X
6	Wind along Y
7	Wind against Y

Figure 13.4-3. *The "Names of loadings" table*

The section can consist from:

- the names of loadings with their numbers (Fig. 13.4-3). The numbers of loadings instead of their names are used in other sections;
- maximum and minimum displacements of the design model (Fig. 13.4-4). A table can be printed for the performed selection or for the whole design model;
- the actual values of displacements for the list of nodes, loadings specified in the **Documenter** (Fig. 13.4-5) and the specified factors.

The heading of the table with the values of displacements contains units of measurement and parameters of selection: lists of nodes, loadings, and output factors.

If there are dynamic loadings the following information can be output in the **Mode number** column (see Sec. 12.1 **Selecting Loading**):

SD – amplitude from the total dynamic load;

LS+SD – amplitude from the static (for example, wind) and total dynamic loads;

RD and **RI** – at the calculation for harmonic oscillations with the real and complex components of the action with the indicated number;

T – at the direct integration of the equations of motion with the indicated number of the savepoint.

There are some peculiarities of identification described in the respective sections for the nonlinear analysis, variation of models, and erection.

Selection: values of displacements
Units of measurement: inch
Parameters of selection:
List of nodes/elements: All
List of loadings/combinations: All
List of factors: All

Selection: values of displacements						
Name	Maximum values			Minimum values		
	Value	Node	Loading	Value	Node	Loading
X	0,422	302	6	-0,582	302	7
Y	5,469	302	6	-7,325	302	7
Z	0,082	72	6	-1,904	209	1
U_X	0,338	789	1	-0,327	776	1
U_Y	0,217	783	1	-0,133	1736	1
U_Z	24,824	302	6	-33,629	302	7

Figure 13.4-4. *The "Selection: values of displacements" table*

The average component shows which direction (sign of the respective displacement) should be taken into account in the analysis of the results of the consideration of pulsation. Moreover, some codes (for example, SNiP 2.01.07-85* "Loads and Actions") allow to limit the number of the considered natural oscillation modes of the structure. Therefore, despite the specified number of modes, less modes can be taken into account when performing the calculation. Sign of the value of displacement (and force) from all the considered modes defines the sign of the respective displacement (force) from the static component of the wind load.

Values of displacements
Units of measurement: mm
Parameters of selection:
List of nodes/elements: 32-46 405 867-877 2425-2427
List of loadings/combinations: All
List of factors: All

Values of displacements							
Node	Loading	Value					
		X	Y	Z	U_X	U_Y	U_Z
32	1	-0,002	0,005	-34,936	0,113	0,034	-0,011
32	2	2,32e-004	5,129e-005	-3,169	0,012	0,004	-0,001
32	3	-1,511e-004	1,791e-004	-0,714	0,003	0,003	5,603e-005
32	4	0,168	0,053	-0,962	-0,001	0,014	0,01
32	5	-0,154	-0,046	0,892	0,001	-0,013	-0,009
32	6	0,021	0,154	0,972	-0,016	2,428e-004	0,008
32	7	-0,021	-0,154	-0,971	0,016	-2,447e-004	-0,008
33	1	-0,001	0,008	-34,748	0,131	-0,011	-0,007
33	2	2,285e-004	3,468e-004	-3,099	0,014	4,557e-004	-0,001
33	3	-7,306e-005	1,96e-004	-0,593	0,003	0,001	6,441e-005
33	4	0,167	0,017	-0,237	-0,001	0,01	0,008
33	5	-0,153	-0,015	0,221	0,001	-0,009	-0,007
33	6	0,02	0,14	0,953	-0,013	0,001	0,006

Figure 13.4-5. *Fragment of the "Values of displacements" table*

Displacements from Combinations of Loadings

Displacements of nodes obtained in the result of the calculation of combinations of static loadings are determined as an algebraic sum of displacements from each loading included in the combination taking into account the coefficients specified for them. If the combination includes one or several dynamic loadings, displacements from all modes for each of them (except for the pulsation component of the wind flow) are usually taken according to the formula:

$$\delta = \sqrt{\sum_{i=1}^{KF} \delta_i^2},$$

where KF is the number of calculated natural oscillation modes,

δ_i is the generalized displacement from the *i*-th natural oscillation mode.

Displacements are calculated according to the following formula for the pulsation component of the wind flow:

- according to SNiP — $\delta = \delta_c \pm \sqrt{\sum_{i=1}^{KF} \delta_i^2}$;
- according to MGSN — $\delta = \delta_c \pm \delta_d$,

where KF is the number of calculated natural oscillation modes;

δ_i is the generalized displacement from the *i*-th natural oscillation mode;

δ_d is the dynamic component which is determined taking into account the correlation of natural oscillation modes and which is output as a separate line in the table;

c is the number of the loading of the static component of the wind action.

The plus or minus sign of the radical expression is taken in accordance with the sign of δ_c.

Displacements calculated according to these formulas are taken then with the specified coefficient in the respective combinations.

The above procedures of calculating displacements are obviously nonlinear, and the values of displacements which occur for example in the process of seismic oscillations obtained according to the "root of the sum of squares" rule correspond to different moments in time. The maximum displacement is actually evaluated in each node independently from other nodes, and the sign is not taken into account.

13.5. Tables of Forces and Stresses in Elements

Except for a small group of finite elements the values of forces (stresses) are calculated in the local coordinate system which is individual for each element (see Chapter 3). A certain user-defined coordinate system for the output of forces (stresses) different from the direction of axes of the local coordinate system can be specified for flat and solid finite elements. It enables to use the obtained results more correctly. For example, it is recommended to specify these directions of the force output when selecting the reinforcement in floors, where it is necessary to analyze the obtained values of forces and stresses after reducing the results of the calculation to the uniform axes, although the local axes of the elements can be arbitrarily directed.

In the general case values of forces for bar FE are calculated for three sections: in the beginning, in the middle, and in the end of the bar. A greater number of sections for calculating forces can be specified as well (up to 255).

Selection: values of forces
Units of measurement: T, m
Parameters of selection:
List of nodes/elements: All
List of sections: All
List of loadings/combinations: All
List of factors: All

Selection: values of forces										
Name	Maximum values					Minimum values				
	Value	Element	Section	Loading	Mode number	Value	Element	Section	Loading	Mode number
N	14,406	113	1	6	1	-68,315	202	1	1	
M_k	0,01	59187	1	6	1	-0,013	217	1	6	1
M_y	5,052	59124	3	6	1	-5,811	59123	1	6	1
Q_z	6,986	59124	1	6	1	-2,284	101	1	7	1
M_z	1,521	207	3	7	1	-1,668	204	1	7	1
Q_y	1,725	59124	1	1		-2,686	99	1	6	1
N_X	927,235	30454	1	6	1	-831,55	16225	1	6	1
N_Y	419,215	25586	1	6	1	-569,617	25586	1	1	
τ_{XY}	474,695	49804	1	6	1	-288,341	61080	1	7	1
M_X	10,666	3328	1	1		-8,09	2653	1	1	
M_Y	8,15	6040	1	1		-6,971	4749	1	1	
M_{XY}	4,796	54752	1	7	1	-4,5	5229	1	1	
Q_X	863,67	2647	1	6	1	-245,17	2647	1	1	
Q_Y	1196,152	2647	1	7	1	-1972,14	2653	1	6	1
R_z	4,964	54864	1	7	1	-28,862	54836	1	1	

Figure 13.5-1. *The "Selection: values of forces" table*

The values of forces and stresses in plate and solid FE are by default calculated for the point which is the center of mass of the element, although the forces can be calculated in all of its nodes as well.

The section can consist from:

- the names of loadings (Fig. 13.4-3). The numbers of loadings instead of their names are used in other sections;
- maximum and minimum values of force factors (Fig. 13.5-1). A table can be printed for the performed selection or for the whole design model;
- the actual values of force factors (all or the specified ones) for the list of elements and loadings specified in the **Documenter** (Fig. 13.5-2).

Values of forces
Units of measurement: T, m
Parameters of selection:
List of nodes/elements: 3-8 74-106 109-446 7327-7330 9105-9115 12696-12706
List of loadings/combinations: All
List of factors: N, M_k, M_y, Q_z, M_z, Q_y, N_X, N_Y, τ_{XY}, M_X, M_Y, M_{XY}

Values of forces								
Element	Section	Loading	Value					
			N	Mk	My	Qz	Mz	Qy
3	1	1	-137,051	0,208	5,589	-2,702	11,855	5,685
3	1	2	-10,39	0,015	0,492	-0,29	1,243	0,71
3	1	3	-3,126	0,008	-0,062	0,066	0,083	0,031
3	1	4	1,257	-0,201	1,038	-0,718	0,698	0,511
3	1	5	-1,193	0,181	-0,972	0,674	-0,63	-0,464
3	1	6	-3,316	-0,169	-0,498	0,384	0,815	0,64
3	1	7	3,306	0,169	0,498	-0,384	-0,819	-0,642
3	2	1	-136,143	0,208	1,13	-2,702	2,474	5,685
3	2	2	-10,39	0,015	0,014	-0,29	0,071	0,71
3	2	3	-3,126	0,008	0,047	0,066	0,032	0,031
3	2	4	1,257	-0,201	-0,147	-0,718	-0,145	0,511

Figure 13.5-2. *Fragment of the "Values of forces" table*

The units of measurement and the following parameters of selection are indicated before each of the above tables: lists of nodes, loadings, and the output factors.

If there are dynamic loadings, the **Mode number** column appears, which besides the information on the oscillation modes can also contain the following characteristics of lines **SD**, **LS+SD**, **T**, **RD** and **RI**, which indicate the amplitude from the total dynamic load, etc. (see Sec. 12.1 **Selecting Loading**).

There are some peculiarities of identification described in the respective sections for the nonlinear analysis, variation of models, and erection.

Values of forces
Units of measurement: T, m
Parameters of selection:
List of nodes/elements: 9263
List of loadings/combinations: 1
List of factors: All

Values of forces											
Element	Section	Loading	Value								
			N_X	N_Y	τ_{XY}	M_X	M_Y	M_{XY}	Q_X	Q_Y	R_z
9263	1	1	-1,202	-1,979	-0,612	-3,552	-3,23	-3,352	-10,384	-11,442	-15,366
9263	2	1	-1,051	-1,223	-0,628	-7,689	-4,097	-2,174	-10,384	-11,442	-15,313
9263	3	1	-1,243	-1,987	-0,313	-2,96	-9,264	-3,039	-10,384	-11,442	-14,92
9263	4	1	-1,092	-1,231	-0,329	-7,763	-9,425	-1,862	-10,384	-11,442	-14,874
9263	5	1	-1,147	-1,605	-0,47	-5,491	-6,504	-2,608	-10,384	-11,442	-15,116

Figure 13.5-3.

If an option of the calculation of force factors in the nodes of multinode finite elements was specified when creating a design model, then in the tables with the results of the value of force factors the first numbers of sections correspond to the values in nodes, and the last number of the section corresponds to the center of mass of the element. The values of the force factors for a four-node finite element are output in the given example (Fig. 13.5-3) in five "sections": first four ones — in the nodes, and the fifth one — in the center.

The values of force factors obtained in the result of the calculation of combinations of static loadings are calculated as an algebraic sum of the respective factors from each loading included in the combination taking into account the coefficients specified for them.

The total forces calculated for the main dynamic force and having the identification **S** are taken for the combination of dynamic loadings.

The calculated values of force factors from the dynamic actions obtained according to the "root of the sum of squares" rule or another law are nonlinear and correspond to different moments in time. The maximum value is actually evaluated in each element independently from other elements, and the sign is not taken into account. It can lead to the loss of stability in nodes.

13.6. Stresses and Forces

Forces in Bars

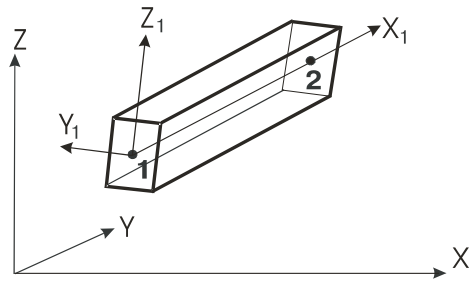


Figure 13.6-1.

Forces in bars are output in the local rectangular coordinate system.

The forces are determined in the flexible part of the element (Fig. 13.6-1) and with respect to the principal axes of inertia in the section perpendicular to the longitudinal axis of the bar. The first section refers to the beginning of the flexible part, and the last one — to its end. The orientation of the local coordinate system of a bar is considered in detail in Sec. 3.3.

It is recommended to display the directions of the local coordinate axes on the design model (Fig. 13.6-2) for the convenience of reading the results of the calculation. Then the location of the cross-sections of the bar, the numbers of which increase with the X_1 coordinate, and also in the direction of the action of bending moments and shear forces, becomes clear.

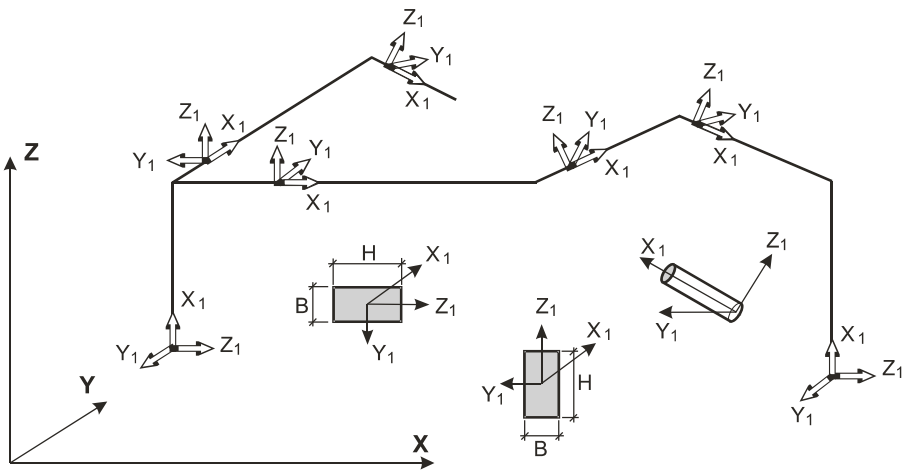


Figure 13.6-2.

Forces in Different Types of Bars

Forces which can appear in different types of bar finite elements, as well as the possible degrees of freedom of their nodes are given in Table 13.6-1.

Table 13.6-1

FE type	Name of FE	Location	Degrees of freedom	Calculated forces
1	Plane truss bar	XOZ plane	X, Z	N, M_Y, Q (Q_Z), r_X
2	Plane frame bar	XOZ plane	X, Z, U_Y	N, M_Y, Q (Q_Z), r_X, r_Z
3	Grade beam bar	XOY plane	Z, U_X, U_Y	M_K (M_X), M_Y, Q_Z, r_Z
4	Space truss bar	Any	X, Y, Z	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X
5	Space bar	Any	X, Y, Z, U_X, U_Y, U_Z	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X, r_Y , r_Z
6	Space bar allowing for shear	Any	X, Y, Z, U_X, U_Y, U_Z	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X, r_Y , r_Z
7	Grade beam bar on the elastic subgrade	XOY plane	Z, U_X, U_Y	M_K (M_X), M_Y, Q_Z, r_Z

FE type	Name of FE	Location	Degrees of freedom	Calculated forces
10	Universal bar	according to the index of the model type	according to the index of the model type or the type of element	N, M_K, M_Y, Q_Z, M_Z, Q_Y, r_X, r_Y , r_Z

Moments and shear forces in bars of types 1 and 4 appear if there are local loads on the elements.

When designating forces in bar elements it is assumed in the tables that the direction of the action of a factor is designated by a lowercase letter (unlike the multinode elements where the uppercase letters are used for this purpose).

Table 13.6-2

Designation of the force factor	Name	Units of measurement (in the basic system of units)	Positive sign of the force indicates
N	Longitudinal force	T	Tension
M_K (or M_X)	Torque, rotation about the longitudinal X_1 axis of the bar	Tm	Counterclockwise action if the section belonging to the end of the bar is viewed from the end of the X_1 axis
M_Y	Bending moment, rotation about the Y_1 axis; causes tension-compression of the lower and upper fibers along the height of the section (in the direction of the Z_1 axis)	Tm	Tension of the lower (with respect to the direction of the Z_1 axis) fiber of the section
Q_Z	Shear force in the direction of the Z_1 axis along the height of the section and corresponding to moment M_Y	T	Shear in the direction of the Z_1 axis of the section belonging to the end of the bar
M_Z	Bending moment, rotation about the Z_1 axis; causes tension-compression of the right and left fibers of the section along the width of the section (in the direction of the Y_1 axis)	Tm	Tension of the upper (with respect to the direction of the Y_1 axis) fiber of the section
Q_Y	Shear force in the direction of the Y_1 axis along the width of the section and corresponding to moment M_Z	T	Shear in the direction of the Y_1 axis of the section belonging to the end of the bar
r_X	Reactive earth pressure in the X direction when specifying the subsoil parameter C_{1X}	T/m	Action in the direction of the X_1 axis (the minus sign indicates that the soil is compressed)
r_Y	Reactive earth pressure in the Y direction when specifying the subsoil parameter C_{1Y}	T/m	Action in the direction of the Y_1 axis (the minus sign indicates that the soil is compressed)
r_Z	Reactive earth pressure in the Z direction when specifying the subsoil parameter C_{1Z}	T/m	Action in the direction of the Z_1 axis (the minus sign indicates that the soil is compressed)

Positive Directions of Forces in Bars

Positive directions of forces in bars are assumed as follows:

- for shear forces Q_Z and Q_Y — in the directions of the respective axes Z_1 and Y_1;
- for moments M_X, M_Y, M_Z — counterclockwise if viewed from the end of the respective axis X_1, Y_1, Z_1;
- a positive longitudinal force always creates tension in the bar.

Internal forces in any section (Fig. 13.6-3) are balanced (directed towards each other). In order to output the results it is necessary to select one of the cross-section planes (one group of the

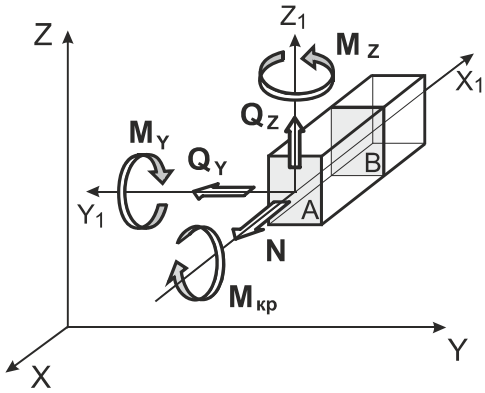


Figure 13.6-3. *Positive directions of forces in bars*

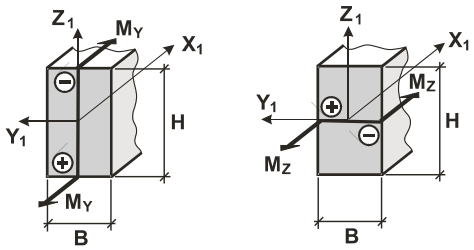


Figure 13.6-4.

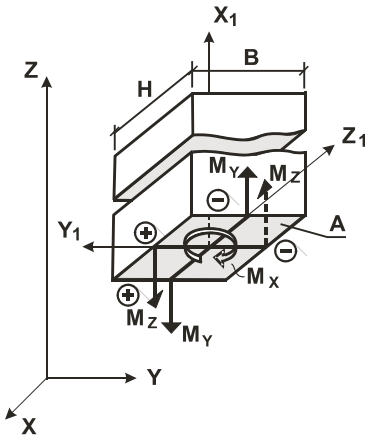


Figure 13.6-5.

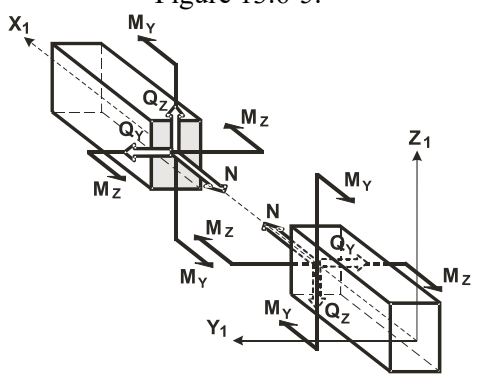


Figure 13.6-6.

direction of forces).

A section which is often called belonging to the end of the bar is selected. A section at the beginning of the bar (Fig. 13.6-3) is a section which we can see and which is under the action of the "dropped" part of the rest of the structure. A section at the end of the bar is not last, but almost last.

If the positive moment M_Y (girder moment) creates tension in the lower (with respect to the direction of the Z_1 axis) fiber of the section of the bar and rotates the section counterclockwise (viewed from the end of the Y_1 axis), the forces should be shown for the plane A (Fig. 13.6-4, 13.6-5) as actions of internal forces from the dropped part of the bar adjacent to the beginning of the flexible part.

Positive directions of internal forces in a bar section are shown in the figures, and the rules for reading them are given in Table 13.6-2. Units of measurement of the calculated forces are given in the table in the basic system of units.

In Fig. 13.6-4, 13.6-5 sign "+" (plus) indicates the tension, and sign "–" (minus) — compression face of the section under the action of positive moments M_Y and M_Z.

If considerable concentrated local loads are applied to the bar, the maximal absolute values of shear forces and moments can be lost when outputting the results (Fig. 13.6-7, *a*, *b*), and, for example, one of the signs of forces can be lost as well (Fig. 13.6-7, *b*). It can also happen when the concentrated load is applied "almost" in the force output point.

In such cases it is advisable to specify an additional node in the bar (Fig. 13.6-7, *c*), i. e. divide the bar into two parts, then one group of forces will be at the end of one, and the other — at the end of the other bar (the load here is not local, but nodal).

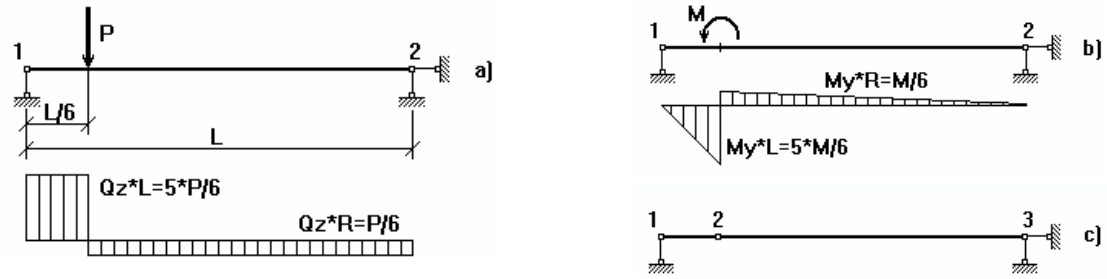


Figure 13.6-7.

Flat Finite Elements

Local Coordinate System of Flat Elements

The origin of the local coordinate system (Sec. 3.4) of the elements of plates is in the node with the number 1 (node 1); two following nodes (nodes 2 and 3) completely define the direction of the axes X_1 and Y_1 for plates and shells, and axes X_1 and Z_1 — for deep beams.

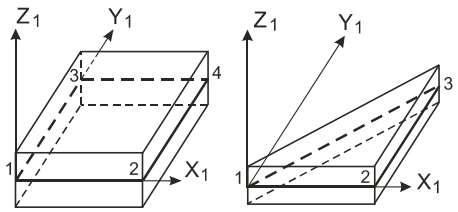


Figure 13.6-8. *Plates and shells*

Directions of the local axes (Fig. 13.6-8) for plates and shells are as follows:

- the X_1 axis of the plate is directed from node 1 to node 2 (the node which is specified second);
- the Y_1 axis of the plate lies in the plane of the element, it is perpendicular to the X_1 axis, and is directed towards the third node;
- the Z_1 axis is orthogonal to X_1 and Y_1 and makes a right-hand system with them.

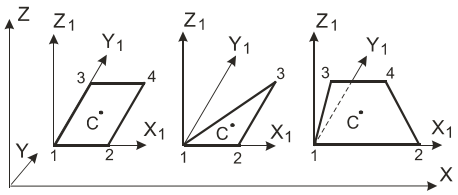


Figure 13.6-9. *Plates and shells*

The same coordinate system as for plates and shells should seemingly be selected for deep beam FE. However, taking into account that the deep beam elements are usually located in the XOZ plane (and in the planes parallel to it) of the global coordinate system, such a system is assumed where the axes X_1 and Z_1 lie in the plane of the element, and the Y_1 axis is perpendicular to its plane (Fig. 13.6-10); i. e., if it is compared to plates and shells, the Z_1 and Y_1 axes are interchanged.

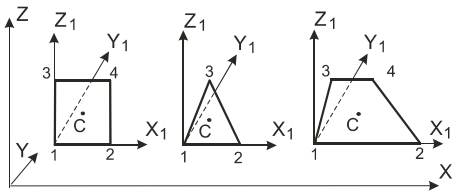


Figure 13.6-10. *Elements for solving the plane problem of the theory of elasticity (deep beams)*

Forces and stresses can be calculated in the coordinate system different from the local one for all flat finite elements. The rules for specifying these systems are described in Sec. 3.4. Their use is convenient for further analysis of the results of the calculation.

The finite element of type 22 (triangular deep beam) lying in the plane parallel to XOZ has a local coordinate system coinciding with the global one.

Group of Forces (Stresses) in Flat Elements

Forces (stresses) in flat finite elements are usually calculated for the center of gravity of an element (point C in Fig. 13.6-9 and 13.6-10), although forces (stresses) can be calculated in nodes. If a special coordinate system is not specified, they are calculated in the local coordinate system of the element. The

calculated forces and stresses, as well as the directions of the degrees of freedom of their nodes are given in Table 13.6-3.

Table 13.6-3

FE type[1]	Short name	Plane (or the one parallel to it)	Displacements of nodes	Forces, stresses
Plate finite elements				
11, 13 12, 14, 15 19, 20	Rectangular Triangular Quadrangular *(All elements can be on the elastic subgrade)*	XOY	Z, U_X, U_Y	M_X, M_Y, M_{XY}, Q_X, Q_Y, (R_Z)
Deep beam finite elements				
21 22 23, 26 24, 25, 28 30 27	Rectangular Triangular Rectangular Triangular Quadrangular Quadrangular	XOZ XOZ arbitrary arbitrary XOZ arbitrary	X, Z X, Z X, Y, Z X, Y, Z X, Z X, Y, Z	N_X, N_Y[2], N_Z, τ_{XZ}
Shell finite elements				
41 42, 45 44, 50	Rectangular Triangular Quadrangular *(All elements can be on the elastic subgrade)*	arbitrary	X, Y, Z, U_X, U_Y, U_Z	N_X, N_Y, τ_{XY}, M_X, M_Y, M_{XY}, Q_Y, Q_X, (R_Z)

The calculated forces and stresses are described in more detail in Table 13.6-4, which also provides the sign conventions for them.

Table 13.6-4

FE type	Designation of the force factor	Units of measurement (in the basic system of units)	Description	Sign convention
Membrane stresses				
21–30 41–50	$N_X(\sigma_x)$	T/m^2	Normal stress, acting along the X_1 axis	Positive sign corresponds to tension
21–30 41–50	$N_Y(\sigma_y)$	T/m^2	Normal stress, acting along the Y_1 axis	Positive sign corresponds to tension *(For elements 21–30 it is calculated only for plane-strain systems)*
21–30	$N_Z(\sigma_z)$	T/m^2	Normal stress, acting along the Z_1 axis	Positive sign corresponds to tension
41–50	$T_{XY}(\tau_{xy})$	T/m^2	Shear stress	Shear of the section of the element in the directions opposite to the X_1 and Y_1 axes
21–30	$T_{XZ}(\tau_{xz})$	T/m^2	Shear stress	Shear of the section of the element in the directions opposite to the X_1 and Z_1 axes
Bending stresses (forces)				
11–20 41–50	M_X	Tm/r.m	Moment acting on the section orthogonal to the X_1 axis	Positive moment causes tension of the lower (with respect to the Z_1 axis) fiber of the section
	M_Y	Tm/r.m	Moment acting on the section orthogonal to the Y_1 axis	
	M_{XY}	Tm/r.m	Torque (acting on the section orthogonal to the X_1 axis).	Counterclockwise rotation of the section of the element if viewed from the end of the X_1 axis

[1] Two last numbers of the element type are given in the **FE type** column.
[2] Force N_Y appears in the calculations of plane-strain systems.

FE type	Designation of the force factor	Units of measurement (in the basic system of units)	Description	Sign convention
	Q_X	T/r.m	Shear force in the section orthogonal to the X_1 axis	Positive shear force acts in the direction of the Z_1 axis in the part of the FE without node 1
	Q_Y	Tm/r.m	Shear force in the section orthogonal to the Y_1 axis	
Reactive earth pressure				
11–30 41–50	R_X	T/m^2	Reactive earth pressure at the calculation of plates and shells on the elastic subgrade	Positive force acts in the direction of the X_1 axis (the minus sign indicates that the soil is compressed)
11–20 41–50	R_Y	T/m^2	Reactive earth pressure at the calculation of plates and shells on the elastic subgrade	Positive force acts in the direction of the Y_1 axis (the minus sign indicates that the soil is compressed)
21–30 41–50	R_Z	T/m^2	Reactive earth pressure at the calculation of plates and shells on the elastic subgrade	Positive force acts in the direction of the Z_1 axis (the minus sign indicates that the soil is compressed)

Moments and shear forces are determined per unit length in any system of units (for example, 1 meter) regardless of the specified sizes of the sides of the elements (0.001 m or 10.000 m).

Sign Conventions for Forces (Stresses) in Flat Finite Elements

For each point for the calculation of forces (stresses) the following two sections passing through it are actually considered:

- perpendicular to the X_1 axis;
- perpendicular to the Y_1 axis.

It is clear that internal forces (forces, stresses) in any section are balanced (directed in opposite directions).

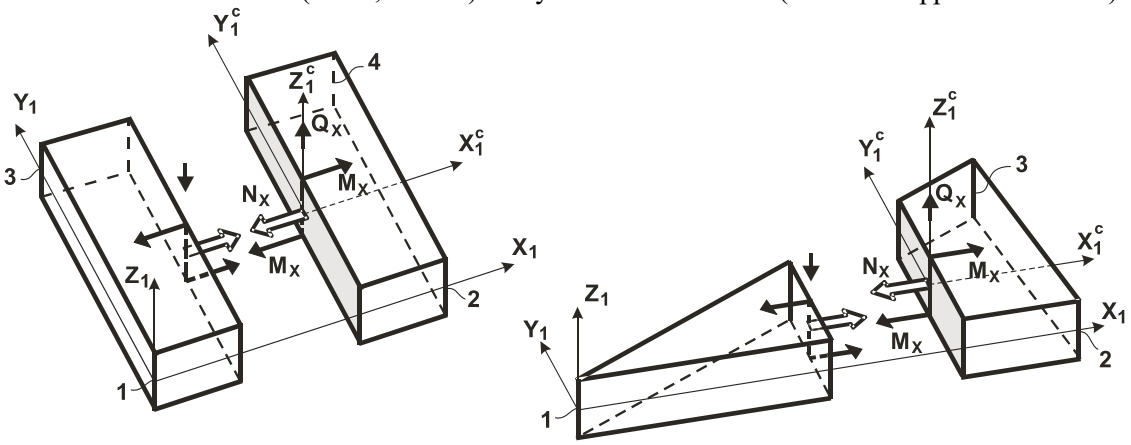


Figure 13.6-11.

In order to output the results of the calculation it is necessary to select one of the planes of the section. The plane of the section belonging to the part of the element which includes the last node is selected in the given examples (Fig. 13.6-11, 13.6-12, 13.6-13). Thus, those directions of the group of forces (stresses) are selected with which the dropped (with node 1) part of the plate acts on its second part — the one which includes the last node (node 3 for triangular, or node 4 for quadrangular elements).

Components of forces acting on the selected plane are calculated in the first section (Fig. 13.6-11):

N_X — normal stress in the section (T/m^2) from tension-compression of shell elements; positive N_X cause tension of the section in the direction parallel to the X_1 axis;

M_X — bending moment per unit length of the section (Tm/r.m), positive M_X causes compression in the upper (with respect to the direction of the Z_1 axis) and tension in the lower fiber of the section in the direction parallel to the X_1 axis;

Q_X — shear force per unit length of the section (T/r.m), positive direction of Q_X coincides with the direction of the Z_1 axis.

The following values are calculated in the second section (Fig. 13.6-12):

N_Y — normal stress in the section (T/m^2) from tension-compression of shell elements; positive N_Y cause tension of the section in the direction parallel to the Y_1 axis;

M_Y — bending moment per unit length of the section (Tm/r.m), positive M_Y causes compression in the upper (with respect to the direction of the Z_1 axis) and tension in the lower fiber of the section in the direction parallel to the Y_1 axis;

Q_Y — shear force per unit length of the section (T/r.m), positive direction of Q_Y coincides with the direction of the Z_1 axis.

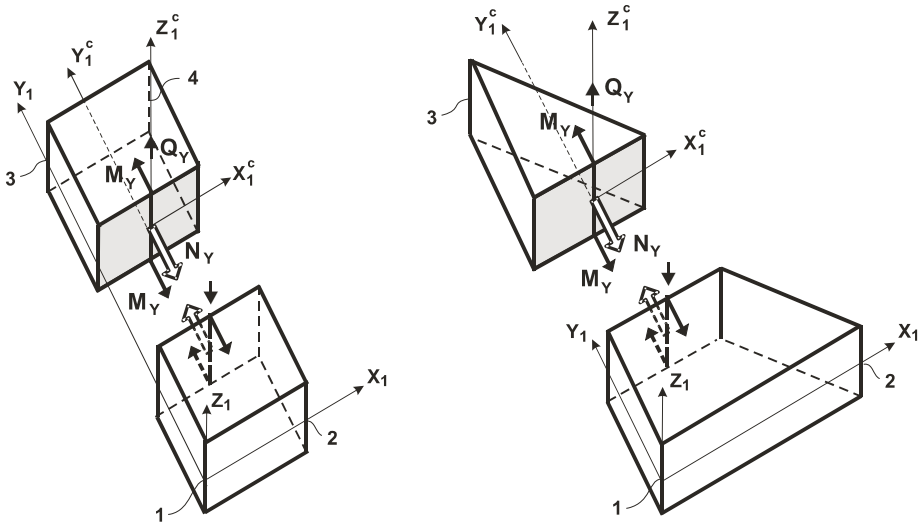


Figure 13.6-12.

Pairs of forces and stresses are also calculated in each of these sections (Fig. 13.6-13):

T_{YX}, equal to T_{XY}, — shear stresses (T/m^2), positive direction of shear is opposite to the direction of the Y_1 axis; positive shear stresses tend to stretch a section of that part of the element which includes the last node (diagonal of the square coming from the node 4 in a square finite element);

M_{XY}, equal to M_{YX}, — torque (Tm/r.m); positive M_{XY} causes compression in the upper (with respect to the Z_1 axis) and tension in the lower fiber of the section in the direction parallel to the Y_1 axis, i. e. the moment twists the section belonging to the end of the element clockwise if the section is viewed in the direction of the X_1 axis, and counterclockwise if it is viewed from the end of the X_1 axis.

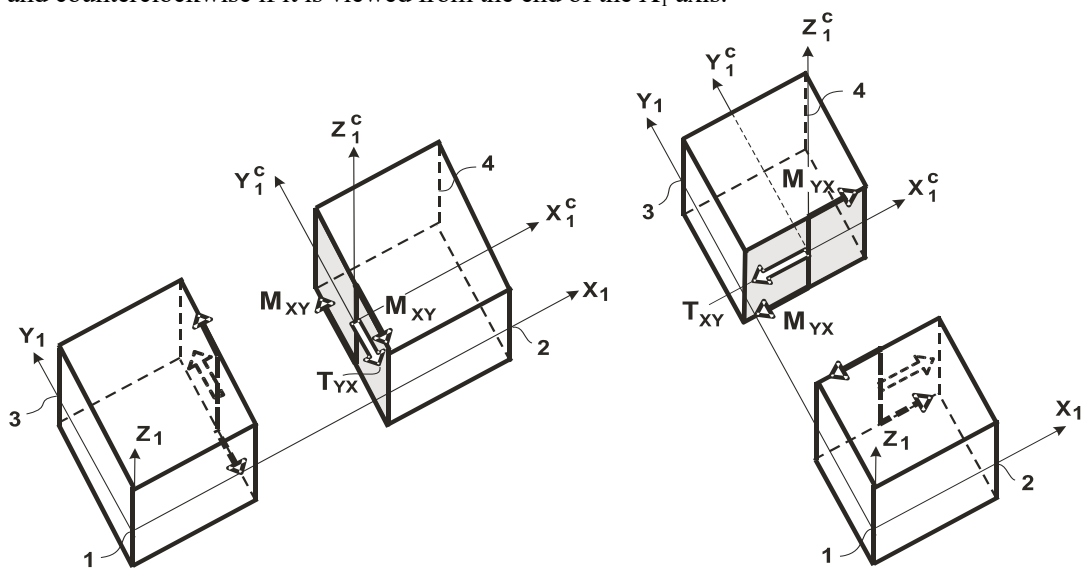


Figure 13.6-13.

The same sections are used for deep beam finite elements (Fig. 13.6-14), only the stresses N_Y and T_{XY}, corresponding to shell elements should be replaced by N_Z and T_{XZ}, corresponding to the deep beam.

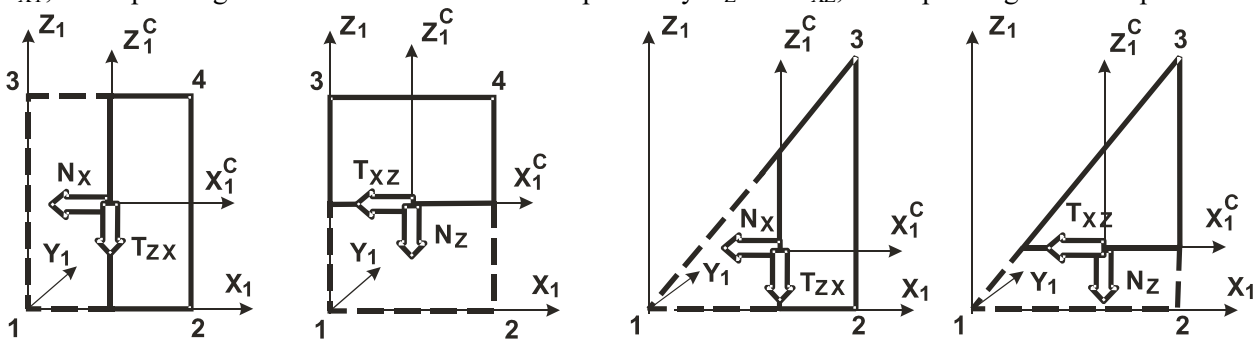


Figure 13.6-14.

Forces and stresses are always calculated for the center of gravity of an element by default, and if specified, they can also be calculated in its nodes.

The action of forces on flat finite elements can be illustrated as follows (Fig. 13.6-15 and Fig. 13.6-16.):

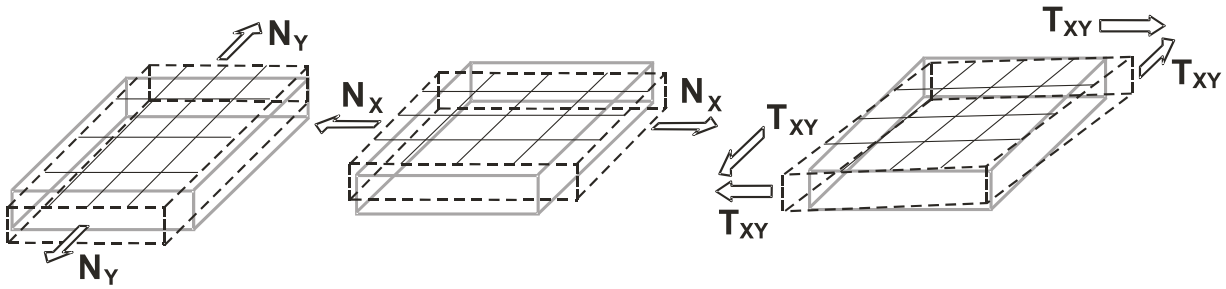


Figure 13.6-15. *Membrane group of forces*

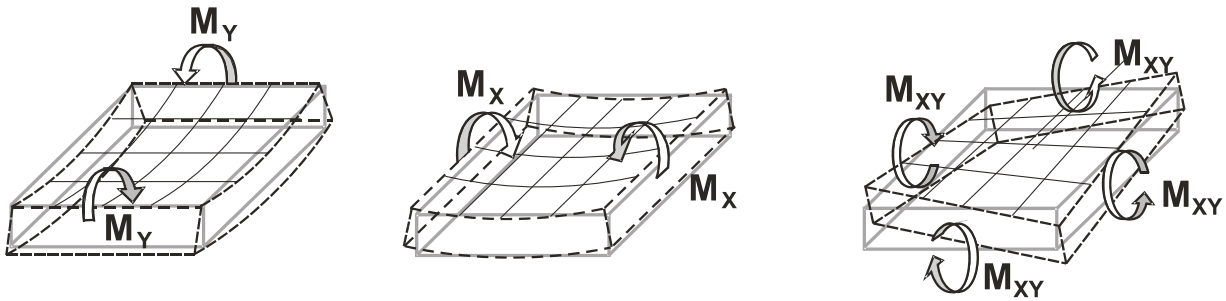


Figure 13.6-16. *Bending group of forces*

Positive Directions of Stresses in Deep Beam Finite Elements

The following stresses are calculated in the finite elements intended for the analysis of plane-stress (deep beams) or plane-strain systems (Sec. 3.4):

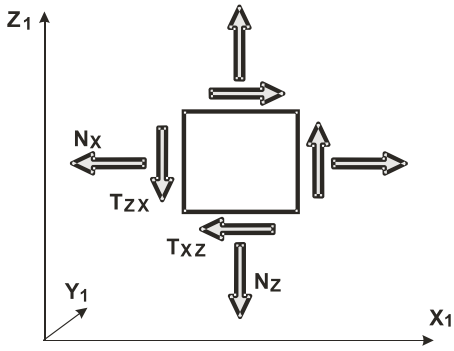


Figure 13.6-17.

- normal stresses N_X, N_Z;
- shear stress T_{XZ};
- normal stress N_Y for plane-strain systems.

Positive directions of N_X and N_Z cause tension in the sections of a deep beam passing through the center of gravity of the element perpendicular to the X_1 and Z_1 axes, respectively. Direction of stresses is assumed for the part of the section which includes the last of the main (3-rd — for triangular or 4-th — for quadrangular) nodes, i. e. the action of the dropped (with node 1) part on the remaining (with nodes 3 or 4) part of the element is determined.

If tension or compression in the element is only parallel to the axes X_1 and Z_1, T_{XZ} is always equal to zero, i. e. sections perpendicular to the axes X_1 and Z_1 in this case coincide with the principal planes.

Figure 13.6-17 shows stresses acting on the faces of an elementary rectangle cut in the vicinity of the center of gravity of a deep beam finite element, and the rules for reading the stresses are given in Table 13.6-4.

It should also be noted here that when solving a plane-strain problem not only the stresses N_X, N_Z, T_{XZ} acting in the plane of the element are output, but also the value of the stress N_Y normal to the plane which acts in the direction orthogonal to the plane of the element, and its positive sign indicates tension.

Positive Directions of Forces in Plate FE

The following forces are calculated for plate finite elements:

- moments M_X, M_Y and M_{XY};
- shear forces Q_X and Q_Y;
- reactive earth pressure R_Z (only if the elastic subgrade is taken into account).

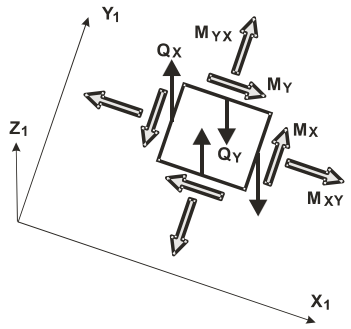


Figure 13.6-18.

Positive moments M_X, M_Y, M_{XY} cause tension in the lower (with respect to the direction of the Z_1 axis) fiber of sections passing through the center of gravity of the FE perpendicular to the X_1 and Y_1 axes respectively. Direction of forces M_X, M_Y, M_{XY}, Q_X, Q_Y is assumed for the part of the section which includes the last of the main (3-rd — for triangular or 4-th — for quadrangular) nodes of the element, i. e. the action of the dropped (with node 1) part on the remaining part of the element is determined. Positive direction of shear forces Q_X and Q_Y coincides with the direction of the Z_1 axis.

Figure 13.6-18 shows shear forces and vectors of moments acting on the faces of an elementary rectangle cut in the vicinity of the center of gravity of a plate element, and the rules for reading the forces are given in Table 13.6-4.

Positive Directions of Stresses and Forces in Shell FE

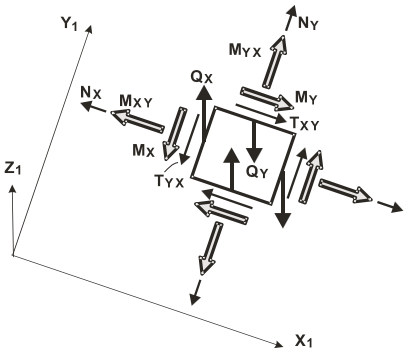


Figure 13.6-19.

The following forces are calculated in shell finite elements:

- normal stresses N_X, N_Y;
- shear stress T_{XY};
- moments M_X, M_Y and M_{XY};
- shear forces Q_X and Q_Y;
- reactive earth pressure R_Z (only if the elastic subgrade is taken into account).

A shell element combines both properties of deep beam and plate elements, therefore all the above explanations apply to this section as well.

Figure 13.6-19 shows stresses, shear forces and vectors of moments acting on the faces of an elementary rectangle cut in the vicinity of the center of gravity of a shell element, and the rules for reading the stresses and forces are given in Table 13.6-4.

Solid Finite Elements

If the coordinate system for the stress output is not specified, forces are calculated in the local coordinate system for the finite elements of types 31 (parallelepiped), 33 and 35 (triangular prisms), and in the global coordinate system — for the elements of types 32, 34, 36, 37 and 38 (Sec. 3.5). Elements of type 31, 33 and 35 have a local coordinate system (Fig. 13.6-20), where:

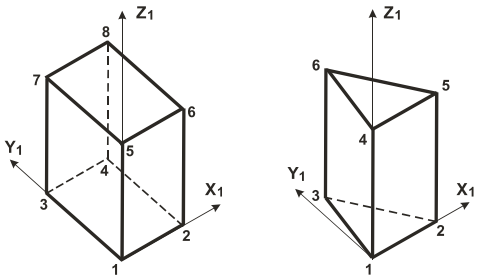


Figure 13.6-20.

- the X_1 axis is directed from node 1 to node 2;
- the Y_1 axis lies in the plane formed by the first three nodes and is directed towards the third node;
- the Z_1 axis makes a right-hand system with the X_1 and Y_1 axes.

By default stresses are calculated only in the centers of gravity of finite elements of the spatial problem of the theory of elasticity. They can also be calculated in the nodes of the elements with the help of the graphical input.

Normal N_X, N_Y, N_Z and shear T_{XY}, T_{XZ}, T_{YZ} stresses are calculated.

Figure 13.6-21 shows positive directions of stresses, names of axes, along which they are directed, and planes in which they act. Table 13.6-5 provides rules for reading the results, and designations of the pairs of stresses are given in brackets. By default stresses are considered with respect to the local coordinate system $X_1Y_1Z_1$ for the elements of types 31, 33 and 35, and with respect to the global coordinate system XYZ — for the elements of types 32, 34, 36, 37 and 38.

Table 13.6-5

Designation of the force factor	Units of measurement (in the basic system of units)	Description	Positive sign of the force
N_X	T/m^2	Normal stress acting along the X_1 axis	Indicates tension
N_Y	T/m^2	Normal stress acting along the Y_1 axis	Indicates tension
N_Z	T/m^2	Normal stress acting along the Z_1 axis	Indicates tension
T_{XY}	T/m^2	Shear stress acting in the plane parallel to X_1OY_1	
T_{XZ} (T_{ZX})	T/m^2	Shear stress acting in the plane parallel to X_1OZ_1	
T_{YZ} (T_{ZY})	T/m^2	Shear stress acting in the plane parallel to Y_1OZ_1	

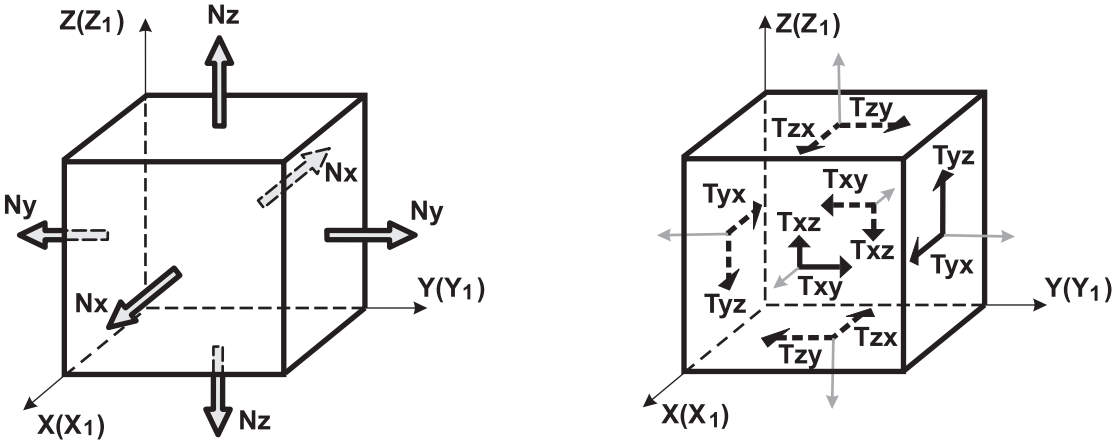


Figure 13.6-21.

Multilayer Shell Elements

These finite elements are used for plates of medium thickness and laminated structure (see Sec. 3.7), when it is necessary to take into account:

- the deformations of transverse shear, compression of layers and curvature (elements of types 71–80);
- the interlayer shear and curvature (elements of types 81–90).

Since it is impossible to describe the nonlinear nature of variation of displacements across the thickness of the package of layers in the system of the usual six degrees of freedom in the node, their six equivalents are added (A_X, A_Y, A_Z , B_X, B_Y, B_Z — for types 71–80 and A_X, A_Y, A_Z , C_X, C_Y, C_Z — for types 81–90). Each node has 12 degrees of freedom.

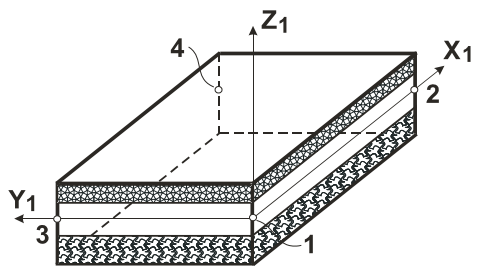


Figure 13.6-22.

The local coordinate system for multilayer shells is constructed in the same way as for single-layer ones:

- the X_1 axis is directed from node 1 (beginning of the FE) to node 2;
- the Y_1 axis is perpendicular to the X_1 axis and is directed towards node 3 (for a rectangle — into node 3);
- the Z_1 axis is perpendicular to the plane of the element and is directed from node 1 to form a right-hand coordinate system.

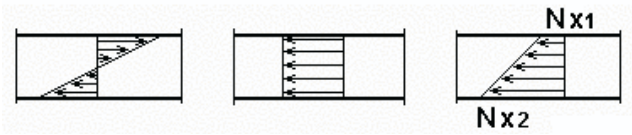
Values given in Table 13.6-6 are determined in the result of the calculation for each layer on its upper and lower boundaries in the center of gravity (and if specified, in the nodal points as well).

Table 13.6-6

Designation of the force factor	Units of measurement (in the basic system of units)	Description	Positive sign of the force
N_X	T/m^2	Stress on the plane of the lower or upper surface of the layer acting along the X_1 axis	Indicates tension
N_Y	T/m^2	Stress on the plane of the lower or upper surface of the layer acting along the Y_1 axis	Indicates tension
N_Z	T/m^2	Stress normal to the surface of the layer acting along the Z_1 axis	Indicates tension
T_{XY}	T/m^2	Shear stress acting in the plane parallel to X_1OY_1	
T_{XZ}	T/m^2	Shear stress acting in the plane parallel to X_1OZ_1	

Designation of the force factor	Units of measurement (in the basic system of units)	Description	Positive sign of the force
T_{YZ}	T/m^2	Shear stress acting in the plane parallel to Y_1OZ_1	
W_Z	mm	Values of horizontal displacements and deflections in the directions of the axes of the global coordinate system	Positive displacements coincide with the directions of the Z axis

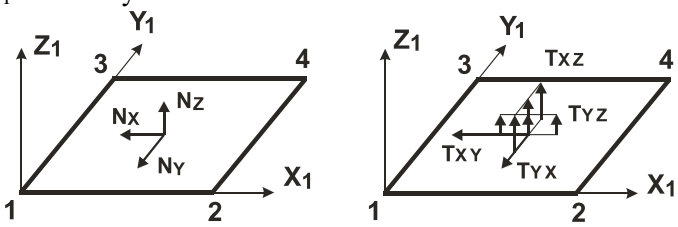
Stresses N_X, N_Y, T_{XY} are directed parallel to the planes of the layer and local axes X_1 and Y_1, they can be treated as stresses from the membrane and bending groups of forces in this layer. Thus, for example, stresses N_X (Fig. 13.6-23, *c*) are the total stresses from the bending moment M_Y (Fig. 13.6-23, *a*) and the longitudinal force N_X (Fig. 13.6-23, *b*) in this layer, and T_{XY} are the total stresses from the torque M_{XY} and the shear force T_{XY}.



a) *b*) *c*)

Figure 13.6-23.

Stresses N_Z, T_{XZ}, T_{YZ} are directed perpendicular to the planes of the layer and parallel to the Z_1 axis; Nz characterizes the compression of the layers along the vertical; T_{XZ} and T_{YZ} are the measure of intensity of shear forces Q_X and Q_Y in this layer.



a) *b*)

Figure 13.6-24.

Positive directions of stresses for the upper plane of the layer are shown in Fig. 13.6-24. Fig. 13.6-24, *a* shows positive directions of normal stresses, and Fig. 13.6-24, *b* shows positive directions of shear stresses. The direction of N_Z is reversed for the lower plane.

Axisymmetric Finite Elements

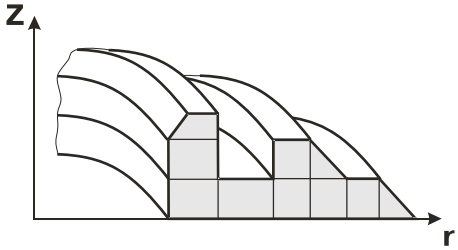


Figure 13.6-25.

These elements enable to determine the stress-strain state of the solids of revolution under the action of the axisymmetric load (Sec. 3.6).

The cross-section of the solid is divided into elements and is described in the rOZ coordinate system, where each node has 2 degrees of freedom in the directions of the r and Z axes (Fig. 13.6-25). Values of displacements of nodes and stresses are calculated in the global coordinate system rOZ.

Figure 13.6-26 shows positive directions of stresses, and Table 13.6-7 provides rules for reading them.

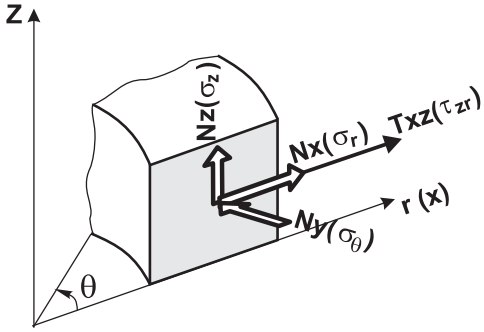


Figure 13.6-26.

Table 13.6-7

Designation of the force factor	Units of measurement (in the basic system of units)	Description	Positive sign of the force indicates
N_X (σ_r)	T/m^2	Stress acting in the plane of the element and directed along the r axis	Tension
N_Z (σ_z)	T/m^2	Stress acting in the plane of the element and directed along the Z axis	Tension
T_{XZ} (τ_{zr})	T/m^2	Shear stress	Shear stress in the plane of the section
N_Y (σ_θ)	T/m^2	Stress normal to the plane of the element	Tension

Special Finite Elements

Special finite elements (Sec. 3.8) include: “null element” (FE of type 154); elements modeling elastic constraints in nodes and between nodes (types 51 and 55), and two-node and single-node peripheral elements of elastic subgrade (types 53 and 54).

General Type “Null Element” (Type 154)

Forces in a “null element” (Fig. 13.6-27) are calculated in the local coordinate system constructed according to the same rules as those for the coordinate system of bar elements.

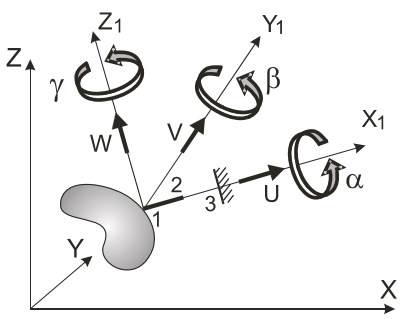


Figure 13.6-27.

Despite the fact that the local axes of the element are constructed similarly to those of the bar, a “null element” should not be confused with a bar. It “adjusts” to the specified type of the model automatically taking into account the number of the degrees of freedom, specified stiffness properties, and calculated forces.

Table 13.6-8 provides a list of calculated forces for different types of the model which are actually reactions in its first node.

Table 13.6-8

Model type	Force		Direction of the force action*	Positive sign of the force indicates*
	Index	Units of measurement		
1, 2, 4, 5, 8, 9, 11	R_X	T	Along the X_1 axis	Coincidence with the direction of the X_1 axis
4, 5, 8, 9	R_Y	T	Along the Y_1 axis	Coincidence with the direction of the Y_1 axis
1, 2, 4, 5, 8, 9, 11	R_Z	T	Along the Z_1 axis	Coincidence with the direction of the Z_1 axis

Model type	Force		Direction of the force action*	Positive sign of the force indicates*
5, 8, 9	R_{UX}	Tm	About the X_1 axis	Clockwise rotation if viewed from the end of the X_1 axis
5, 8, 9	R_{UY}	Tm	About the Y_1 axis	Clockwise rotation if viewed from the end of the Y_1 axis
5, 8, 9	R_{UZ}	Tm	About the Z_1 axis	Clockwise rotation if viewed from the end of the Z_1 axis
8, 9, 11	R_{AX}	T	Along the X_1 axis	Coincidence with the direction of the X_1 axis
8, 9	R_{AY}	T	Along the Y_1 axis	Coincidence with the direction of the Y_1 axis
8, 9	R_{AZ}	T	Along the Z_1 axis	Coincidence with the direction of the Z_1 axis
8	R_{BX}	Tm	About the X_1 axis	Clockwise rotation if viewed from the end of the X_1 axis
8	R_{BY}	Tm	About the Y_1 axis	Clockwise rotation if viewed from the end of the Y_1 axis
8	R_{BZ}	Tm	About the Z_1 axis	Clockwise rotation if viewed from the end of the Z_1 axis
9	R_{CX}	T	Along the X_1 axis	Coincidence with the direction of the X_1 axis
9	R_{CY}	T	Along the Y_1 axis	Coincidence with the direction of the Y_1 axis
9	R_{CZ}	T	Along the Z_1 axis	Coincidence with the direction of the Z_1 axis
* For elements modeling the elastic constraints (type 51 — always and type 55 — provided that the coordinates of nodes coincide), the direction of the action of forces and the positive sign of forces are determined in the global coordinate system.				

Elements Modeling Elastic Constraints (Type 51, 55)

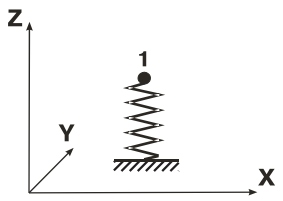


Figure 13.6-28.

Element of type 51 (Fig. 13.6-28) models the constraint of finite rigidity in the directions of the degrees of freedom of its single node (in the global coordinate system).

Forces (reactions in springs) corresponding to the degrees of freedom in the directions of which the constraints of finite rigidity are imposed (R_X, R_Y, R_Z, R_{UX}, R_{UY}, R_{UZ}) are determined in the result of the calculation. Rules for reading them are given in Table 13.6-8. When, for example, the elastic subgrade is modeled, it will be the earth pressure in the given direction.

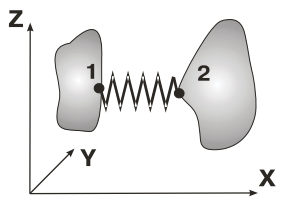


Figure 13.6-29.

Element of type 55 (Fig. 13.6-29) enables to take into account the compliance of the material between the two nodes.

Reaction components (R_X, R_Y, R_Z, R_{UX}, R_{UY}, R_{UZ}) in the global or local (in the case of its specification) coordinate systems are determined in the result of the calculation. It is assumed that these reactions are external actions of the system on the first node of the element. Rules for reading them are given in Table 13.6-8. It should be noted that

in the global coordinate system the sign of the output forces changes when the node numbering order is changed;

in the local coordinate system the sign of the longitudinal reaction RX does not depend on the ode numbering order and its positive value can be interpreted as the compression of the element.

In the case when an element has a zero length (the coordinates of nodes coincide), the reactions are always output in the global coordinate system.

Elements Modeling Peripheral Elastic Subgrade (Type 53, 54)

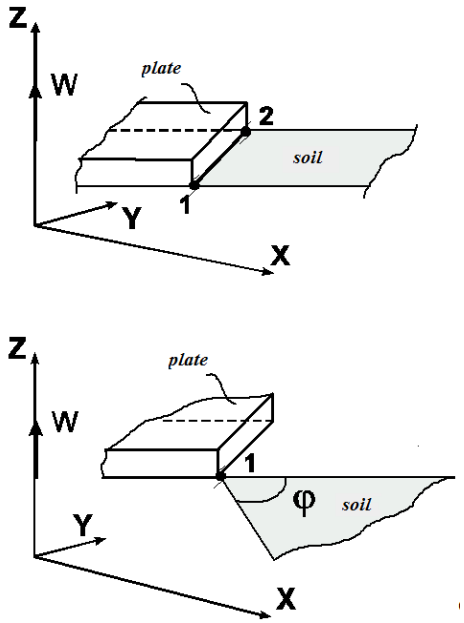


Figure 13.6-30.

Elements of types 53 and 54 (Fig. 13.6-30) are used for modeling the pressure of the strip and the angular area of soil beyond the plate and perpendicular to its contour (due to the shear resistance of soil). Resistance of soil is considered only in the direction of the Z axis.

Earth pressure R_Z in the nodes of the element which are considered as its sections is determined in the result of the calculation. Earth pressure is a nodal reaction and shows how much of the action the soil strip can take beyond the plate. Rules for reading them are given in Table 13.6-8. The positive value of R_Z indicates the compression of soil.

13.7. Reactions in Constraints

SCAD enables to output the values of reactions in nodes with constraints. In order to calculate the reactions in constraints it is necessary to check the **Calculate reactions in constraints** checkbox in the **Calculation Parameters** dialog box.

The positive values of reactions in constraints are directed oppositely to the positive values of the respective components of the nodal load (active forces).

Tables with reactions are created in the **Documenter** and are similar to the tables with the values of forces.

13.8. Natural Oscillations

The frequency of each oscillation mode taken into account in the dynamic loading is always indicated in the tables. If necessary the relative values of ordinates in the directions of the degrees of freedom in nodes (in the global coordinate system) can be also shown, as well as the actions equivalent to statics in the form of dynamic loads (T), applied in the nodes of the design model.

Eigenvalues, Frequencies, Periods of Oscillations

Figure 13.8-1 provides an example of the output of eigenvalues, frequencies and periods of oscillations for several dynamic loadings of the considered problem. Each line contains the following information: number of the loading, serial number of the oscillation mode, eigenvalues calculated for this mode, frequencies and periods, and the percentage of the accumulated modal masses in the directions. The total values of modal masses accumulated in the considered loading are given in the line marked with the S index in the *Mode* column.

Frequencies of natural oscillations
Designations: S - Sum of modal masses

Frequencies of natural oscillations								
Loading	Mode	Eigenvalue	Frequency		Periods (sec)	Modal masses (%)		
			1/sec	Hz		Mx	My	Mz
12	1	0,386	2,591	0,413	2,424	79,733	0	0
12	2	0,125	8,013	1,276	0,784	10,459	0	0
12	3	0,07	14,231	2,266	0,441	3,669	0	0
12	S					93,861	100	0
13	1	0,386	2,591	0,413	2,424	79,733	0	0
13	2	0,125	8,013	1,276	0,784	10,459	0	0
13	3	0,07	14,231	2,266	0,441	3,669	0	0
13	S					93,861	100	0
14	1	0,329	3,037	0,484	2,068	79,729	0	0
14	2	0,106	9,4	1,497	0,668	10,473	0	0
14	3	0,06	16,714	2,661	0,376	3,669	0	0
14	S					93,87	100	0
15	1	0,329	3,037	0,484	2,068	79,729	0	0
15	2	0,106	9,4	1,497	0,668	10,473	0	0
15	3	0,06	16,714	2,661	0,376	3,669	0	0
15	S					93,87	100	0

Figure 13.8-1. *Frequencies of natural oscillations*

Natural Modes

The ratios between the values of amplitudes in the nodes of the design model for each degree of freedom in the node are given in the tables for each of the considered natural oscillation modes. The maximal value of amplitude is specified as 1000, the values of other amplitudes are determined in fractions of 1000. The tables with natural oscillation modes and their minimax values are similar to the respective tables created when outputting the displacements.

Distribution of the Weights of Masses

The distribution of the weights of masses (Fig. 13.8-2) indicates, for example, the way the masses of the structure were distributed in the concentration nodes. The tables of the distribution of the weights of masses are similar to the tables of displacements.

Distribution of the weights of masses
Units of measurement: T, m
Parameters of selection:
List of nodes/elements: All
List of loadings/combinations: All
List of factors: All

Distribution of the weights of masses							
Node	Loading	Value					
		P_X	P_Y	P_Z	P_{UX}	P_{UY}	P_{UZ}
5	12	25,982	0	25,982	0	0	0
5	13	25,982	0	25,982	0	0	0
5	14	19,184	0	19,184	0	0	0
5	15	19,184	0	19,184	0	0	0

Figure 13.8-2. *Distribution of the weights of masses*

Accelerations

When performing the dynamic calculations the accelerations in the nodes of the design model are output in the tables (Fig. 13.8-3).

Accelerations
Units of measurement: mm/sec^2
Parameters of selection:
List of nodes/elements: 562-600
List of loadings/combinations: All
List of factors: All

Accelerations						
Node	Loading	Mode number	Value			
			a_x	a_y	a_z	a_{sum}
562	13	RD1	0,007	0,003	-8,344e-005	0,008
562	13	RI1	4,465e-005	2,949e-005	-7,783e-006	5,407e-005
562	13	S1	0,007	0,003	8,38e-005	0,008
563	13	RD1	0,009	0,002	0,002	0,01
563	13	RI1	4,867e-005	2,605e-005	4,719e-006	5,54e-005
563	13	S1	0,009	0,002	0,002	0,01
564	13	RD1	0,009	0,004	0,005	0,011

Figure 13.8-3. *Fragment of the "Accelerations" table*

Here a_x, a_y, a_z — accelerations in the directions of the x, y, z axes of the global coordinate system respectively,

$$a_{sum} = \sqrt{a_x^2 + a_y^2 + a_z^2}\ .$$

13.9. Design Combinations of Forces (DCF)

The DCF postprocessor selects the combinations of individual loadings which can be the most dangerous for each section of the checked elements of the design model. The selection of each combination is determined by its strength criterion. Forces (stresses) corresponding to the determined dangerous combinations are used by the **SCAD** postprocessors and program-satellites (**Kristall**, **ARBAT**) to take into account the requirements of the respective codes to the elements of steel and reinforced concrete structures. An example of printing the DCF is shown in Fig. 13.9-1.

The following information is given in the columns of the table:

- **UNG** — code of the unified group;
- **Element number**;
- **Number of the section**, for which the combination is selected;
- **CL** — number of the column of coefficients the forces (stresses) of each loading were multiplied by when included in the current loading;
- **Criterion** — number of the criterion (No.) and its value according to which the given combination was selected. Numbers of the criteria and formulas for calculating their values are given in Chapter 18;
- **Type**— type of the structure according to Table P.4.1 of the Design Codes for Nuclear Plants NP-031-01 (see Chapter 18);
- **Value** (of the force and stress) — columns with the values of force factors of the current combination;
- **Type** (of the combination) — **C, CL, N, NL** identifiers in this column define the type of combination given in the line (C — design; CL — design long-term, N — characteristic, NL — characteristic long-term);
- **CS** (crane, seismic, special, transport) — characters written in this column designate:
  - **Cr** — the combination includes loadings from crane actions;
  - **Se** — the combination includes the loading from the seismic action;
  - **Sp** — the combination includes a special loading;
  - **Tr** — the combination includes loadings from vehicles;
  - **space** — the combination does not include loadings with crane, seismic and other actions...;
- **Formula** — expression showing the serial numbers of loadings and combinations of loadings included in the current combination, and coefficients with which they are taken.

DCF with an automatic selection of coefficients
Units of measurement: T, m
Parameters of selection:
List of nodes/elements: 9263
List of factors: All
List of types of combinations: All
Designations:
C - design;
CL - design long-term;
N - characteristic;
NL - characteristic long-term;
Se - seismic;
Cr - crane;
Sp - special;
Tr - transport.

DCF with an automatic selection of coefficients											
UNG	Elem.	Sec.	CL	Crit.	Type	Value			Type	CS	Formula
						N	M_y	Q_z			
--	67	1	1	1	1	-0,758	-1,234	0,801	C		L1 +L2
--	67	1	1	1	1	-0,758	-1,234	0,801	CL		L1 +L2
--	67	1	1	1	1	-0,454	-1,111	0,721	N		0,909*L1 +0,952*L2
--	67	1	1	1	1	-0,454	-1,111	0,721	NL		0,909*L1 +0,952*L2

Figure 13.9-1. *Fragment of the "Design combinations of forces" table (DCF)*

If the unification is specified at the calculation of DCF, the results are printed for the unification groups at the beginning of the table. The following rules are assumed for the designation of the unified groups in the UNG column:

- the first number indicates the type of unification;
- the second number indicates the number of the unified group.

The next columns contain the information on the combinations with the indication of the numbers of elements which were established as the most dangerous for the specified criterion for all elements of the unification group. Figure 13.9-2 shows a fragment of the table of combinations for elements included in the unified group 1.

DCF with an automatic selection of coefficients											
UNG	Elem.	Sec.	CL	Crit.	Type	Value			Type	CS	Formula
						N	M_y	Q_z			
2-5	2	1	1	1	1	-2,154	-4,614	1,73	C		L1 +L2
2-5	2	1	1	2	1	-3,782	-8,25	3,156	C		C1
2-5	2	1	1	23	1	-2,185	-7,602	2,554	C		L1 +L2 +L4
2-5	2	1	1	24	1	-3,75	-5,262	2,333	C		L1 +L2 +L3

Figure 13.9-2. *Fragment of the table of combinations with a unified group*

If EN 1990 is selected for calculating load combinations, the following unfavorable combinations are calculated: fundamental, accidental, accidental for fire, post-accidental, seismic, characteristic, frequent, and quasi-permanent. Detailed data are given on the formula for the calculation of fundamental combinations according to EN 1990. The **Type** (of combinations) column can contain the following identifiers:

F - Fundamental combination (formula (6.10))
Fa - Fundamental combination (formula (6.10.a))
Fb - Fundamental combination (formula (6.10.b))
A - Accidental combination (formula (6.11a))
Af - Accidental combination for fire (formula (6.11b))
PA - Post-accidental combination (formula (6.11.3))
S - Seismic combination (formula (6.12))
CH - Characteristic combination (formula (6.14))
FR - Frequent combination (formula (6.15))
QP - Quasi-permanent combination (formula (6.16))

13.10. DCF at the Analysis of the Progressive Collapse

The columns of the table with the values of the design combinations at the analysis of the progressive collapse are the same as those of the DCF table. Figure 13.10-1 shows a fragment of the table of combinations.

DCF for progressive collapse																	
UNG	Elem.	Sec.	CL	Crit.	Type	Value									Type	CS	Formula
						N_X	N_Y	τ_{XY}	M_X	M_Y	M_{XY}	Q_X	Q_Y	R_z			
--	1805	1	1	0	1	6,561	17,611	3,559	-0,068	-0,023	0,14	0,474	0,202	0	C		L1
--	1805	1	1	0	1	6,561	17,611	3,559	-0,068	-0,023	0,14	0,474	0,202	0	CL		L1
--	1806	1	1	0	1	1,7	10,996	8,062	-0,142	0,029	0,149	-0,408	-0,475	0	C		L1
--	1806	1	1	0	1	1,7	10,996	8,062	-0,142	0,029	0,149	-0,408	-0,475	0	CL		L1
--	1807	1	1	0	1	1,258	7,624	9,47	-0,238	-0,022	0,109	-0,828	-0,095	0	C		L1
--	1807	1	1	0	1	1,258	7,624	9,47	-0,238	-0,022	0,109	-0,828	-0,095	0	CL		L1

Figure 13.10-1. *Fragment of the table of design combinations for the analysis of the progressive collapse*

13.11. Design Combinations of Reactions in Constraints

The columns of the table with the values of the design combinations of reactions in constraints are the same as those of the DCF table. Figure 13.11-1 shows a fragment of the table of combinations.

Design combinations of reactions in constraints
Units of measurement: T, m
Parameters of selection:
List of nodes/elements: 1 6 r 10 2
List of types of combinations: All
Designations:
C - design;
CL - design long-term;
N - characteristic;
NL - characteristic long-term;
Se - seismic;
Cr - crane;
Sp - special;
Tr - transport.

Design combinations of reactions in constraints											
Node	CL	Crit.	Value						Type	CS	Formula
			R_X	R_Y	R_Z	R_{UX}	R_{UY}	R_{UZ}			
6	1	4	-11,878	-8,811	47,23	27,901	-35,464	1,849	C		L1 +0,9*L16 +L18 +0,95*L23 +L24 +0,7*L28
6	1	4	-4,12	-2,838	53	9,22	-12,244	0,624	CL		L1 +0,315*L16 +0,35*L18 +0,95*L23 +L24 +0,245*L28
6	1	4	-9,888	-7,317	43,623	23,204	-29,506	1,541	N		0,909*L1 +0,75*L16 +0,833*L18 +0,792*L23 +0,833*L24 +0,583*L28
6	1	4	-3,422	-2,34	48,432	7,637	-10,157	0,521	NL		0,909*L1 +0,262*L16 +0,292*L18 +0,792*L23 +0,833*L24 +0,204*L28
6	1	5	-6,156	-9,741	58,269	31,174	-16,55	2,398	C		L1 +0,95*L2 +L18 +0,9*L19 +L22 +0,95*L25 +0,7*L27
6	1	5	1,403	-3,659	62,361	12,123	5,895	1,129	CL		L1 +0,95*L2 +0,35*L18 +0,315*L19 +L22 +0,95*L25 +0,245*L27

Figure 13.11-1. *Fragment of the "Design combinations of reactions in constraints" table*

The following numbering of the criteria is used for the design combinations of reactions in constraints:

No.	Criterion	No.	Criterion	No.	Criterion	No.	Criterion
1	*max* R_X	2	*min* R_X	3	*max* R_Y	4	*min* R_Y
5	*max* R_Z	6	*min* R_Z	7	*max* R_{UX}	8	*min* R_{UX}
9	*max* R_{UY}	10	*min* R_{UY}	11	*max* R_{UZ}	12	*min* R_{UZ}
13	$max\sqrt{R_X^2 + R_Y^2 + R_Z^2}$			14	$max\sqrt{R_X^2 + R_Y^2}$		
15	$max\sqrt{R_{UX}^2 + R_{UY}^2 + R_{UZ}^2}$			16	$max\sqrt{R_{UX}^2 + R_{UY}^2}$		

13.12. Design Combinations of Displacements (DCD)

The columns of the table with the design combinations of displacements are similar to those of the DCF tables.

DCD with an automatic selection of coefficients
Units of measurement: mm, degree
Parameters of selection:
List of nodes/elements: 1125-1135
List of types of combinations: All
Designations:
N - characteristic;
NT - characteristic temporary;

DCD with an automatic selection of coefficients						
Node	Crit.	Value			Type	Formula
		X	Y	Z		
1126	4	-0,048	-0,003	-0,113	N	0,909*L1 +0,833*L18 +0,833*L22 +0,75*L28
1126	5	-0,037	-0,002	-0,097	N	0,909*L1 +0,833*L18
1126	8	-0,046	-0,003	-0,112	N	0,909*L1 +0,833*L18 +0,833*L22
1126	1	0,078	0,003	-16,646	NT	0,792*L2 +0,833*L17 +0,75*L19 +0,833*L23 +0,583*L27
1126	2	0,015	0,001	0,027	NT	0,833*L17
1126	3	0,024	0,002	0,038	NT	0,833*L17 +0,75*L19 +0,583*L27
1126	4	-0,042	-0,003	-0,07	NT	0,833*L18 +0,833*L22 +0,75*L28
1126	5	-0,031	-0,002	-0,054	NT	0,833*L18

Figure 13.12-1. *Fragment of the "Design combinations of displacements" (DCD) table*

The following numbering of the criteria is used for the design combinations of displacements:

No.	Criterion	No.	Criterion	No.	Criterion	No.	Criterion
1	max X	2	min X	3	max Y	4	min Y
5	max Z	6	min Z	7	$max\sqrt{X^2+Y^2+Z^2}$		
8	$max\sqrt{X^2+Y^2}$						

13.13. Design Combinations of Deflections

The columns of the table with the design combinations of deflections are similar to those of the DCF tables.

Design combinations of deflections
Units of measurement: mm
Parameters of selection:
List of nodes/elements: 1932 1933
List of types of combinations: All
Designations:
N - characteristic;
NT - characteristic temporary;

Design combinations of deflections								
Elem.	Sec.	CL	Crit.	Value			Type	Formula
				X	Y	Z		
1932	1	1	2	-0,019	0,013	-0,049	N	L1 +0,95*L2 +L3 +L5 +L6 +L24
1932	1	1	4	-0,02	0,007	-0,042	N	L1 +L2 +L3 +0,95*L24
1932	1	1	6	-0,02	0,013	-0,052	N	L1 +L2 +L3 +L5 +L6 +0,95*L23 +0,95*L24
1932	1	1	7	-0,021	0,008	-0,044	N	L1 +L2 +L3 +0,95*L23 +0,95*L24
1932	1	1	2	-0,019	0,013	-0,049	NT	0,95*L2 +L3 +L5 +L6 +L24
1932	1	1	4	-0,02	0,007	-0,042	NT	L2 +L3 +0,95*L24

Figure 13.13-1. *Fragment of the "Design combinations of deflections" table*

The same criteria as for DCD apply for the design combinations of deflections (see Sec. 13.12).

13.14. Design Combinations of Punching

The columns of the table with the design combinations of punching are similar to those of the DCF tables.

Design combinations of punching
Units of measurement:
- Forces: T
- Units of length for force factors: m

Parameters of selection:
List of nodes/elements: All
Designations:
Se - seismic;
Cr - crane;
Sp - special;
Tr - transport.

Design combinations of punching											
Node	CL	Crit.	Value						Type	CS	Formula
			R_X	R_Y	R_Z	R_{UX}	R_{UY}	R_{UZ}			
32	1	1	8,408	-3,159	136,979	9,56	28,912	0,001	C		0,833*L7 +0,909*L10 +C1
32	1	2	7,612	-2,841	123,672	8,558	26,169	0,001	C		0,909*L1 +0,833*L6 +0,909*L10

Figure 13.14-1. *Fragment of the "Design combinations of punching" table*

The following numbering of the criteria is used for the design combinations of punching:

№	Criterion	№	Criterion	№	Criterion	№	Criterion
1	*max* R_X	2	*min* R_X	3	*max* R_Y	4	*min* R_Y
5	*max* R_Z	6	*min* R_Z	7	*max* R_{UX}	8	*min* R_{UX}
9	*max* R_{UY}	10	*min* R_{UY}	11	*max* R_{UZ}	12	*min* R_{UZ}
13	$max\sqrt{R_X^2+R_Y^2+R_Z^2}$	14	$max\sqrt{R_X^2+R_Y^2}$	15	$max\sqrt{R_{UX}^2+R_{UY}^2+R_{UZ}^2}$	16	$max\sqrt{R_{UX}^2+R_{UY}^2}$

13.15. Loads from the Structural Fragment

Loads from the structural fragment
Units of measurement: T, m
Parameters of selection:
List of nodes/elements: 32-46
List of loadings/combinations: 1, 2
List of factors: All

Loads from the structural fragment							
Node	Loading	Value					
		R_X	R_Y	R_Z	R_{UX}	R_{UY}	R_{UZ}
32	1	1,09	-3,246	91,56	10,562	4,853	7,284e-005
32	2	-0,06	-0,096	8,111	0,314	-0,068	8,911e-006
33	1	0,618	-5,216	151,262	16,193	1,611	7,831e-005
33	2	-0,097	-0,306	13,818	0,875	-0,32	8,784e-006
34	1	0,348	-5,395	158,927	16,863	1,418	9,516e-005

Figure 13.15-1. *Fragment of the "Loads from the structural fragment" table*

The results of the calculation of loads from the structural fragment are output (Fig. 13.15-1) in the similar way as the displacements of nodes. The following designations are used in the table:

- R_X, R_Y, R_Z — linear loads in the node in the respective directions;
- R_{UX}, R_{UY}, R_{UZ} — moments in the node.

In those cases when the loads were calculated for the combinations of loadings, numbers of combinations are output instead of the numbers of loadings in the tables. The sign convention corresponds to the sign convention assumed for specifying the nodal loads (see Sec. 8.1).

13.16. Punching Loads

The results of the calculation of punching loads are output (Fig. 13.16-1) in the similar way as the loads from the structural fragment. The following designations are used in the table:

- R_X, R_Y, R_Z — linear loads in the node in the respective directions;
- R_{UX}, R_{UY}, R_{UZ} — moments in the node.

Punching loads
Units of measurement:
- Forces: T
- Units of length for force factors: m

Parameters of selection:
List of nodes/elements: All
List of loadings/combinations: All
List of factors: All

Punching loads								
Node	Loading	Mode number	Value					
			R_X	R_Y	R_Z	R_{UX}	R_{UY}	R_{UZ}
32	1		4,195	-1,576	68,203	4,752	14,419	3,219e-004
32	2		0,253	-0,075	5,168	0,229	0,909	2,406e-005
32	3		0,14	-0,056	1,546	0,19	0,462	4,613e-006

Figure 13.16-1. *Fragment of the "Punching loads" table*

In those cases when the loads were calculated for the combinations of loadings, numbers of combinations are output instead of the numbers of loadings in the tables. The sign convention corresponds to the sign convention assumed for specifying the nodal loads.

13.17. Stability Analysis

Depending on the settings of the postprocessor of the stability analysis , the following tables with the results of the calculation can be generated for each static loading or combination of loadings:

- stability factors of safety;
- buckling modes;
- unsupported lengths of bar elements.

Stability Factors of Safety

The **Stability factors of safety** table (example — Table 13.17-1) provides the value of the stability factor of safety for each loading (or combination of loadings), for which the stability analysis of the system was performed, and in those cases when this factor could not be calculated, it provides the reason for such a situation.

Table 13.17-1

Stability factors of safety		
Number	**Name of the loading/combination**	**Value**
2	Snow	0.0647
3	Live short-term -1	Factor of safety > 2.0000
9	Live long-term -3	Factor of safety > 2.0000
11	Wind against the OY axis (pressure)	0.0978
12	Wind along the OY axis (suction)	0.0877
15	Wind against the OX axis (pressure)	0.1254
16	Wind along the OX axis (suction)	0.0997

Buckling Modes

The ratios between the values of amplitudes in the nodes of the design model for each loading or combination are given in the tables with the buckling modes. The maximal value of amplitude is specified as 1, the values of other amplitudes are determined in fractions of 1.

The tables with buckling modes and their minimax values are similar to the respective tables created when outputting the displacements.

Buckling modes can be viewed in the graphical postprocessor where they are output in the **Displacements** section under the identifier S_i or SC_i (for combinations), where i is the number of the loading.

Unsupported Lengths of Bar Elements

The results of the calculation of the unsupported lengths of bar elements for each static loading (or combination of loadings) are output in the table (see Table 13.17-2). It provides the number of the element, its type, numbers of its nodes and one (for plane models) or two (for spatial models) values of the unsupported length L_{Z1} and L_{Y1}, which indicate the unsupported length of the out-of-plane buckling of the bar in the plane X_1OY_1 and X_1OZ_1. For tension bars unsupported length is meaningless; in this case string *** is used instead of the length value.

Table 13.17-2

Unsupported lengths of elements						
Element	Element type	Node No.1	Node No.2	Loading	L_{Y1}	L_{Z1}
1	5	12	14	1	4,025	4,025
2	5	14	15	1	10,051	8,92
3	5	15	16	1	11,01	9,772
4	5	16	17	1	20,215	17,941
5	5	17	18	1	24,739	21,956

13.18. Principal and Equivalent Stresses

Tables with the results of the calculation of the principal and equivalent stresses (Fig. 13.18-1 and 13.18-2) are generated according to the same principle as the tables of forces and stresses (Sec. 13.5). Moreover, the number of the characteristic point the calculated values refer to (see Chapter 25 **Principal and Equivalent Stresses** for the position of the characteristic points for each type of cross-section) are specified for each bar element after the number of the section. The number of the layer where the calculations were performed is indicated for the shell and plate elements in the "Point" column (1 — lower; 2 — middle; 3 — upper).

Principal and equivalent stresses for forces
Units of measurement: T, m
Parameters of selection:
List of nodes/elements: 1
List of loadings/combinations: All
List of factors: All

Principal and equivalent stresses for forces										
Element	Section	Point	Loading	Value						
				N_X	τ_{XY}	τ_{XZ}	σ_1	σ_3	σ_{E04}	σ_{S04}
1	1	1	1	-340,736	0	0	0	-340,736	340,736	340,736
1	1	2	1	1917,598	0	0	1917,598	0	1917,598	1917,598
1	1	3	1	-68,44	0	0	0	-68,44	68,44	68,44
1	1	4	1	-2326,775	0	0	0	-2326,775	2326,775	2326,775
1	1	5	1	-1333,756	0	36,62	1,005	-1334,76	1335,263	1335,263
1	1	6	1	924,579	0	163,918	952,779	-28,201	967,188	967,188
1	1	7	1	788,431	-180,889	0	827,951	-39,52	848,402	848,402
1	1	8	1	-1197,607	-53,592	0	2,393	-1200,001	1201,199	1201,199

Figure 13.18-1. *Fragment of the "Principal and equivalent stresses for forces" table*

Selection of principal and equivalent stresses for forces
Units of measurement: kN, m
Parameters of selection:
List of nodes/elements: All
List of loadings/combinations: All
List of factors: All

Selection of principal and equivalent stresses for forces										
Name	Maximal values					Minimal values				
	Value	Element	Section	Point	Loading	Value	Element	Section	Point	Loading
N_X	74247,315	19	3	1	1	-77032,788	19	3	1	3
τ_{XY}	9173,815	106	1	2	1	-14575,805	106	1	2	2
τ_{XZ}	10643,325	106	1	1	5	-13217,817	106	3	1	6
σ_1	74247,315	19	3	1	1	0	1	1	1	1
σ_3	0	1	1	1	2	-77032,788	19	3	1	3
σ_{E04}	77032,788	19	3	1	3	0,002	141	2	5	3
σ_{S04}	77032,788	19	3	1	3	0,002	141	2	5	3

Figure 13.18-2. *The "Selection of principal and equivalent stresses for forces" table*

The following values are output as "force" factors in the table (depending on the type of the element, initial data and the stress-strain state):[1]

- principal stresses (T/m^2) — σ_1, σ_2, σ_3;
- Euler angles (rad) — TETA, PSI and FI;
- Lode-Nadai coefficient — MU;
- normal stress in the characteristic points of the bar cross-section (T/m^2) — N_X;
- shear stresses in the characteristic points of the bar cross-section (T/m^2) — τ_{XY}, τ_{XZ};
- equivalent stresses reduced to the equivalent tension according to one of the four failure theories (T/m^2) — σ_{E1}, σ_{E2}, σ_{E3}, σ_{E4};
- equivalent stresses reduced to the equivalent compression according to one of the four failure theories (T/m^2) — σ_{S1}, σ_{S2}, σ_{S3}, σ_{S4}.

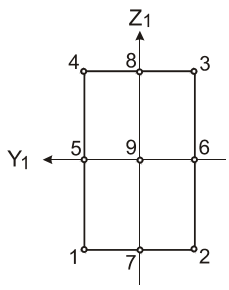


Figure 13.18-3.

Figure 13.18-1 shows the results of the calculation of the principal and equivalent stresses for a bar of rectangular cross-section. Figure 13.18-3 provides an example of the arrangement of characteristic points in such a section. The numbers of these points are given in the "Point" column.

The number of the point of the shell finite element corresponds to the layer where the calculations were performed — lower (1), middle (2), upper (3).

13.19. Results of the Check of the Elements of Reinforced Concrete and Steel Structures and of the Selection of Reinforcement and Rolled Steel Sections

The structure of the tables with the results of the selection of reinforcement in bar and plate elements is described in Chapter 19. This chapter also provides the information on the control of the generation of the report with the results of the check of the specified reinforcement. Similar information but with the results of the check of elements of steel structures is given in Chapter 20.

[1] Units of measurement are given in the basic system of units

14. Erection

The **Erection** mode enables to model the behavior of a structure (determine its stress-strain state) in the process of its erection (see Sec. 2.7). The process of erection of a structure, and, respectively, the calculation is divided into several stages (erection stages). The calculation of each subsequent stage is performed taking into account the stress-strain state of the structure determined on the basis of the results of the calculation of the previous stages. At the transition from one stage to the other you can include and exclude structural elements from the model, take into account different types of static and dynamic loads, change the elastic modulus of the material, and the boundary conditions. The calculation enables to obtain design combinations of forces, combinations of loadings, perform the selection of reinforcement in the elements of reinforced concrete structures, and check and select the rolled steel sections in the elements of steel structures.

A design model can be prepared in advance as a standard project (file with the **.spr** extension) or generated directly in the **Erection** mode (project file with the **.mpr** extension). If necessary the geometry and the characteristics of the elements and nodes of the design model can be adjusted in the **Erection** mode similarly to the standard mode of the data preparation.

14.1. Creating a Project

When creating a new project it is necessary to select the **Erection** radio button in the **New Project** dialog box (Fig. 14.1-1).

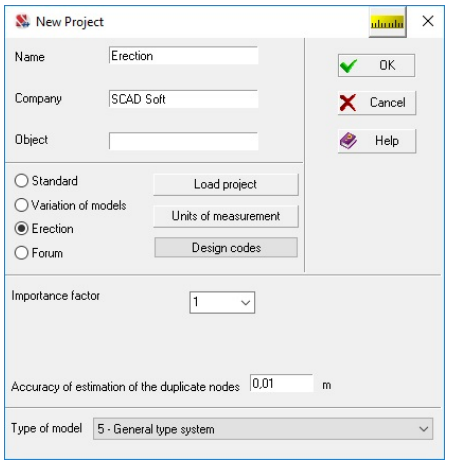


Figure 14.1-1. *The* **New Project** *dialog box*
(the **Erection** radio button is selected)

If the design model was created in advance as a standard project, it can be selected in the **Open** dialog box, which appears after clicking the **Load project** button. The selected standard project has to be saved as a project of the **Erection** mode. It is performed in the standard **Save As …** dialog box, which will be loaded after closing the dialog box by clicking the **OK** button.

You should keep in mind that the file of the **Erection** mode (with the **.mpr** extension) must have a name different from the file name of the standard project (with the **.spr** extension). Otherwise, a situation arises when the files with the results of the calculation of a standard problem and of a problem modeling the erection process will have the same name, which means that one problem in the calculation process will replace the results of the other problem.

An export from the **.mpr** format to the**.spr** format can be performed at any erection stage. In this case all elements, boundary conditions, and loads described in the specified erection stage will be taken into account in the standard model.

When creating a new project for the **Erection** mode, the first erection stage will be created automatically with a zero number of the included elements. The name of the first stage will have to be additionally specified.

14.2. Erection Toolbar

The **Erection** toolbar (Fig. 14.2-1) will be available only when loading a project with the **.mpr** extension. Some buttons, for example, for the specification of constraints, hinges, subsoil parameters disappear from the **Assign** toolbar.

The toolbar contains the following set of buttons:

— initialization of the new erection stage;

— adjust settings of the erection stage;

— delete the erection stage;

— add elements in the current erection stage;

— delete elements in the current erection stage;

— change the elastic modulus;

— install hinges;

— delete hinges;

— specify/delete/change constraints;

— specify/change subsoil parameters;

— invokes **Cross** for the calculation of subsoil parameters;

— merged displacements;

— generate a list of loadings;

— save information on the current stage as a separate problem;

— control the erection stages;

— confirm and reject the current operation.

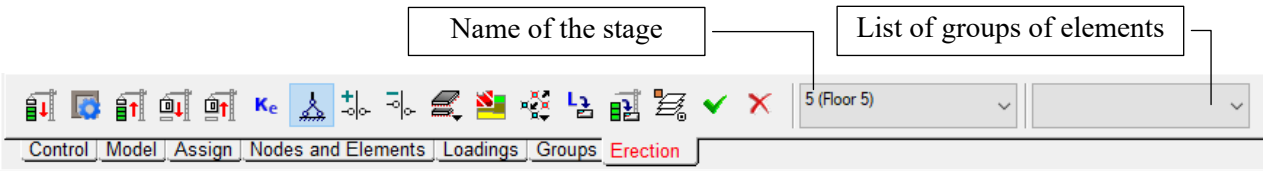


Figure 14.2-1. *Toolbar of the* **Erection** *mode*

Buttons of the toolbar become available when the respective data for the erection stage are specified. Pay attention to the icons in the buttons. The down arrow indicates the specification of a new erection stage or adding elements, and the up button indicates their removal.

The following procedure is recommended for creating an erection stage:

- click the button , the **Erection Stage** dialog box will appear. Specify the name of the stage, and if the calculations have to be performed for the deformed model, check the respective checkbox;
- click the button and specify the groups of added elements, or select the elements in the model which are included in the current stage, and click the button ;
- click the button and specify the groups of excluded elements, or select the elements in the model which are deleted in the current stage, and click the button ;
- if the elastic modulus of some elements changes in the current stage, it is necessary to click the button , the **Elastic Modulus** dialog box will appear, where you have to specify the value of the coefficient to the elastic modulus specified at the description of the stiffness properties of the elements, select the elements or groups of elements in the model the elastic modulus of which changes and click the button ;

– if the constraints in nodes change in the current stage, click the button and perform the operations of installing or deleting the constraints in the selected nodes;
– if the hinges change in the current stage, use the buttons , to perform the operations of installing or deleting the hinges in the selected nodes;
– click the button and specify the main (accumulated) loading in the **Generate Loadings** dialog box, including, if necessary, the groups of loads with the respective coefficients, as well as the independent loadings acting in the current erection stage.

If necessary the current state of the design model can be saved as a standard problem — (here "a standard problem" means a file with the **.spr** extension, not related to the erection mode and having a name different from the file name with the**.mpr** extension).

Initialization of the New Erection Stage

Clicking this button invokes the **Erection Stage** dialog box (Fig. 14.2-2), where you have to:

- specify the name of the stage;
- indicate whether the deformations obtained in the calculation of the previous stage should be taken into account.

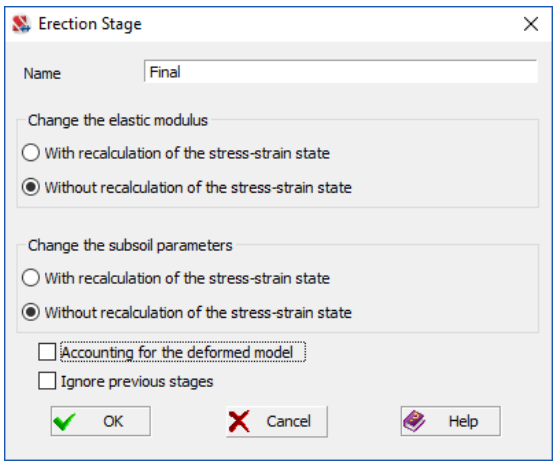


Figure 14.2-2. *The* **Erection Stage** *dialog box*

Regardless of whether the **Accounting for the deformed model** checkbox is checked the nodes of the elements added in the new erection stage and adjacent to the nodes of the elements installed in the previous stage receive coordinates in the process of calculation taking into account the deformed model (allowing for displacements of these nodes obtained at the calculation of the previous erection stage).

All the other nodes are placed in the "design position" corresponding to the initial model. If the checkbox is not checked, the analysis of the stress-strain state is performed taking into account these displacements, but without taking into account the changes of the shape of elements. Otherwise, the calculation accounts for the changes of the shape as well.

When creating each erection stage it is also necessary to select options defining the order of consideration of the variation of the elastic moduli and subsoil parameters. If the stiffness properties of the elements of the system (elastic modulus or the subsoil parameter of the elastic subgrade) change at the transition from one erection stage to the other, it is not difficult to take it into account when specifying the model of the modification of the structure. There can be two variants of this operation:

- the element has acquired a new stiffness property (for example, the stiffness of the compressed elastic subgrade has increased), but the stress-strain state of the system has not changed (mode without recalculation of the stress-strain state);

- the element has changed its stiffness property remaining loaded (for example, the elastic modulus of the element strongly heated by the fire has fallen), i. e. the internal force the element had before the modification is transferred to the new element, and the redistribution of forces occurs in the system without any load changes (mode with recalculation of the stress-strain state).

Obviously, the first variant can hardly be applied when the stiffness property decreases and the redistribution of forces in the system takes place.

While the recalculation of the SSS is usually not required when the stiffness increases, it is required when the stiffness decreases. Having "supposedly" divided the elements into two parts: the shell and the elastic subgrade, selected the reactions of the elastic subgrade from the general reactions, and transferred them to the new element, we will take into account the redistribution of forces in the system without any load changes (mode *with recalculation of the stress-strain state*).

The **Ignore previous stages** checkbox enables to perform the calculation without taking into account the stress-strain state of the structure calculated for the previous stages. Thus, it is possible to simulate several erection processes and obtain design combinations that will include the most unfavorable ones for all stages and all erection processes (similarly to **Variations of Models**).

After closing the dialog box a new line (name of the stage) appears in the list of erection stages, and all the subsequent operations performed in the **Erection** toolbar will apply to this stage.

Adjusting Settings of the Erection Stage

If it is necessary to adjust the specified settings of the stage (for example, change the name of the first stage, which is by default specified as "1"), you have to select the respective stage in the drop-down list of the erection stages in the toolbar and click the **Adjust settings of the Erection stage** button — . Make the adjustments in the **Erection Stage** dialog box (Fig. 14.2-2) that will appear, and close it by clicking the **OK** button.

Deleting the Erection Stage

In order to delete an erection stage, it is necessary to select it in the drop-down list of the toolbar and click the **Delete the erection stage** button. If not the last stage is selected to be deleted, all the stages following it are deleted as well (if agreed with the user).

Adding Elements in the Current Erection Stage

In order to specify the elements added to the structure in the current erection stage, click this button, select the respective elements in the model, and click the **OK** button. The elements can be selected directly in the model by one of the cursors, or by specifying the names of the groups of elements in the drop-down list in the **Erection** toolbar. The added elements are displayed in the color indicating that the element is included in one of the erection stages. Thus, the model will be displayed in two colors: one color will be used for the elements already included in the current erection stage, and other one — for those which are not yet included.

The filters button enables to display the elements included in the model in the current erection stage (see Sec. 10.2).

Deleting Elements in the Current Erection Stage

In order to exclude elements from the current stage, click this button, select the respective elements in the model, and click the **OK** button in the toolbar. The elements are displayed in a color different from the color of the added elements. If necessary these elements can be added in the model in other stages.

Changing the Elastic Moduli for the Current Erection Stage

In order to change the elastic modulus of the elements in the calculation of the current erection stage invoke this operation, specify the value of the coefficient in the **Elastic Modulus** dialog box (Fig. 14.2-3), and close it by clicking the **OK** button. Select the elements in the model the modulus of which changes in the current stage, and click the **OK** button. If the groups of elements have different values of the elastic modulus, it is necessary to repeat the above actions for each group.

The specified coefficients will be used in the calculation of the current erection stage for changing the elastic moduli of the elements specified when generating the initial design model. If the **Apply to all subsequent stages** checkbox is checked, the specified coefficients will be used in all the subsequent stages as well. In order to view the values of the coefficients of variation of the elastic modulus in the erection stages check the respective checkbox in the **Erection Stages** tab (Fig. 14.2-4) of the **Set the Information Display Filters** dialog box (see Sec. 10.2).

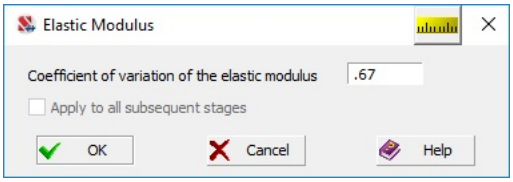


Figure 14.2-3. *The* **Elastic Modulus** *dialog box*

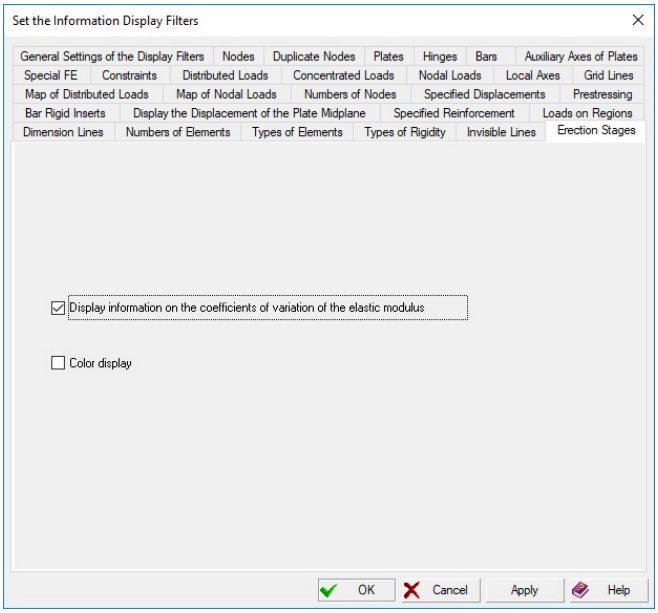


Figure 14.2-4. *The* **Erection Stages** *tab* (**Set the Information Display Filters**)

Creating a List of Loadings

There are two types of loadings in the **Erection** mode — the accumulated (main) loading and independent loadings.

There can be only one accumulated loading, it acts on each erection stage and it is considered in DCF as a permanent load. The loads of the accumulated loading include the loads from the self-weight, and they can also include the loads from the materials stored during the erection process, etc.

Independent loadings which include the live load, wind, seismic load and other dynamic actions etc. can act both on the resulting structure and on the different erection stages. They act as independent loadings in DCF and can have any type. You should keep in mind that the main loading will be automatically added to each independent loading, i. e., if the main loading is the self-weight, the respective snow load will be equal to “snow + self-weight”.

The preparation of the data on loadings involves the following:

- all the main loadings acting in one stage are combined into one loading in advance (the presence of such a loading is obligatory), and if necessary, into a number of groups of loads. For example, if the elements of a structure are made from different materials with different safety factors for load

(reinforced concrete, steel, etc.), the self-weight of steel structures can be given as a loading, and the self-weight of other structures can be given as groups of loads with the respective coefficients;

- independent loads acting on each erection stage are combined into loadings. Groups of loads for independent loadings are not provided;
- if there are loads of a certain type acting on the whole (resulting) model, for example, self-weight, they can be specified as one loading. Only the loads from this loading which are applied to the elements and nodes existing on this stage will be automatically considered on each erection stage.

The **Generate Loadings** dialog box (Fig. 14.2-5) appears after the initialization of the erection stage and clicking the button . It includes three lists. The left one contains the full list of all loadings and groups of loads created at the moment of initialization of the current stage. The accumulated loading (there must be such a loading) is transferred from the left list to the right upper list given in the form of a table, as well as the groups of loads with the respective coefficients. Independent loadings acting on the current erection stage are transferred to the lower list.

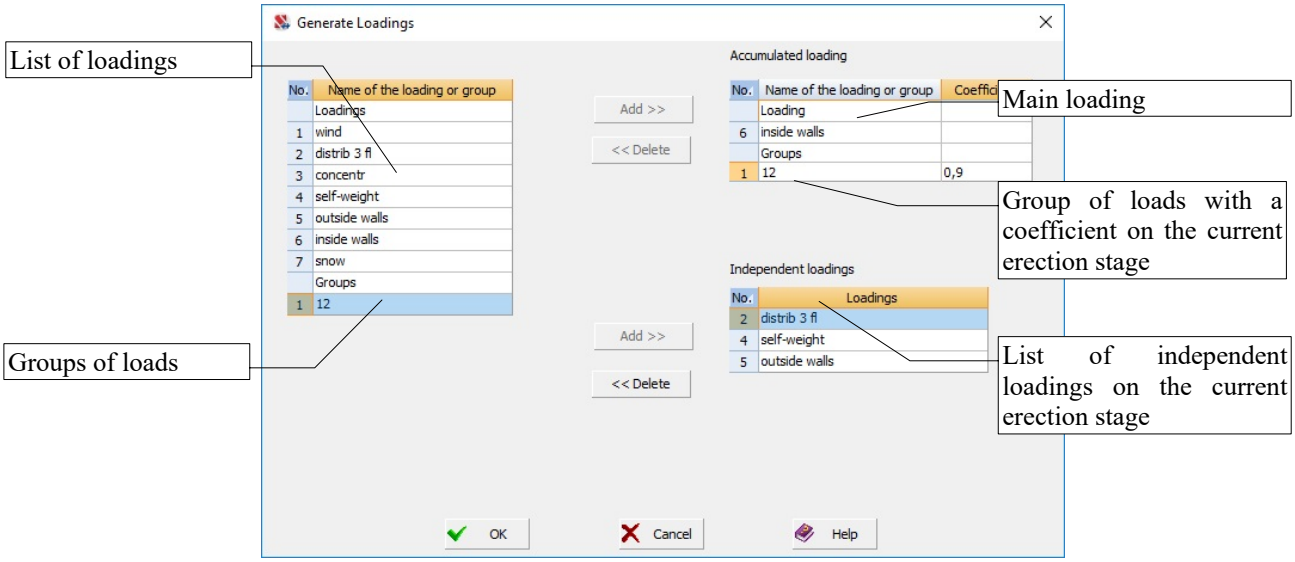


Figure 14.2-5. *The* **Generate Loadings** *dialog box*

In order to transfer a loading (group of loads) into the right tables select it in the left table and click the respective **Add** button. If the operation was performed incorrectly, the loading or group selected in the right list can be deleted by clicking the respective button.

Groups of loads are specified with coefficients the values of which are given in the respective column of the table. When the main loading is generated, loadings and groups of loads are summed according to the rules described in Section 8.4. Both accumulated and independent loadings always have a coefficient equal to 1.

The last erection stage is considered as operational, i.e. loadings which usually act on the finished structure are included as independent ones on this stage. They include live loads, snow, wind, dynamic loads typical for buildings and structures of this type, etc.

Specifying and Modifying Constraints

After selecting the respective erection stage the constraints can be specified and modified (or deleted) in the same way as in the mode of the generation of the design model. Constraints can be installed even in the nodes which are not adjacent to the existing elements because this operation is performed for the current erection stage and all the subsequent ones.

Specifying Hinges

Hinges can be installed in the same way as in the mode of the generation of the design model for the current erection stage and all the subsequent ones.

Deleting Hinges

Hinges can be deleted in the same way as in the mode of the generation of the design model for the current erection stage and all the subsequent ones.

Specifying and Changing the Subsoil Parameters

Subsoil parameters can be specified and changed in the same way as in the mode of the generation of the design model for the current erection stage and all the subsequent ones.

Specifying Merged Displacements

Merged displacements can be specified and changed in the same way as in the mode of the generation of the design model for the current erection stage and all the subsequent ones.

Save Information on the Current Stage as a Separate Problem

In this case the current state of the project is saved as a separate file with the **.spr** extension. This operation is performed in the standard **Save As…** dialog box.

Control the Erection Stages

The operation is used to control the parameters of erection stages and enables to perform the following actions: rename the stages and set the calculation parameters for the erection stages (calculation for a deformed model, recalculation of the stress-strain state at the variation of the elastic moduli and/or subsoil parameters).

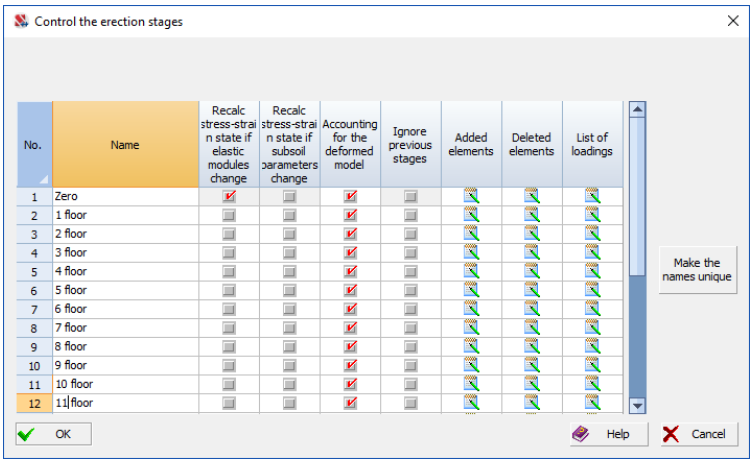


Figure 14.2-6. *The* **Control the Erection Stages** *dialog box*

Operations are performed in the **Control the Erection Stages** dialog box (Fig. 14.2-6), which contains a table with a list of stages. The table has the following columns:

- **No.** — number of the stage;

- **Name** — name of the stage;
- Indicator of the calculation for a **deformed model**;
- Indicator of the recalculation of the stress-strain state at the variation of the **elastic moduli**;
- Indicator of the recalculation of the stress-strain state at the variation of the **subsoil parameters**.

In order to rename a stage place the cursor over the name of the respective stage and double left click. The table line will become available for editing.

The buttons in the **Added elements** and **Deleted elements** columns enable to edit (in the text format) the lists of added/deleted elements for the respective erection stage. The buttons in the column **List of loadings** enable to create/modify list of loadings for the respective erection stage (like operation).

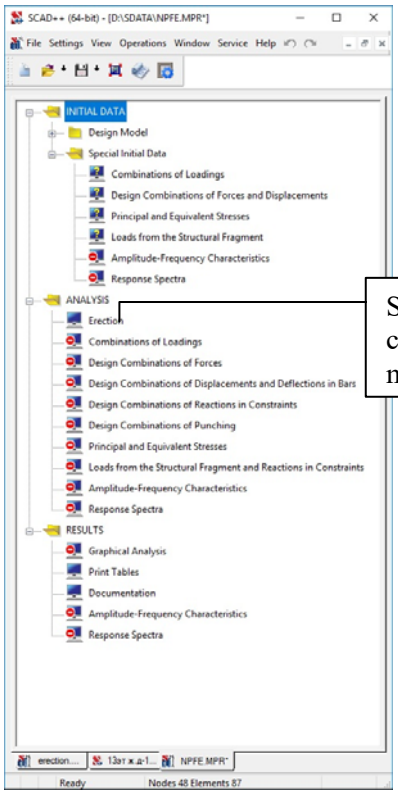
If the design model has stages with the same names, clicking the **Make the names unique** button enables to rename them automatically by adding the suffix No. XXX to the name.

Creating and Modifying the Main Model

When working in the **Erection** mode the user can either create a design model "from scratch", or modify the existing one. Working with the **Control**, **Model**, **Assign**, **Nodes and Elements**, **Loadings** and **Groups** toolbars is almost the same as working with a standard model. The following rules for working with the model in the **Erection** mode should be taken into account:

- new elements added to the design model will not be activated on the current erection stage and all the subsequent ones. This operation should be performed according to the rules described above (see Sec. **Adding Elements in the Current Erection Stage**);
- buttons for specifying the constraints, hinges, merged displacements, and subsoil parameters can be used only in the **Erection** toolbar (they are not available in the **Assign** toolbar), they enable to perform the operations of specification only for the current erection stage;
- the button in the filters toolbar — **Show elements included/not included in the given erection stage**, enables to indicate elements included in the model on the current erection stage;
- in order to view the values of the coefficients of variation of the elastic modulus on the erection stages check the respective checkbox in the **Erection Stages** tab (Fig. 14.2-4) of the **Set the Information Display Filters** dialog box (see Sec. 10.2).

14.3. Analysis



Section for invoking the calculation in the **Erection** mode

The **Erection** section appears in the **ANALYSIS** group of functions of the project tree in the **Erection** mode (Fig. 14.3-1). A standard **Calculation Parameters** dialog box becomes available after activating this section, where you can specify the calculation parameters according to the rules described above (see Chapter 11).

Figure 14.3-1. *Project tree in the* **Erection** *mode*

14.4. Analysis of Results

Analysis of the results of the calculation in the **Erection** mode is performed in the graphical postprocessor according to the same rules as for a standard model (see Chapter 12). In addition to the parameters described above a number of the erection stage is indicated for each loading in the list of loadings (Fig. 14.4-1, *a*).

After selecting a line with the name of the respective erection stage from the list of loadings in the toolbars of the postprocessor only the elements which are present in the model in the current stage will remain in the work area (Fig. 14.4-1, *b*).

You should keep in mind that the values of factors for independent loadings will be displayed taking into account the fact that their value is obtained according to the formula “independent + main”.

The following designations for the identification of the output information are used in addition to those described in the **Selecting Loading** section (Sec. 12.1):

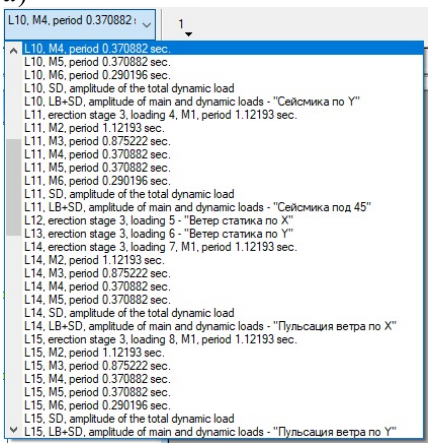
LB + LS – amplitude from the main and static load (static wind) of the dynamic loading;

LB + LS + SD – amplitude from the main, static, and dynamic components of the dynamic loading;

LB + S1, LN + S2, ... – amplitude from the main, and dynamic envelope from the real and imaginary components of the dynamic loading;

LB + T1, LN + T2, ... – amplitude from the main and dynamic component at the given moment in time.

a)



b)

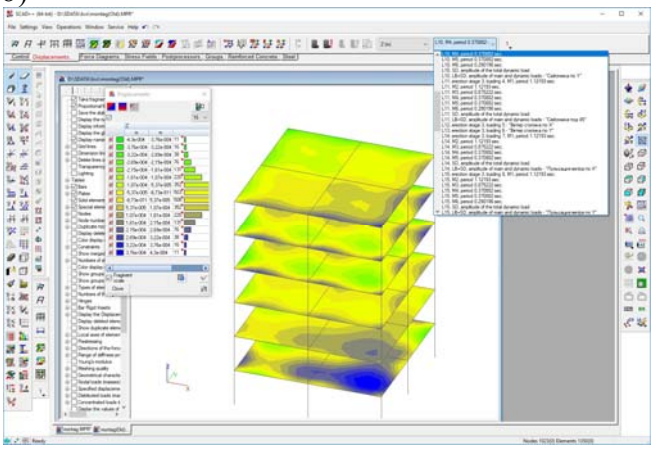


Figure 14.4-1. *The* **Displacements** *tab*, *the* **Analysis of results** *mode*
a) list of loadings, *b*) isofields of displacements on one of the erection stages

Erection Stages History

The control of the stress-strain history (for the accumulated loading) of elements and nodes is performed with the help of the **Erection Stages History** operation, which is invoked from the **Information on the Element** and **Information on the Node** dialog boxes respectively.

14.5. Design Combinations of Forces

Design combinations of forces for the **Erection** mode are generated with the help of the DCF postprocessor. It is necessary to take into account the following when specifying the relationship between loadings in this postprocessor (see Chapter 18):

- loadings acting on different erection stages are automatically considered as mutually exclusive, but it is not displayed in the table of mutual exclusions of the respective dialog box;
- the logic of the interaction of all loadings acting on one erection stage is specified by the user. The accumulated (main) loading is always considered as permanent;
- when generating DCF for all other loadings (except for the main one) only the part these loadings were replaced by in the result of adding with the main loading is taken into account.

14.6. Calculation of Elements of Steel and Reinforced Concrete Structures

The calculation of elements of steel and reinforced concrete structures (check and selection) should obviously be performed for the last erection stage. All unfavourable combinations of forces which appeared on the previous stages will be taken into account. Therefore, in order to enter the **Steel** or **Reinforced Concrete** postprocessors you have to (for example) select a loading which corresponds to the last stage in the *Displacements* or *Forces* tabs.

The calculation of elements of steel and reinforced concrete structures can be also performed based on the results of the calculation for individual erection stages. In this case it is necessary to perform the following actions:

- specify all the loadings which belong to the erection stages following the considered one as inactive in the **Design Combinations of Forces** dialog box;
- perform the generation of DCF;
- adjust the value of the design strength of concrete with the help of the hardening factor in the initial data of the **Reinforcement** dialog box in such a way so that this value corresponds to the considered erection stage;
- perform the selection or check of the reinforcement in particular elements of the model;
- perform the selection or check of the elements of steel structures.

15. Direct Integration of the Equations of Motion

15.1. Problem Formulation

One of the peculiarities of this problem formulation is the fact that displacements, velocities, and accelerations of the nodes of the finite element model are determined in the absolute reference system connected with the conventionally immobile earth. The seismic action is specified as the forced displacements of supports, therefore not the accelerograms but seismograms (relationship between displacements and time) obtained from accelerograms by the double time integration are used as the input information. This approach enables to solve the problems with asynchronous oscillations of supports in a natural way. It is recommended to use the **Accelerogram Editor** to obtain vibrograms from the specified accelerograms.

A traditional problem formulation in the reference system connected with the rigid mobile base, and a problem formulation in the absolute reference system used in **SCAD** are shown in Fig. 15.1-1, 15.1-2 respectively.

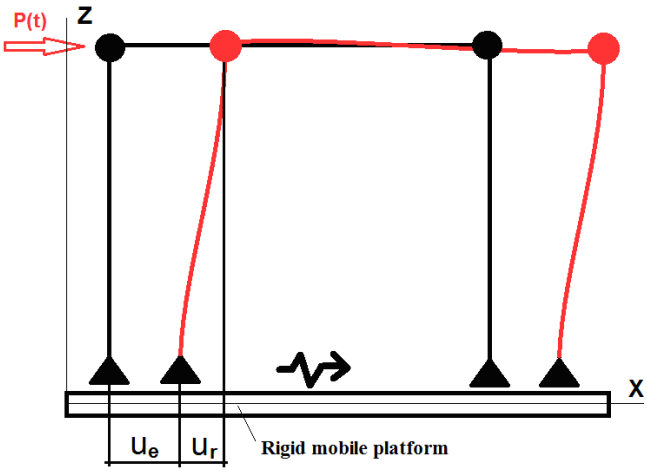


Figure 15.1-1. *Solution of the problem in the relative reference system connected with the rigid mobile platform exhibiting translational motion (classical formulation)*

$$\begin{cases} \mathbf{u} = \mathbf{u}_e + \mathbf{u}_r; \\ \ddot{\mathbf{u}} = \ddot{\mathbf{u}}_e + \ddot{\mathbf{u}}_r; \\ \mathbf{M}\ddot{\mathbf{u}}_r + \mathbf{K}\mathbf{u}_r = -\mathbf{M}\ddot{\mathbf{u}}_e(t) + \mathbf{p}(t) \end{cases}$$

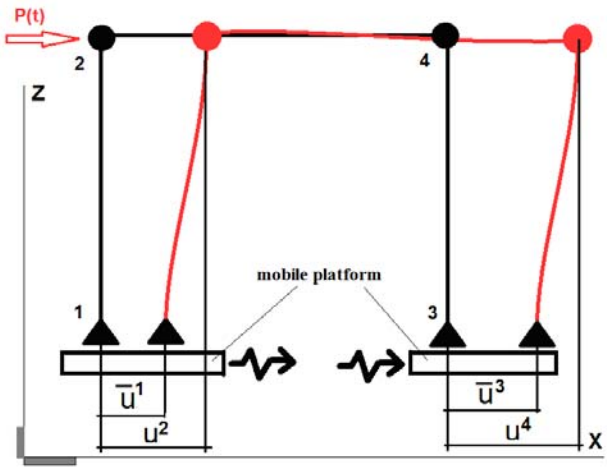


Figure 15.1-2. *Solution of the problem with respect to the conventionally immobile absolute reference system (**SCAD**)*

Equations of motion of the finite element (FEM) model in the absolute immobile reference system are given as follows:

$$\underbrace{\begin{pmatrix} \mathbf{M}_{11} & 0 \\ 0 & \mathbf{M} \end{pmatrix}}_{\hat{\mathbf{M}}} \cdot \begin{pmatrix} \ddot{\bar{\mathbf{u}}} \\ \ddot{\mathbf{u}} \end{pmatrix} + \underbrace{\begin{pmatrix} \mathbf{K}_{11} & \mathbf{K}_{12} \\ \mathbf{K}_{21} & \mathbf{K} \end{pmatrix}}_{\hat{\mathbf{K}}} \cdot \begin{pmatrix} \bar{\mathbf{u}} \\ \mathbf{u} \end{pmatrix} + \underbrace{\begin{pmatrix} \mathbf{C}_{11} & \mathbf{C}_{12} \\ \mathbf{C}_{21} & \mathbf{C} \end{pmatrix}}_{\hat{\mathbf{C}}} \cdot \begin{pmatrix} \dot{\bar{\mathbf{u}}} \\ \dot{\mathbf{u}} \end{pmatrix} = \begin{pmatrix} 0 \\ \mathbf{p}(t) \end{pmatrix}, \tag{15.1.1}$$

where $\hat{\mathbf{K}}, \hat{\mathbf{M}}, \hat{\mathbf{C}}$ are stiffness, mass, and dissipation matrices respectively covering the degrees of freedom which have to be determined (vector of nodal values of the unknowns $\mathbf{u}$), as well as the known displacements in nodes – forced displacements $\bar{\mathbf{u}}(t)$, $\mathbf{p}(t)$ – given forces.

It follows from (15.1.1):

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = -\mathbf{K}_{21}\bar{\mathbf{u}}(t) - \mathbf{C}_{21}\dot{\bar{\mathbf{u}}}(t) + \mathbf{p}(t), \tag{15.1.2}$$

The left-hand side of this equation contains accelerations, velocities and displacements that have to be determined, and the right-hand side contains known values.

Two different approaches are implemented to account for dissipation. The first approach is based on the Rayleigh hypothesis, and the second one allows for the internal resistance of the material.

According to the Rayleigh hypothesis,

$$\hat{\mathbf{C}} = \alpha \cdot \hat{\mathbf{M}} + \beta \cdot \hat{\mathbf{K}}, \tag{15.1.3}$$

where α, β are the coefficients of proportionality, term $\alpha \cdot \hat{\mathbf{M}}$ stands for the damping of lower modes, and term $\beta \cdot \hat{\mathbf{K}}$ – for the damping of upper ones. Such a representation enables to divide a system of equations at the decomposition of the load for the natural oscillation modes of the undamped system [1], [2].

If we substitute (15.1.3) into (15.1.2), we obtain:

$$\mathbf{M}(\ddot{\mathbf{u}} + \alpha \cdot \dot{\mathbf{u}}) + \tilde{\mathbf{C}}\dot{\mathbf{u}} + \mathbf{K}(\mathbf{u} + \beta \cdot \dot{\mathbf{u}}) = -\mathbf{K}_{21}(\bar{\mathbf{u}} + \beta \cdot \dot{\bar{\mathbf{u}}}) - \tilde{\mathbf{C}}_{21}\dot{\bar{\mathbf{u}}} + \mathbf{p}(t). \tag{15.1.4}$$

Here $\mathbf{K}, \mathbf{M}$ are traditional stiffness and mass matrices covering only the degrees of freedom which have to be determined. $\tilde{\mathbf{C}}$ – dissipation matrix allowing for local dampers (see Sec. 1.6).

In the case of non-Rayleigh damping (taking into account the internal resistance of the material), the dissipation matrix is given as:

$$\mathbf{C} = \sum_{e=1}^{N_e} \gamma_e \mathbf{P}_e^T \mathbf{K}_e \mathbf{P}_e, \tag{15.1.5}$$

where the summation is performed for all finite elements of the design model, N_e, γ_e – coefficient of internal inelastic resistance of the material, $\mathbf{K}_e$ – stiffness matrix of the finite element e, $\mathbf{P}_e$ – a permutation matrix that defines the position of the elements of the stiffness matrix $\mathbf{K}_e$ of a finite element e in the global stiffness matrix $\mathbf{K}$. The dissipation matrix $\mathbf{C}$ is a symmetric sparse matrix with the same nonzero structure as the stiffness matrix $\mathbf{K}$. This approach enables to take into account multi-component damping, when the finite elements of the design model have different values of γ_e in the case of different materials. The dissipation matrix $\mathbf{C}$ can not be given as a linear combination of matrices $\mathbf{K}$ and $\mathbf{M}$, and the application of the method of expanding by natural oscillation modes does not divide the equations of motion into separate independent equations for each mode. Therefore, the direct integration methods are applied for the solution of the problem (2).

It should be noted that the simultaneous specification of the Rayleigh damping and non-Rayleigh damping is possible only when the Rayleigh damping and local dampers are used. Simultaneous use of Rayleigh damping and dissipation model on the basis of internal inelastic resistance is impossible – if the **Material damping** is selected in the **Parameters of Dynamic Actions** dialog box, the Rayleigh damping will be ignored. **Therefore, when this dissipation model is used, the damping parameters have to be specified for each material.**

Designate $\bar{\mathbf{u}} = \sum_{s=1}^{M_s} \mu_s \cdot \mathbf{k}_s \cdot f_s\left(t - t_0^s\right)$, $\dot{\bar{\mathbf{u}}} = \sum_{s=1}^{M_s} \mu_s \cdot \mathbf{k}_s \cdot \dot{f}_s\left(t - t_0^s\right)$, where $\mathbf{k}_s$ – vectors of the spatial configuration specifying the mode of the forced displacements, $f_s(t)$ – function defining the load variation with time, $s = 1, 2, \ldots, M_s$ – *reference loadings* or simply *references* – numbers of static loadings at the calculation for forced displacements. Similarly $\mathbf{p}(t) = \sum_{p=1}^{N_p} \mu_p \cdot \mathbf{k}_p \cdot f_p\left(t - t_0^p\right)$. Here $p = 1, 2, \ldots, N_p$ – *reference loadings* or simply *references* – numbers of static loadings at the calculation for force actions, $\mathbf{k}_p$ – vectors of the spatial load configuration. Values μ_s, t_0^s, μ_p, t_0^p refer to the scale factors and the delay time respectively.

We will call each term of the above sums as the *s-th* or *p-th component of the dynamic loading* respectively.

Designate $\mathbf{b}_s = -\mathbf{K}_{21} \cdot \mathbf{k}_s$, $\mathbf{c}_s = -\mathbf{C}_{21} \cdot \mathbf{k}_s$, $s = 1, 2, \ldots, M_s$. If we supplement the above differential equation with the initial conditions, we obtain

$$\begin{cases} \mathbf{M}(\ddot{\mathbf{u}} + \alpha \cdot \dot{\mathbf{u}}) + \mathbf{K}(\mathbf{u} + \beta \cdot \dot{\mathbf{u}}) = \sum_{s=1}^{M_s} \mathbf{b}_s \cdot \mu_s \cdot \left[f_s\left(t - t_0^s\right) + \beta \cdot \dot{f}_s\left(t - t_0^s\right)\right] + \sum_{p=1}^{N_p} \mathbf{k}_p \cdot \mu_p \cdot f_p\left(t - t_0^p\right) \\ \mathbf{x}(0) = \mathbf{x}_0 \\ \dot{\mathbf{x}}(0) = \dot{\mathbf{x}}_0 \end{cases} . \quad (15.1.6)$$

Here the matrix **C** represents both a combination of Rayleigh damping with local dampers (if there are any), and a combination of the material damping with local dampers.

In the case of Rayleigh damping coefficients α and β can be calculated according to the following formulas

$$\alpha = \frac{2\omega_i\omega_j\left(\xi_i\omega_j - \xi_j\omega_i\right)}{\omega_j^2 - \omega_i^2}, \ \beta = \frac{2\left(\xi_j\omega_j - \xi_i\omega_i\right)}{\omega_j^2 - \omega_i^2}, \quad (15.1.7)$$

where ω_i, ω_j, – two natural cyclic frequencies [rad/sec] for the oscillation modes i and j, and ξ_i, ξ_j – modal damping given as a percentage of the critical damping. It is assumed that $\omega_j > \omega_i > 0$ and $\xi_j \geq \xi_i > 0$.

If we take $i = 1, j = 2$, then

$$\alpha = \frac{2\omega_1\omega_2\left(\xi_1\omega_2 - \xi_2\omega_1\right)}{\omega_2^2 - \omega_1^2}, \ \beta = \frac{2\left(\xi_2\omega_2 - \xi_1\omega_1\right)}{\omega_2^2 - \omega_1^2} \ . \quad (15.1.8)$$

In the case when $\omega_2 = \omega_1$ or the first two frequencies are very close ($\frac{\omega_2 - \omega_1}{\omega_1} < 0.0001$), it is assumed that $\alpha = 2\xi_1\omega_1$, and $\beta = 0$.

Initial conditions are defined by the initial displacements of the system (vector $\mathbf{u}_0$) and initial velocities (vector $\dot{\mathbf{u}}_0$). It is assumed in the current implementation that $\mathbf{u}_0 = \dot{\mathbf{u}}_0 = 0$.

Method of the solution. The α – HHT method in the "predictor-corrector" form is used for solving the problem [3], [5], [6]. The shift parameter α introduces "false dissipation", and as a result this method suppresses high-mode oscillations even if there is no physical damping. If we introduce the parameter $\omega\Delta t$, where ω – test frequency, and Δt – selected integration step, then the frequencies well approximated by this step Δt, satisfy the inequality $\omega\Delta t < 1$. The frequencies for which $\omega\Delta t > 1$ are poorly approximated at the selected step Δt, and if the contribution of the modes of the oscillations of the corresponding high frequencies is large, then the numerical integration of problem (6) will be performed with a significant error. It often leads to the fact that the response of the design model to high (poorly approximated) modes distorts the numerical solution. In such cases, the use of a method that introduces false dissipation often saves the situation. When $\alpha \neq 0$ the α – HHT method is mathematically incorrect, since the false damping appears also at the frequencies $\omega\Delta t < 1$, which are well approximated at the selected step Δt, although much less than for poorly approximated high frequencies. However, a low suppression of low frequencies within a relatively small number of cycles is acceptable for many practical applications.

Parameter $\alpha \in [0, -1/3]$ provides a stable numerical solution, and the greatest suppression of high modes occurs when $\alpha = -1/3$. When $\alpha = 0$ there is no false damping, and the α – HHT method turns into the absolutely stable and strictly correct Newmark method.

The whole time interval $[t_{start}, t_{end}]$ is divided into a finite number of steps $N_{step} = T_{dur}/\Delta t + 1, \quad T_{dur} = t_{end} - t_{start}$. The integration step Δt remains constant within a single problem formulation. If it is necessary to integrate the equations of motion with a different step, you have to divide the whole time interval into subintervals $[t_0, t_1], \quad [t_1, t_2], \quad \ldots, [t_{k-1}, t_k]$, each of which is integrated with a step constant within an interval Δt_k (Fig. 15.1-3). The first problem formulation – beginning of the calculation – is performed for the first subinterval $[t_0, t_1]$, and the other problem formulations – continuation of the calculation – for the subintervals $[t_1, t_2], \quad \ldots, [t_{k-1}, t_k]$ respectively.

The results (displacements, velocities, accelerations) are recorded at some moments in time which coincide with the integration points (Fig. 15.1-4).

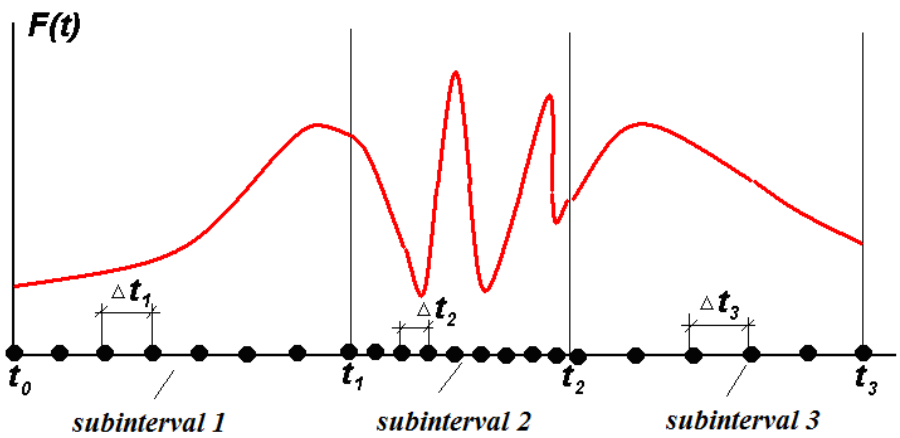


Figure 15.1-3. *Division into subintervals*

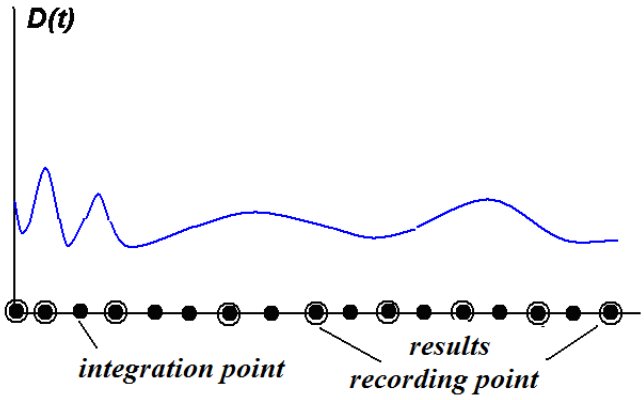


Figure 15.1-4. *Integration and results recording points*

In order to achieve the acceptable accuracy of the solution the integration step is usually much less than the step for recording the results. Results can be recorded either with a variable step, or with a constant one.

The implementation is based on the sparse matrix technology which enables to speed up the calculations significantly and to reduce the time of the analysis.

15.2. Specifying the Initial Information

Specifying the Characteristics of the Dynamic Loading

The initial data for the calculation are specified in the multi-tab **Parameters of Dynamic Actions** dialog box (Fig. 15.2-2), which is invoked by clicking the button in the **Loadings** toolbar of the preprocessor. Check the **Direct integration** checkbox in the **Type of action** group and the **Direct Integration of the Equations of Motion** tab will become available (Fig. 15.2-3).

In order to perform the calculation it is necessary to specify the following data:

- integration step;
- process duration;
- data on the modal damping, which can be specified by indicating the parameters of damping (in fractions of the critical value) for the first two natural modes. Alternatively, the coefficients α and β can be specified explicitly by the user (if the **User specified** radio button is selected). Clicking the button invokes the dialog box (Fig. 15.2-1), which enables to specify two natural frequencies and the respective parameters of modal damping. Clicking the **OK** button invokes the calculation of the coefficients α and β according to the formulas (7) and automatically records these values in the respective text fields; moreover, damping can be taken into account on the basis of the material damping (damping parameter has to be specified in the stiffness properties of the finite elements);
- list of the result output time;
- parameter of α-HHT method.

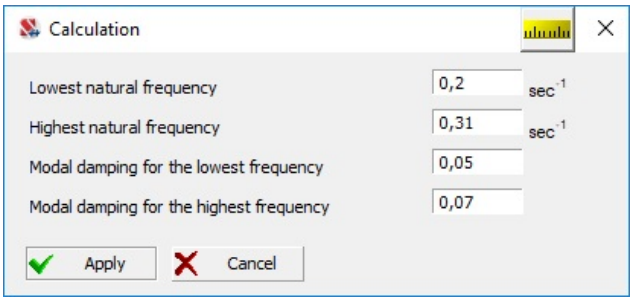


Figure 15.2-1. *Determination of the parameters of modal damping*

The following information is specified in the lines of the table:

- delay time – time interval which enables to move the given load in time with respect to the moment of the beginning of the integration process;
- scaling factor, by which the load is multiplied at any given time;
- number and name of the reference static loading;
- the full path and the name of the text file defining the time function $\varphi_p(t)$.

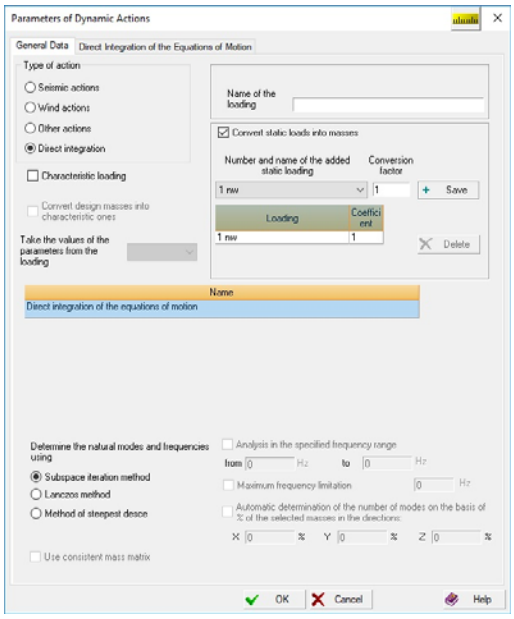


Figure 15.2-2. *The* **General Data** *tab of the* **Parameters of Dynamic Actions** *dialog box*

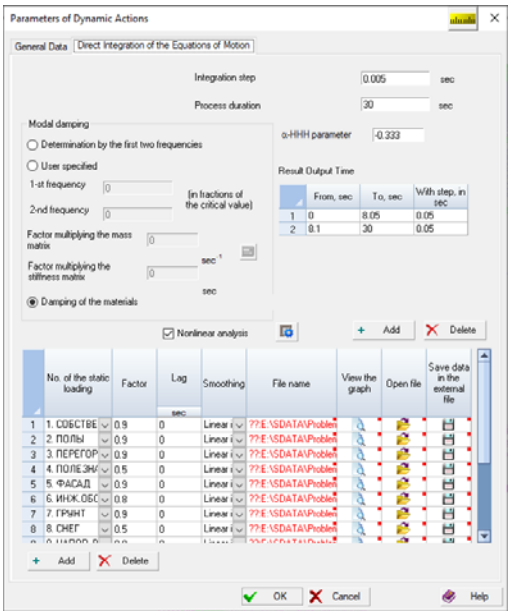


Figure 15.2-3. *The* **Direct Integration of the Equations of Motion** *tab*

If when specifying the result output time the process time is not yet exhausted, the step of the output of the following results is determined as the difference between the last two specified values. For example, suppose that the process duration is 1 second, and the values 0 0.02 0.03 are specified in the "Result Output Time" dialog box. Results of the calculation will be saved at 0, 0.02 sec 0.03 sec 0.04 sec 0.05 sec ... 1 sec. The first three values are explicitly specified by the user, and each subsequent one is determined as the preceding value plus the increment, where the increment is calculated as 0.03 – 0.02 = 0.01 sec. This implies that at least two values should be specified in the "Result Output Time" dialog box.

In order to specify a dynamic loading you have to specify the reference static loads first in the same way as for the standard static loads. You can specify not only the nodal forces, but also the concentrated forces and distributed loads applied to the finite elements between the nodes, and the forced displacements of the supports as well, which enables to perform the calculation of structures for the seismic actions.

It is necessary to perform the following actions to specify the dynamic loads:

- click the **Add** button, and a new line will appear in the table with the description of loads;
- select a static loading in the drop-down list of the **No. of the static loading** column on the basis of which the time-varying load will be generated;
- specify the scaling factor μ_1 in the **Factor** column;
- specify the delay time t_0^1 in the **Lag** column;
- click the button to select a text file which defines the function $\varphi_1(t)$ (the file has extension **.thi** by default);
- if it is necessary to specify the second component, click the **Add** button, a new line will appear in the table, and repeat the above actions.

In order to perform the control of the graph of the time function click the button , and the **Time Function** dialog box (Fig. 15.2-4) will appear. Buttons in the lines of the table enable to select and replace the respective text file with the time function.

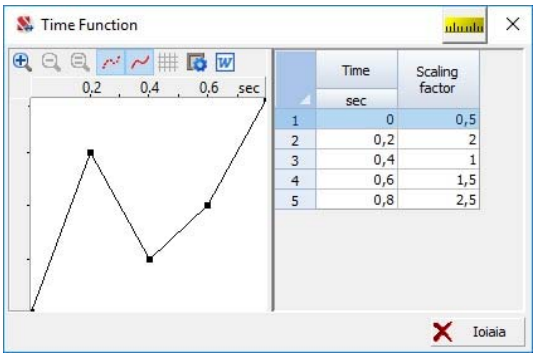


Figure 15.2-4. *Graph of the time function*

The content of each text file with a time function is copied and stored "inside" the file of the model. Therefore, in order to transfer the data to another computer you just have to copy the spr-file. A situation may arise when the path specified earlier no longer corresponds to the current location of the file. In this case the line of the table looks as follows:

Two question marks and the red color of the file name indicate the absence of the file on the disk, and the button enables to save the time function in the file on the external disk (the standard **Save** dialog box for selecting the storage location is used).

In order to delete a certain component of the dynamic load from the list, it is necessary to select the respective line of the list with a cursor and click the **Delete** button. Information on the dynamic loading will be saved in the model after closing the dialog box by clicking the **OK** button.

The "linear interpolation" option is used in the "**Smoothing**" column by default. The thing is that the values of the time function $f_s\left(t-t_0^s\right)$ are specified at different time points. It is necessary to perform interpolation during the numerical integration – to determine the value of the time function between the interpolation nodes – the time points at which the function $f_s\left(t-t_0^s\right)$ is specified. For the vast majority of problems, it is sufficient to apply linear interpolation. However, in the case of forced displacement of supports (seismic analysis), it may turn out that linear interpolation is too "rigid" and excites parasitic high frequencies because the velocities $\dot{f}_s\left(t-t_0^s\right)$ at the interpolation nodes undergo discontinuities, and the accelerations $\ddot{f}_s\left(t-t_0^s\right)$ are zero throughout the entire interpolation interval between the nodes, and are represented by Dirac delta functions in the interpolation nodes [6].

Smoothing has to be applied in this case, and possibly suppression of high modes with the help of the α – HHT method. The "Cubic parabola" or "Hermite polynomial" option is selected for smoothing. In either case, the time function $f_s\left(t-t_0^s\right)$ will be approximated between interpolation nodes by a polynomial of the third degree, the velocity function $\dot{f}_s\left(t-t_0^s\right)$ – by a polynomial of the second degree, and the acceleration $\ddot{f}_s\left(t-t_0^s\right)$ – by a linear interpolation. The functions $f_s\left(t-t_0^s\right)$, $\dot{f}_s\left(t-t_0^s\right)$, $\ddot{f}_s\left(t-t_0^s\right)$ do not have discontinuities at the interpolation nodes. The "Cubic parabola" option is applicable only for seismic analysis. **The input action is the given accelerogram. SCAD automatically determines the velocities and displacements during the integration of the equations of motion. The "Hermite polynomial" option is applicable for all types of dynamic action. In this case, the input action is the forced displacement function** (time function $f_s\left(t-t_0^s\right)$). In case of the seismic action, the given accelerogram has to be integrated twice using the **Accelerogram Editor**.

Example of the text file defining the time function $\varphi_p(t)$ with fixed time step:

File contents	Explanations
0 0.1 #	File type – 0 Time step – 0.1 Control character – #
1	$\varphi_p(0)=1$
2	$\varphi_p(0.1)=2$
-1	$\varphi_p(0.2)=-1$
0.5	$\varphi_p(0.3)=0.5$
1	$\varphi_p(0.4)=1$
0	$\varphi_p(t)=0 \mid t=0.5, t=0.6, t=0.7,.....,t=T_{dur}$

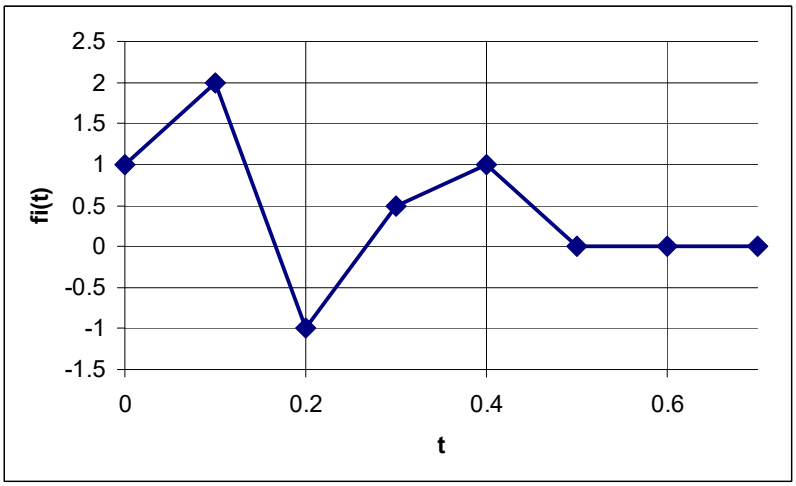


Figure 15.2-5.

Fig. 15.2-5 shows the relationship $\varphi_p(t)$ given in the table for the values $t \le 0.7$. If the integration step does not coincide with the time for which the time function is specified, the linear interpolation is performed. It should be taken into account when specifying the loads rapidly oscillating in time.

It is also possible to use function $\varphi_p(t)$ with variable time step.

Example of the text file defining the time function $\varphi_p(t)$ with variable time step:

File contents	Explanations
10 #	File type – 10 Control character – #
0 1	$\varphi_p(0)=1$
0.1 2	$\varphi_p(0.1)=2$
0.3 -2	$\phi_p(0.3)=-1$
0.7 0.5	$\phi_p(0.7)=0.5$
1 0	$\phi_p(1)=0$, $\phi_p(t)=0,\ t>=1$

In the case of a linear interpolation of the time function related to the specified displacements parasitic high-frequency oscillations can occur in the numerical solution for the accelerations in nodes adjacent to those where forced displacements are specified. Two approaches are used to suppress these oscillations. The first one is based on the α-HHT method [3], which suppresses high-frequency oscillations at $\omega\ \Delta t \geq 1$, where ω is the frequency of the corresponding tone, and Δt is the time integration step. In this case the obtained oscillations with frequencies $\omega\ \Delta t < 1$ are reliable. The degree of suppression depends on the parameter α, $\alpha \in [-1/3,0]$. If $\alpha = 0$, there is no suppression of high frequencies, and the α – HHT method turns into the Newmark method. Maximum suppression takes place at $\alpha = -1/3$.

The second approach is based on the interpolation of a given time function by a cubic parabola or Hermite polynomials. If the user specifies an accelerogram (an acceleration vs. time relationship), the "Cubic parabola" option has to be selected in the **Smoothing** column of the table with the parameters of static loadings. The seismogram will be obtained automatically in the process of solving the problem using interpolation of displacements by a third-degree polynomial. The velocities are interpolated by a second-degree polynomial, and the linear interpolation is used for accelerations. This method is applicable only for seismic analysis.

If a seismogram (a displacement vs. time relationship) is obtained with the help of the **Accelerogram Editor**, the "Hermite polynomial" option has to be selected in the **Smoothing** column. In this case, the displacements will be interpolated by third-degree polynomials, the velocities – by second-degree polynomials, and the accelerations – by linear interpolation. This method is applicable for all types of dynamic loads.

If the "Linear interpolation" option is selected, the given time function is interpolated by a first-degree polynomial. If it is a displacement vs. time relationship, its first derivative (velocity) is interpolated by a piecewise constant function, and the second derivative (acceleration) is zero between the interpolation nodes and is represented by Dirac delta functions in the interpolation nodes (at the given time points) represented as Dirac delta functions. If the displacement of the supports is specified, such interpolation makes the system move by jerks, which generate the above parasitic high-frequency oscillations.

A method should be selected for each problem. α – HHT method and high-order interpolation can be applied simultaneously.

If you want to perform the nonlinear dynamic analysis after this nonlinear static analysis, you have to check the **Nonlinear analysis** checkbox. In this case, all loads applied in the static analysis are transferred to the dynamic analysis, therefore, you do not have to apply any loads in order to perform the dynamic analysis. This implementation of the algorithm corresponds to solving a static problem with static loads at the first stage, and the Cauchy problem at the second stage. Moreover, the stress-strain state of all nonlinear finite elements obtained at the end of the static analysis stage is transferred to the dynamic analysis stage as an initial state. Since the problem is nonlinear, the superposition principle is inapplicable, and the solution of the dynamic problem is "counted" from the solution of the static problem.

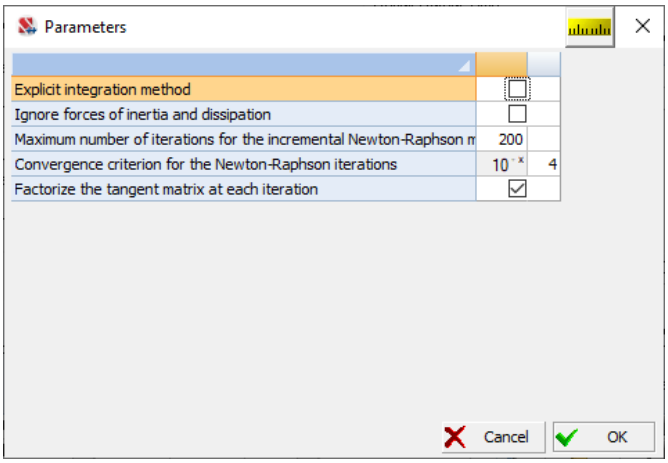


Figure 15.2-6. Parameters of the nonlinear dynamic analysis. Implicit method

Parameters of the nonlinear analysis are specified in the *Parameters* dialog box after clicking the button (Fig. 15.2-6). If the **Explicit integration method** checkbox is unchecked, the implicit predictor-corrector method is selected, the initial data for which are presented in the figure and described below.

If you click the **Ignore the forces of inertia and dissipation** button, a static analysis will be performed using the equations of motion integration algorithm [5]. In a number of cases, this algorithm turns out to be more convenient than the static one, especially for the multiparameter loading. Next, it is necessary to specify the maximum number of iterations of the Newton-Raphson method at a given time step and the allowable value of the error when suppressing the residual during the corrector iterations. If the **Factorize the tangential stiffness matrix at each iteration** option is enabled, the Newton-Raphson method is implemented. Otherwise, the tangential stiffness matrix is assembled and decomposed only at the first iteration (modified Newton-Raphson method). When physical nonlinearity is taken into account, factorization of the tangential stiffness matrix is usually required at each iteration. The implemented algorithm for solving the nonlinear Cauchy problem is described in detail in [5].

If the **Explicit integration method** checkbox is checked, the explicit integration method is selected, which is a combination of approaches [3, 7]. The **Parameters** dialog box will look as follows:

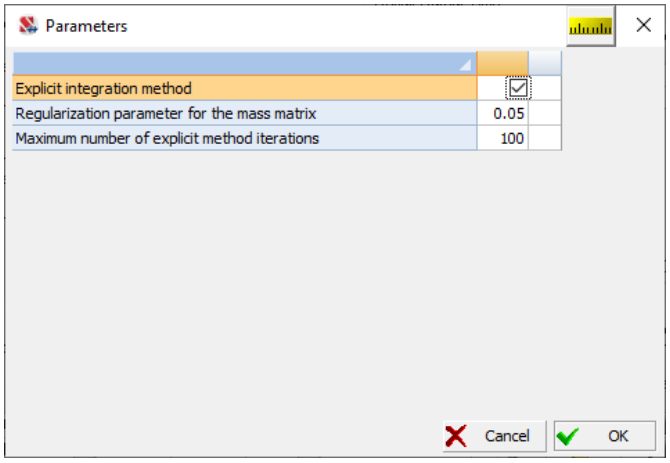


Figure 15.2-7. Parameters of the nonlinear dynamic analysis. Explicit method

An explicit integration method can be more effective than an implicit one for some problems. It does not require assembly and decomposition of the tangent stiffness matrix, and if the structural members are significantly damaged, it can turn out to be much more stable than an implicit one. On the other hand, the explicit method usually requires a much smaller integration step than the implicit one, and the mass matrix must be diagonal and must not be singular. This means that the degrees of freedom corresponding to both translational displacements and rotation angles should not have zero inertial parameters.

There are usually zero elements corresponding to the rotational degrees of freedom in FEA models with a diagonal mass matrix. Therefore, when the explicit method is used, the mass matrix is regularized, and the regularization parameter γ, usually taken in the range [0.0001, 0.1], defines the dummy values $\gamma \cdot \max\{M\}$ added to the zero elements of the diagonal mass matrix. Here $\max\{M\}$ is the largest element of the original mass matrix. If the current element of the mass matrix of the original design model is not equal to zero, then it remains unchanged. The smaller γ, the less distorted the original mass matrix is, but the smaller the integration step Δt required to ensure stability of the numerical solution.

An increase in the number of iterations of the explicit method at each time step Δt increases the accuracy of the solution, but it also increases the computation time, since the internal forces in the system are calculated at each iteration.

It is recommended to start the analysis with the default parameters.

The explicit method of integrating the equations of motion is available only in the nonlinear analysis mode and only when the PARFES solver is selected, which provides a high-performance mode of integrating the equations of motion.

If the **Nonlinear Analysis** checkbox is not checked, a linear analysis of the integration of the equations of motion is performed after this nonlinear static analysis. In this case, the loads and the stress-strain state of the finite elements are not transferred to the dynamic analysis, and the results of the dynamic analysis are counted from the non-deformed state. The influence of nonlinear static analysis on the subsequent linear dynamic analysis is attributed only to the fact that the dynamic analysis uses the tangential stiffness matrix obtained at the end of the nonlinear static analysis – small linear oscillations with respect to the static equilibrium position determined at the nonlinear static analysis stage are considered. All nonlinear finite elements are replaced by their linear counterparts.

Specifying Nodal Actions

Loads acting on the design model are specified as nodal forces and moments. It is necessary to perform the following actions to specify them:

- specify (select from the list in the **Loadings** toolbar) a dynamic loading created earlier with the type of action **Direct integration …**;
- invoke the **Parameters of dynamic loads** operation, the **Nodal Load with Time** dialog box will appear (Fig. 15.2-8);
- specify the name of the file with the .txt extension which describes the graph of the load variation with time for a particular group of nodes in the **Graph of the load variation with time** text field or select it from the list (the **File** button); it should be noted that different loads in a node can refer to different graphs;
- specify the values of scaling factors for the nodal forces and moments of the above group of nodes, and if necessary the delay time of the action compared to that specified in the graph;
- close the dialog by clicking the **OK** button;
- select the nodes included in the described group in the model, and click the **OK** button in the toolbar;
- repeat the above actions for other groups of nodes;
- save the loading after completing the above actions.

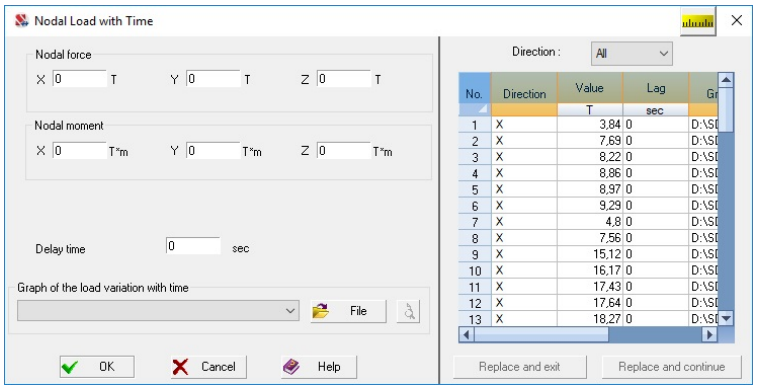


Figure 15.2-8. *The* **Nodal Load with Time** *dialog box*

15.3. Calculation

A log (Fig. 15.3-1) is generated in the calculation process, which in particular contains the following information concerning the direct integration of the equations of motion:

Direct integration of the equations of motion - Newmark method;

Number of integration steps: ... ;

Number of points for results recording: ...;

Integration step dt: ...;

Modal damping parameter: ksi1 = ...;

Modal damping parameter: ksi2 = ...;

Damping parameter alpha - multiplier for matrix M: ...;

Damping parameter beta - multiplier for matrix K: ....

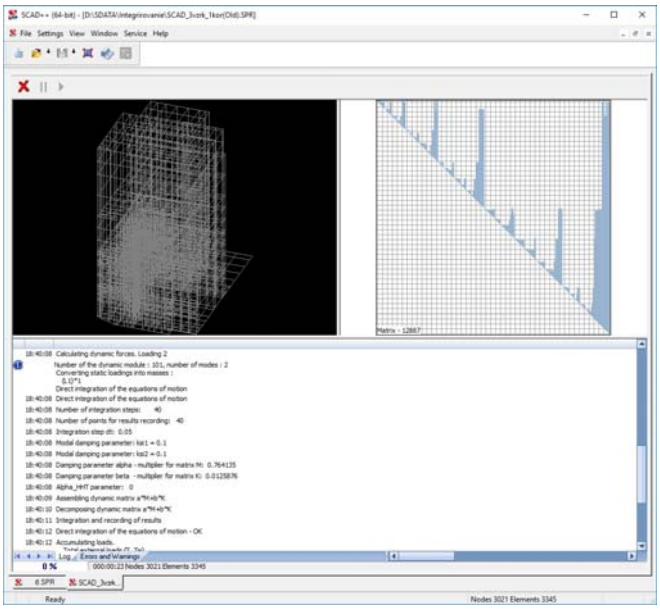


Figure 15.3-1. *Analysis log at the direct integration*

15.4. Results of the Calculation

Results of the Calculation at a Fixed Time

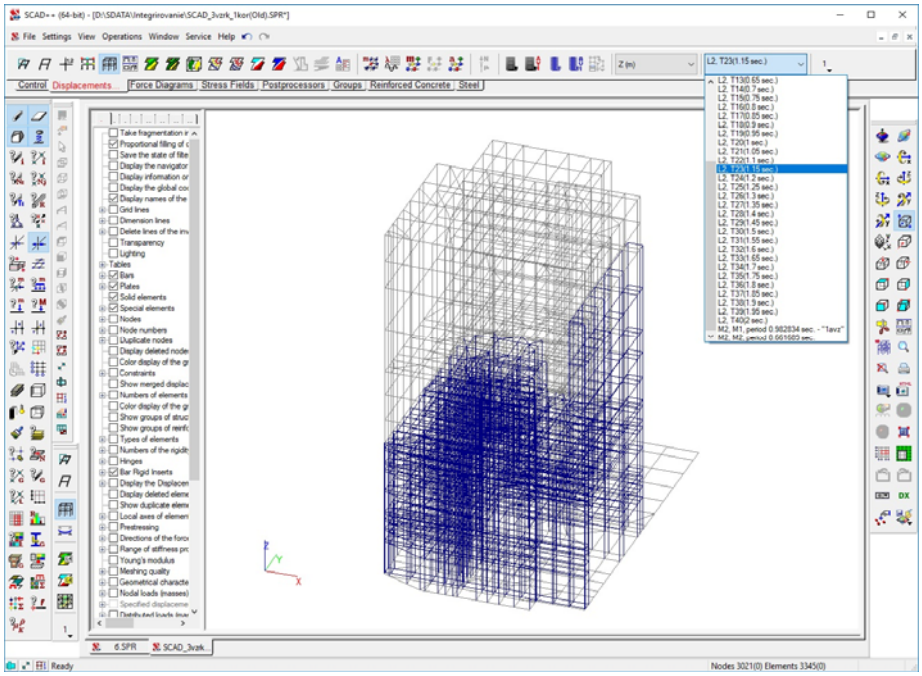


Figure 15.4-1.

The results of the calculation are available in the graphical and tabular postprocessors (Fig. 15.4-1). The number of the static loading and the time for the dynamic loading are selected in the drop-down list of loadings in the respective toolbar (Fig. 15.4-1). Results of the calculation can be displayed for the selected moment in time, for example, as the modes of displacements (Fig. 15.4-2), force diagram (Fig. 15.4-3) or isofields of stresses (Fig. 15.4-4).

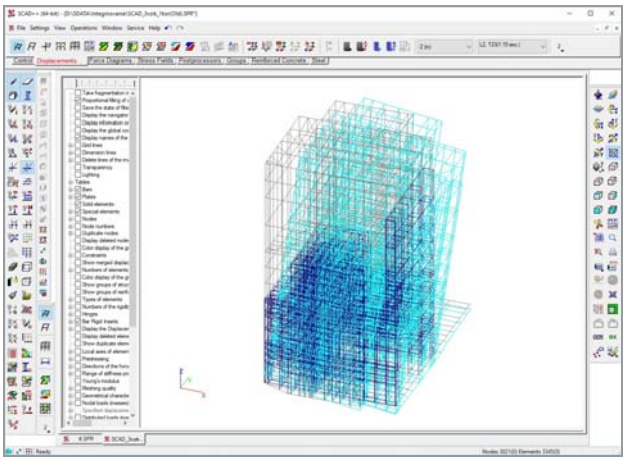


Figure 15.4-2.

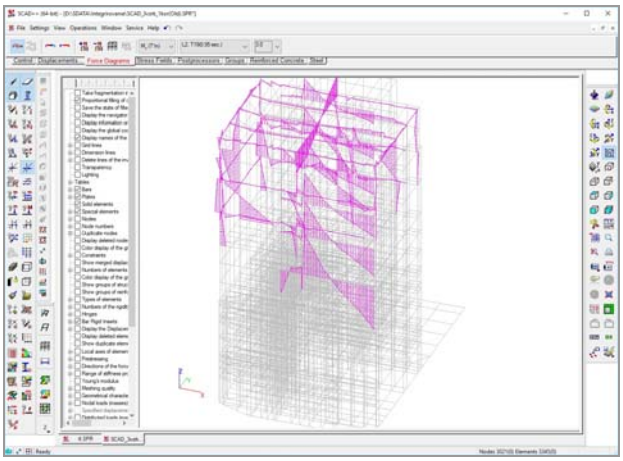


Figure 15.4-3.

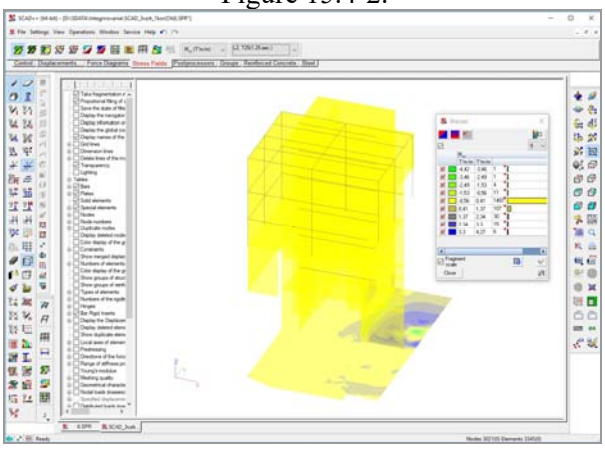


Figure 15.4-4.

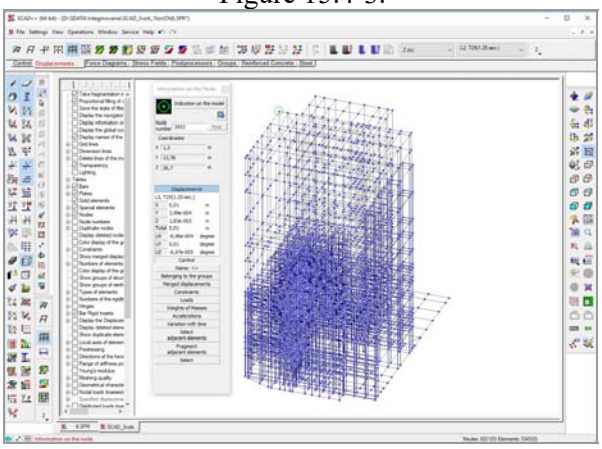


Figure 15.4-5.

Graphs of Displacements, Velocities and Accelerations of Nodes

When analyzing the deformed state of the structure at the selected moment in time you can obtain the graphs of displacements, velocities and accelerations in each node for the whole period of the analysis. The **Variation with time** operation enables to do it which is invoked by clicking the respective button in the **Information on the Node** dialog box (Fig. 15.4-5).

The **Graphs of nodal displacements with time** dialog box (Fig. 15.4-6) appears after invoking the operation, where you can select the type of the analyzed information (displacements, velocities, accelerations) in the list, and also the set of graphs (directions X, Y, Z and the total one), their style and color, background color.

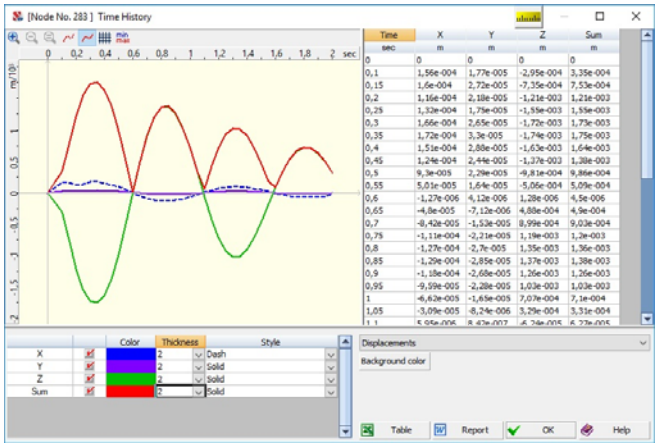


Figure 15.4-6.

If necessary the graphs and the respective tables can be saved as an MS Word report document (the **Report** button) or exported into xls-file for the further processing in MS Excel (the **Table** button).

Clicking the button in the toolbar invokes the dialog box with the information on the minimum and maximum values in the graphs. The report document will also contain this information.

The button in the **Displacements** toolbar enables to obtain the graphs of displacement variation in the selected direction (or the total displacement) for several ***pre-selected*** nodes. This operation invokes a dialog similar to that described above. The user can choose to output absolute displacements or displacements relative to a certain node.

Graphs of Forces (Stresses) in Elements

Graphs of forces/stresses in the element for the entire period of the analysis are plotted similarly to the graphs of displacements. The **Variation with time** operation is invoked from the **Information on the Element** dialog box (Fig. 15.4-7). The **[Element No. …] Time History** dialog box (Fig. 15.4-8) appears after invoking the operation.

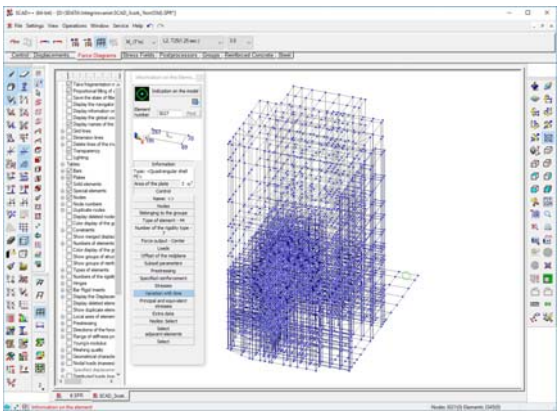


Figure 15.4-7.

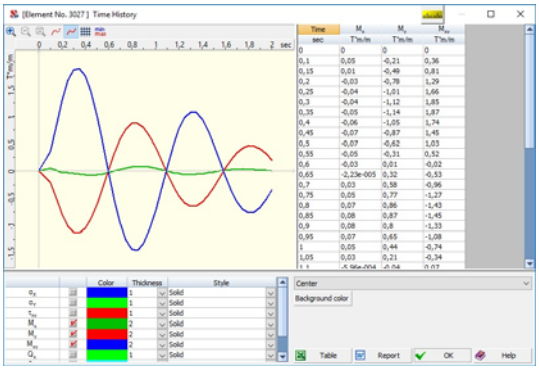


Figure 15.4-8.

The graphs of the variation of force factors with time can be plotted for the given section of bar elements, and for the given node or the center of element for plate and solid elements. You have to make sure that all the simultaneously displayed forces/stresses have identical dimensions.

Clicking the button in the toolbar invokes the dialog box with the information on the minimum and maximum values in the graphs. The report document will also contain this information.

References to Chapter 15

1. Clough R., Penzien J., *Dynamics of Structures*, Moscow, Stroyizdat, 1979.
2. R. W. Clough, J. Penzien *Dynamics of structures*, Third Edition. Computers & Structures Inc., Berkeley, CA, USA, 1995.
3. Miranda I., Ferencz R. M., Hughes T. J. R. *An improved implicit-explicit time integration method for structural dynamics*, Earthquake Engineering and Structural Dynamics, 18, 1989, pp. 643 – 653.
4. Fialko S. *Integration of the equations of motion in SCAD*, SCAD Soft seminar, Kiev, 2012. URL: http://scadsoft.com/download/2012/Fialko_Integration_of_movement_equations.pdf
5. Fialko S. *Application of the finite element method to the strength and load-bearing capacity analysis of thin-walled reinforced concrete structures taking into account the physical nonlinearity*, SCAD Soft, ASV, Moscow, 2018.
6. Fialko S. *The problem of integrating the equations of motions in the seismic analysis implemented in SCAD*, SCAD Soft seminar, Moscow, 2018. URL: https://scadsoft.com/download/2018_04_Msk/23_(2018)_O_postanovke_zadachi_integrirovaniya_uravney_dvizheniya_pri_raschete...(Fialko_S.Y)_Kiev.pdf.
7. Hughes T.J.R., Belytschko T. Course *notes for nonlinear Finite element analysis*, Munich, Germany, 4 – 8 September, 1995.

16. Variation of Models

16.1. Accounting for the Uncertainty of the Parameters of the Design Model

Parameters of the design model used by the researcher are almost always inaccurate values. In fact these parameters are often random values, and when we take a certain value, we deal with a realization of such a random situation.

Loads are one of the most unstable elements of the design model. Even the considered self-weight of the structures according to the current regulations can vary by about ±10…15%, and it is one of the most stable loads. Other loads change in a much wider range of both the value and direction of action, and the location on the structure. This feature of the loading has initiated the development of various methods of searching for the so-called "unfavorable loading", i.e. finding such a combination of different components of external actions, at which the extremum of a certain internal force, stress in the cross-section, displacement in the node, etc. takes place. Starting from the classical works on the justification of the methods of finding the most unfavorable position of a group of moving loads on the influence line, and ending with the works based on the application of the modern methods of solving the multi-criteria optimization problems, this problem has drawn close attention of specialists. The more surprising is the fact that the problem of accounting for the variability of other parameters of the design model turned out to be not so popular, although its practical value is extremely high.

16.2. Differences between Models

SCAD has a special mode which enables to process the results of the calculation of several close *variants* of the design model. The proximity of the variants is interpreted in the sense that they are topologically similar, contain the same number of nodes and elements, and allow only certain differences between the compared design models:

- the possibility of using different types of elements including the elements of the "hidden" type which model the absence of elements without changing their total number;
- the possibility of changing the stiffness properties of finite elements including the use of zero values of some rigidities and different subsoil parameters;
- differences in the system of imposed constraints and/or boundary conditions of elements (hinges, rigid inserts).

16.3. Generate a Package of Models

In order to perform the calculation for several models, it is necessary to create a new project, selecting the **Variation of models** radio button in the **New Project** dialog box (Fig. 16.3-1). After closing this dialog box the **Save As…** dialog box appears where you have to specify the name of the project file (with the **.vpr** extension). This name should not repeat any of the names of the files of the problems included in the package of models. For example, if the files of the problems had names P1, P2, P3, then P123 or any other name different from the above ones can be used as the name of the package of models, so that the names of the work files and the print files were different (see Appendix).

You should keep in mind that the file of the variational project and the files of the problems included in this project should be in the same directory.

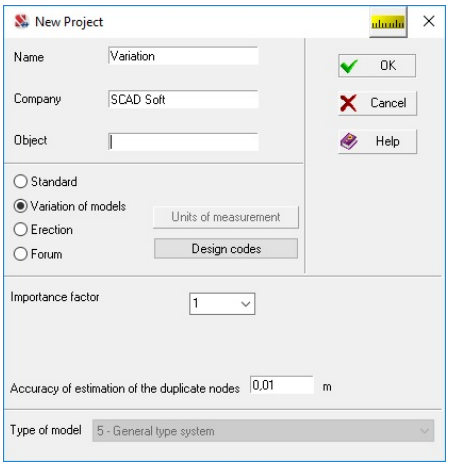


Figure 16.3-1. *The* **New Project** *dialog box*

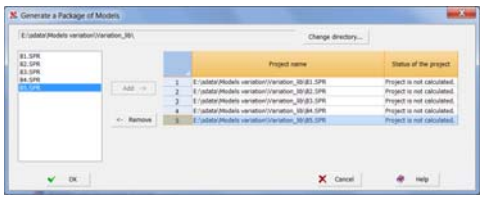


Figure 16.3-2. *The* **Generate a Package of Models** *dialog box*

Only the list of files of the problems included in the package is used as the initial data of the considered mode. The list is specified in the **Generate a Package of Models** dialog box (Fig. 16.3-2), which can be invoked from the respective section of the **Project tree** (Fig. 16.3-3). In order to include the problem in the package select the name of the project file in the left list and transfer it to the right one by clicking the **Add** button. The status of the project (whether the calculation is performed) is reflected in the respective column of the right list.

If the models differ in the number of "hidden" elements, the problems should be included in the project in such an order, so that the “maximum model” is first, i.e. a model in which all elements are present explicitly. A package of models is merged into one design model after the initialization of the calculation in the project tree.

When creating a package of models, the software performs the checks of the regulated similarity of a series of problems. If the result of this check is positive, the results of the calculations of all the variants of the design model are merged into one array (as if one model is analyzed for all variants of the loading, which appeared in all the calculations performed earlier).

The calculation can be performed only when all the problems included in the variation of models are already calculated. You should also make sure that none of the models of the package is open in the current or other copy of **SCAD**.

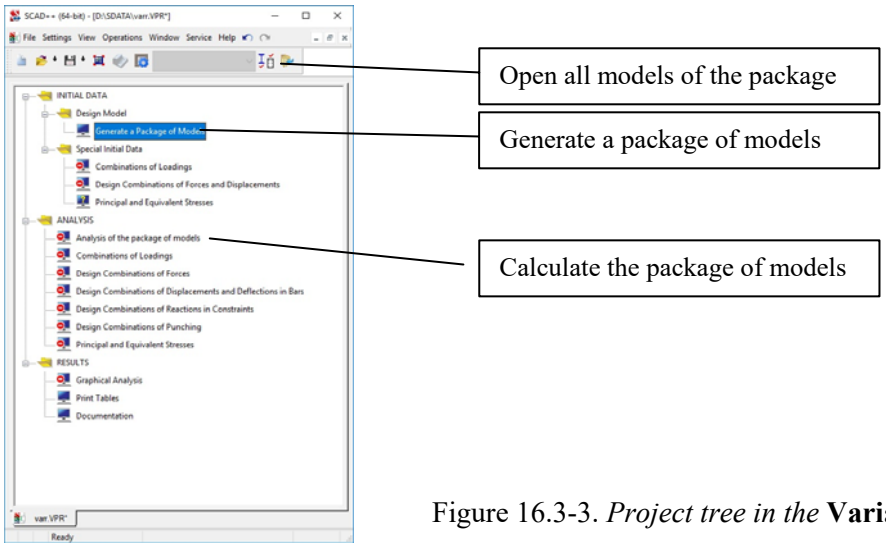


Figure 16.3-3. *Project tree in the* **Variations of Models** *mode*

You can perform the generation of the design combinations of forces (**DCF**), create combinations of loadings, select reinforcement in the elements of reinforced concrete structures, check or select sections of steel structures for the problem thus obtained.

If the reactions in constraints are calculated in all the problems included in the package of models, the reactions in constraints of the variation will be determined automatically during the calculation.

If you want to modify all the problems of the package of models or invoke their calculation, you can use the **Open all models of the package** operation — (Fig. 16.3-3), and all the design models will be loaded into the preprocessor in the result.

16.4. Design Combinations of Forces

The initial data for the generation of DCF (see Chapter 18) are specified according to the same rules as for a standard project. The list of loadings will include a set of loadings of each local problem in the order they appear in the variational package. Mutual exclusions between the sets of loadings belonging to different problems of the package are set automatically by default. However, the software enables to refuse the automatic mutual exclusion with the help of the respective checkbox. It should be done very carefully, because (in the general case) the superposition principle, which is the basis for the calculation of DCF, is not applicable.

If the local problems included in the package contain the initial data for the generation of DCF, the relationship between loadings set for each problem (combinations, mutual exclusions, related loadings) are saved in the initial data of DCF of the variational package. The user can modify these data.

16.5. Results of the Calculation and Their Analysis

In addition to the parameters described in Chapter 8 the number of the problem in the package and the number of the loading in this problem are specified in the lines of the **Select loading** drop-down list (Fig. 16.5-1) in the mode of the graphical analysis of the calculation results (deformations, force diagrams, etc.). The additional information for dynamic loadings is given only for the first oscillation mode.

The check of the load-bearing capacity of elements of reinforced concrete and steel structures, and the selection of reinforcement and rolled steel sections is performed for the design combinations of forces obtained for the whole package of models.

Since the models can have different sizes of sections including the thicknesses of plate elements, it is necessary to specify the model on the basis of which the designing will be performed. The model is selected from the **Specify the model for designing** list in the **General Data** toolbar (Fig. 16.5-2), which appears simultaneously with the project tree when loading the package of problems (file with the **.VPR** extension). If the model is not specified, the necessary information will be selected from the first model in the list of the package of problems by default.

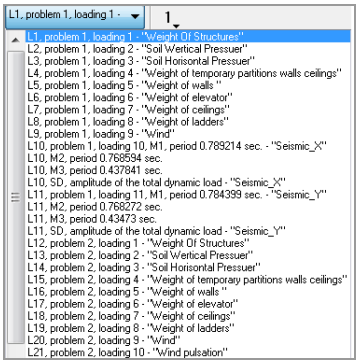


Figure 16.5-1. *The* **Select loading** *drop-down list* for the variation of models

> In order to enter the **Steel** or **Reinforced Concrete** postprocessors you have to (for example) select a loading which corresponds to the model selected for designing in the *Displacements* or *Forces* tabs

Data necessary for performing the operations dealing with designing can be specified in different models. In order to transfer them into a file of the variational project, you should specify the type of data and the project included in the package of problems they are taken from. The **Import data from the project to the file of variations** operation — , which is invoked from the **General Data** toolbar (Fig. 16.5-2) is used for this purpose. After invoking this operation the respective dialog box (Fig. 16.5-3) appears, where you can transfer the following data:

- general settings;
- information on the groups of elements of reinforced concrete structures;

- information on the groups of elements of steel structures;
- information on the groups of unification of elements of steel structures;
- information for checking the specified sections of elements of steel structures;
- information for checking the specified reinforcement.

The main rule of the transfer is that each type of data is taken entirely from one model. It means that structural groups and unification groups can be described either in one or in several models, but a certain set of groups, for example, elements of steel structures, can be taken from one of the models for the variational package.

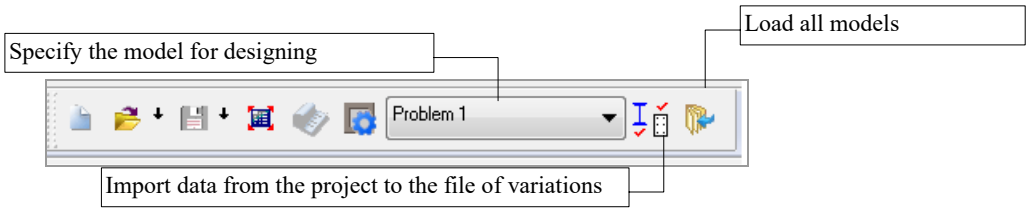


Figure 16.5-2. *The* **General Data** *toolbar in the* **Variation of models** *mode*

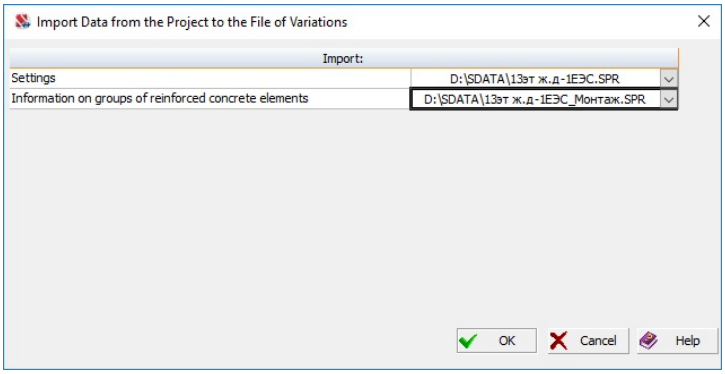


Figure 16.5-3. *The* **Import Data from the Project to the File of Variations** *dialog box*

16.6. Using the Variation of Models

Stiffness Properties

Stiffness properties of the elements of a design model can be quite uncertain. This may be due to the natural spread of values, for example, of such a parameter as the elastic modulus of concrete, the nominal value of which is traditionally selected according to the recommendations of the design codes and the variability of implementations, including the possible changes within the object and variations with time, is traditionally not considered. Despite a fairly narrow corridor of the modulus variability, in many cases, there are serious reasons to consider the variability of this parameter.

Consideration of damages accumulated in the structure during the operation belongs to such cases. These damages should be taken into account in the calculations dealing with the structural assessment of the existing structures, and taken according to the values detected in the examinations. However, not all elements of the existing structure are available for the examination. Therefore, the stiffness properties of such elements are judged by the state of other available parts of the structure. The uncertainty range can be quite large in this case.

The considered cases include many underground structures, elements hidden inside concrete casings, and other structures, for which the so-called “hidden works acceptance certificates” are prepared during the construction.

Compliance of the Subsoil

The elastic parameters of the natural subsoil are characterized by the large variability and, consequently, uncertainty of values. The incompleteness of the available geotechnical data, and the approximate design model of the subgrade (Winkler, Pasternak, half-space, etc) play their role as well. Apparently, in the best

cases these parameters may have a spread of values of about ±30%, and in some situations substantially larger deviations may take place.

Thus, the USA design codes for nuclear plants recommend to introduce an artificial variation of the shear modulus of soil G, assuming that it can change from $G^*(1 + G_v)$ to $G^*/(1 + G_v)$, where G^* is its most probable estimate, and G_v is its coefficient of variation taken not less than 0,5.

The calculation for several models also enables to correctly take into account different soil behavior when loading the structure with permanent (long-term) and short-term, including the dynamic, loads. In this case a model where the values of the deformation modulus are used as the initial data for the calculation of the subsoil parameters should be used for the long-term loads, and a model where the elastic modulus is used should be used for the short-term and dynamic actions.

Nodal Connections

Idealized conditions of the connections between the elements in nodes (rigid, hinged) are usually assumed when selecting a design model, although these connections have certain elastic properties. However, it is very difficult to assess their compliance, since experience shows that this assessment can be practically carried out only in the experiment. There are almost no reliable computational methods for obtaining such assessments. Therefore, engineers have to set the values of compliance of the nodal connections quite arbitrarily, focusing on the results of the experimental studies of "similar" structures.

There is a certain possibility of making particular designs of connections hinged or rigid, and it is often used in designing, when the designer has means for the actual implementation of the solution assumed in the design model. However, this option is not available when an existing structure is checked and the design can not be adapted to the design model. Here the real model often remains unclear, and the check of the structure has to be performed under this uncertainty.

17. Combinations of Loadings

The **Combinations of loadings** mode enables to obtain the indicators of the stress-strain state of the system which is under the action of several loadings simultaneously. Any loading can be included in the combination with a certain factor. This enables, for example, to answer the following question: what are the values of displacements from the action of the “self-weight” loading taken with a factor of 0,9 and the "snow" loading taken with a factor of 0,7. If these loadings were given by their design values, it is not difficult to see that we will obtain displacements from characteristic loads.

If a combination includes a dynamic loading, the value of the total dynamic load obtained with the help of a nonlinear procedure for displacements or forces of all the considered modes of natural oscillations is used in the calculation of forces and displacements of this combination. For example, the following nonlinear procedure is used for the seismic loading: the squares of the results are summed for each mode, and then the square root is taken of this sum. Dynamic loadings are alternating, therefore it is necessary to select the sign of the result. **SCAD** selects the sign in accordance with the sign of the mode with the maximum obtained absolute value of force/reaction/... . The respective static wind loading is considered for the wind loading (the sign of the result is taken in accordance with the sign of the static wind loading). Individual oscillation modes are not included in the combination, since the result of the dynamic analysis is obtained with the help of a nonlinear procedure.

Unlike the **Design combinations of forces** mode where it was not known in advance which combination of loadings would be unfavorable (design) for the stress in a certain section, and the search for such a combination was performed automatically according to the rules of SNiP, you have to specify the considered combination explicitly in the **Combinations of loadings** mode.

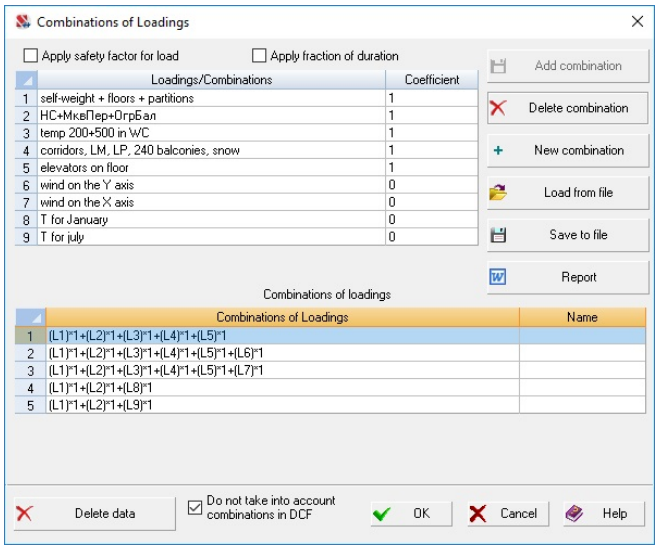


Figure 17-1. *The* **Combinations of Loadings** *dialog box*

Combinations of loadings can be calculated together with the whole problem, or separately, after the main calculation. In the first case the initial data are prepared after completing the generation of the design model and specifying loadings. In the second case the combinations can be specified after completing the analysis of loadings.

The mode of preparing the initial data for the calculation of the combinations of loadings is invoked from the **Project tree** in the **Special Initial Data** section. This mode becomes available only after specifying the loadings. Data are prepared in the **Combinations of Loadings** dialog box (Fig. 17-1).

The data are specified in the following order:

- in the **Coefficients** column specify the values of coefficients with which the loadings are included in the current combination;
- click the **Add combination** button, and the formula of the current combination will be displayed in the information window;

– click the **New combination** button and specify the number of the following combination, the coefficients will have zero values, and the software will be ready for the input of new data.

When creating the combination the software enables to take into account the corresponding safety factors for load and/or fraction of duration for each loading included in it. The coefficients with which the loadings are included in a combination are divided by the safety factor for load and/or multiplied by the coefficient of the fraction of duration when the respective checkboxes in the **Combinations of Loadings** dialog box are checked. It enables to automate the process of creating the combinations which correspond to the characteristic, long-term, or characteristic long-term combinations. These checkboxes described above should not be used if the combination includes other combinations (the latter will be ignored). If EN 1990 is selected for calculating load combinations, these options are not available.

It should be noted that combinations created earlier can be included in the combination together with the loadings. The current combination (its number is specified in the **Number of the combination** field) can be deleted by clicking the **Delete combination** button, and all the specified combinations will be deleted after clicking the **Delete data** button.

When the *number of loadings* changes, the data for the calculation of combinations of loadings should be *clarified*, since some combinations can be deleted.

There are restrictions on the specification of combinations in the **Erection** mode: "independent" loadings of one erection stage can be included in a combination. The main loading will be added automatically to the whole combination, including additions to the main loading (given as groups of loads) with the specified coefficients.

The calculated values of displacements and forces from a combination which includes dynamic loadings are nonlinear and correspond to different moments in time. The sign may not be considered at the summation. The loss of equilibrium in the nodes can occur.

Moreover, the combinations can be created on the basis of the design combinations of forces generated in the **DCF** postprocessor with the help of the **Create combinations of loadings for DCF** operation (button —). To do this, click the **Load from file** button and the standard **Open** dialog box of the Windows environment will appear. The loaded file with the information on the combinations has the **.rsu2comb** extension.

In the **Variation of models** and **Erection** modes the combinations of loadings can be specified for a single model (stage), or using loadings belonging to different design models (erection stages). It should be noted that combining loadings from different problems is usually unacceptable because it violates the superposition principle. Nevertheless, the authors left this possibility (of course, the responsibility rests with the user) in order to enable to create a combination of "the difference between similar loadings" type and to assess the differences of the stress-strain states of various models (stages).

When specifying DCF the software enables to use not only loadings but also their combinations. If such a possibility is not necessary, it is recommended to check the **Do not take into account combinations in DCF** checkbox when specifying the combinations of loadings.

18. Design Combinations

18.1. Fundamental Principles

To determine the design combinations means to find those combinations of individual loadings which may be critical (the most dangerous) for each checked element, each section of the element and each node.
SCAD enables to create the following design combinations:

- design combinations of forces (DCF);
- design combinations of forces of progressive collapse;
- design combinations of deflections (DCD);
- design combinations of displacements;
- design combinations of reactions in constraints (DCR);
- design combinations of punching.

The basis for the selection of unfavorable design combinations is the superposition principle, which means that the calculations are performed as for a linearly behaving system. However, the results of the selection of unfavorable design combinations of forces are used, for example, to check the strength of reinforced concrete structures where the behavior of a section with cracks and yielding of reinforcement is clearly non-linear. There is an apparent contradiction, which can be resolved using recommendations of Sec.5.1.2 of SNiP 52-01-2003: "... it is recommended to take into account the effect of physical non-linearity by adjusting the results of linear analysis based on the data from experimental studies, nonlinear modeling, results of the analysis of similar structures, and expert assessments".

Unfavorable combinations of forces that create maximum elastic stresses in characteristic points or characteristic areas of finite elements are selected using the superposition principle. An individual approach for specifying the DCF selection criteria is used for different types of finite elements (bar, plate, etc.). Maximum displacements of nodes are selected in a similar way.

Linearity also implies that a whole loading will be included in DCF, or it will not be considered (if it creates an unloading effect), and thus, we can have only 2^n combinations (where n is the number of loadings) you have to choose from. It is not difficult to see that for real values of parameter n, the number of possible combinations becomes so large that solving the problem by direct enumeration of variants is impossible, therefore a special algorithm is used to solve this extreme problem.

The software determines the combinations of both design and characteristic SSS. *Long-term* values (as well as the characteristic and characteristic long-term ones) of the respective forces, stresses and displacements are calculated for the obtained unfavorable (according to a certain criterion) combinations of *design values*. The software also performs the search of unfavorable combinations of the characteristic values and calculates the design, design long-term, and characteristic long-term values of the respective forces, stresses, displacements etc. for them. Combinations which include loadings corresponding to the seismic analysis according to SP 14.13330.2014 at the *Maximum design earthquake* design situation are an exception. The characteristic and characteristic long-term values are not calculated in this case.

It should be noted that a criterion used when searching for an unfavorable combination is specified for each combination when printing tables and in the information on the element/node. A situation can arise when, for example, a criterion of maximum normal stresses (used in the analysis of design forces) is specified, but the set of the respective characteristic forces does not result in a maximum characteristic stress.

Design codes regulate the limitations of the characteristic values of displacements, therefore only the characteristic and characteristic long-term combinations of displacements are calculated for each node.

Punching analysis of flat reinforced concrete elements (plates) is required by the design codes for concrete and reinforced concrete structures. **SCAD** enables to calculate the unfavorable (in terms of punching) combinations of loadings. The calculations are performed in the nodes of horizontal plate elements the vertical bars are connected to.

Loadings acting on the considered structure may belong to one of the following types depending on the duration of loads (according to the SNiP classification):

- permanent;
- long-term;

- short-term;
- special.

In order to simplify the algorithm for selecting combinations, the list of types is expanded. Crane loads (although in the codes they belong to short-term ones) and inactive loads, i.e. not involved in the selection of unfavorable combinations are added to this list. For example, a static component of the wind load can be made “inactive” in the case of specifying a dynamic loading "Wind action", where it will be included automatically. Moreover, the list of short-term loads includes the Special load taken into account at the seismic action. Its use enables to take into account the requirements of the industry standards on accounting for the additional load acting simultaneously with the seismic one.

In order to take a correct safety factor for load for each type of loading, the lists of load types corresponding to the codes are provided, for example the data from Table 1 of SNiP 2.01.07-85* are used for permanent loads (the weight of structures and soils), the data from Tables 2 and 3 are used for long-term loads, etc. Other codes also have similar tables of safety factors for load. In those cases when the safety factor for load can not be determined unambiguously, it has to be specified by the user.

Besides the safety factor for load the loading is characterized by the fraction of duration and the simultaneity of loads included in it. This means that it is correct to combine loads within one loading which have identical first two characteristics, and which always act simultaneously.

Combinations of loadings can be involved in the selection of design combinations of loads along with loadings. Each created combination can be considered in the list of loadings. The type of the loading and the type of the load for the combination are obviously specified by the user.

It is specified in the codes that depending on the type of loads included in the combinations the respective reduction factors appear.

The “inactive” type is assigned to all loadings and combinations in the initial state of tables with the initial data.

The analysis is based on the codes selected by the user. You can select another code with the help of the **Setting the Design Codes** item of the ***Settings*** menu.

Design combinations can be calculated together with the whole problem, or separately, after the calculation for static and dynamic actions. In the first case the initial data are prepared after completing the generation of the design model and only after specifying all loadings, and in the second case the data can be specified after completing the analysis of loadings. The design combinations of displacements can be calculated only after completing the main calculation.

The design combinations of loads are determined separately for each element (in fact — for each section of the element) and the determined set of combinations may be different for different elements. Thus, for the whole system we obtain combinations of loadings which physically do not act simultaneously, therefore it is impossible to plot a “diagram of DCF” or “isofields of DCF”. If in addition to this more than one DCF corresponds to each element, this "impossibility" is even more aggravated. It should be noted that if you want to see the result of simultaneous action of several loadings, you should use the **Combinations of loadings** mode.

Logical Relationship between Loadings

When determining the design combinations the logical relationship between loadings reflecting their physical meaning and requirements regulated by different codes are taken into account. There are the following types of logical relationship:

- mutually exclusive, which cannot be simultaneously included in one combination (for example, wind on the left and wind on the right, seismic action along different coordinate axes, etc.);
- related, when loadings cannot be included in a design combination without the loading they were generated from. Forces from braking of a crane trolley can not be included in a combination if this combination does not include the forces from the pressure of the crane wheels, though the load

from the crane can act without the combination with the brake load. Similarly, the forces from the seismic load can not be included in a combination without the loadings (e.g., temporary) which are a part of the mass of the structure which determined the dynamic characteristics of the seismic motion, though temporary loads can be included in a combination without seismic loads;

- simultaneous, for example, wind pressure and uplift specified in different loadings. It is necessary to distinguish between the cases when the loads must act simultaneously, or can act separately.

Moreover, you can specify the sign variation in the description of the characteristics of loadings (with the same absolute value of the vector). Almost all dynamic loads are like this.

The logical relationship obviously cannot be contradictory:

- permanent and temporary loads can not be combined;
- related loading can not have two mutually exclusive generating loadings etc.

18.2. Criteria of DCF

Bars

Extreme values of normal and shear stresses in the control points of the section are used here as a criterion for the determination of DCF (Fig. 18.2-1).

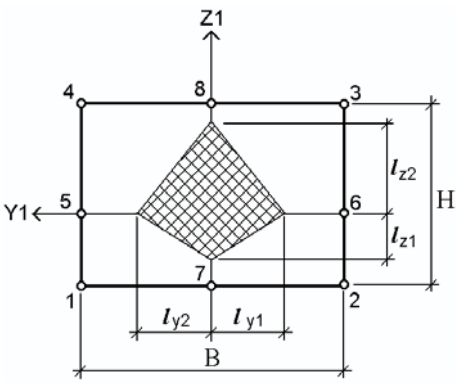


Figure 18.2-1. *Bounding rectangle of an arbitrary section*

The following formula is used for normal stresses:

$$\sigma_k = \frac{N}{F} \pm \frac{M_y z_k}{I_y} \pm \frac{M_z y_k}{I_z}, \qquad (18.2.1)$$

where k is a point of the bar section (k = 1…8). This formula can be transformed in the following way when $y = \pm\frac{B}{2}$ and $z = \pm\frac{H}{2}$:

$$\sigma_k F = N \pm \frac{M_y}{l_{z,i}} \pm \frac{M_z}{l_{y,i}}, \qquad (18.2.2)$$

where $l_{y,i}$ and $l_{z,i}$ are core sizes in the bar section (i = 1, 2).

This approach enables to determine extreme normal stresses in a section of any shape.

An approximate formula is used for shear stresses:

$$\tau_y F = \frac{Q_y}{2} \pm \frac{M_{kp}}{2(l_{y1} + l_{y2})} \qquad \tau_z F = \frac{Q_z}{2} \pm \frac{M_{kp}}{2(l_{z1} + l_{z2})}. \qquad (18.2.3)$$

The numbering of the criteria and their corresponding stresses calculated by the formulas (18.2.1, 18.2.3) is given in Table 18.2-1. Extreme positive and negative values of the longitudinal force (criteria No.17 and 18) and shear forces (criteria No. 27…34) used when selecting reinforcement in the elements of reinforced concrete structures and when checking the load-bearing capacity of elements of steel structures are calculated as well.

Table 18.2-1

Criterion	1	2	3	4	5	6	7	8	9	10
Value	σ_{1+}	σ_{1-}	σ_{2+}	σ_{2-}	σ_{3+}	σ_{3-}	σ_{4+}	σ_{4-}	τ_{y7+}	τ_{y7-}
Criterion	11	12	13	14	15	16	17	18	19	20
Value	τ_{y8+}	τ_{y8-}	τ_{z5+}	τ_{z5-}	τ_{z6+}	τ_{z6-}	N^+	N^-	σ_{7+}	σ_{7-}
Criterion	21	22	23	24	25	26	27	28	29	30
Value	σ_{8+}	σ_{8-}	σ_{5+}	σ_{5-}	σ_{6+}	σ_{6-}	Q_{y+}	Q_{y-}	Q_{y+}	Q_{y-}
Criterion							31	32	33	34
Value							Q_{z+}	Q_{z-}	Q_{z+}	Q_{z-}
Criterion							51	52	53	54
Value							M_{y+}	M_{y-}	M_{z+}	M_{z-}

The following criterion is used to find the unfavorable value of the eccentricity

$$\sigma F = N \pm \frac{M_y}{l_{z,i}} \sin\frac{n\pi}{8} \pm \frac{M_z}{l_{y,i}} \sin\frac{m\pi}{8}, \tag{18.2.4}$$

where n, m=0, 1, 2, 3, 4.

This criterion is important for reinforced concrete structures.

Criteria calculated by the formula (18.2.4) have three-digit numbers according to the Table 18.2-2.

Table 18.2-2

Number of the criterion	1nm	2nm	3nm	4nm	5nm	6nm	7nm	8nm
Value	σ_{++}	$-\sigma_{++}$	σ_{+-}	$-\sigma_{+-}$	σ_{-+}	$-\sigma_{-+}$	σ_{--}	$-\sigma_{--}$

Thus, for example, the criterion with the number 532 means the maximum value

$$N - \frac{M_y}{l_{z,i}} \sin\frac{3\pi}{8} + \frac{M_z}{l_{y,i}} \sin\frac{2\pi}{8}.$$

Membranes (Plane Stress State)

In the general case the principal stresses in the same point of the structure at different loadings have different orientations, therefore their use as a criterion for selecting DCF is impossible. Therefore, the DCF are determined here by the values of normal and shear stresses in the differently oriented areas (the orientation is defined by the angle α in the center of gravity of the finite element). The calculation is performed according to the formulas:

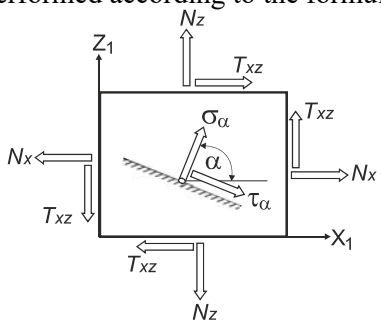


Figure 18.2-2.

$$\sigma(\alpha) = N_x \cos^2\alpha + N_z \sin^2\alpha + T_{xz} \sin 2\alpha\,; \tag{18.2.5}$$

$$\tau(\alpha) = \frac{1}{2}\left(N_z - N_x\right) \sin 2\alpha + T_{xz} \cos 2\alpha\,. \tag{18.2.6}$$

The designations are given in Fig. 18.2-2. Normal stresses are calculated with a fixed step of the angle.

Plates

The approach used here is similar to that described above for membranes. The bending moments and torque in the plate enable to determine the normal and shear stresses on the upper and lower surfaces of the plate. These stresses have equal absolute values therefore by analogy with the formulas (18.2.5) and (18.2.6) we can write the expressions for moments creating extreme stresses

$$M(\alpha) = M_x \cos^2\alpha + M_y \sin^2\alpha + M_{xy} \sin 2\alpha\,; \tag{18.2.7}$$

$$M_k(\alpha) = \frac{1}{2}\left(M_y - M_x\right)\sin 2\alpha + M_{xy}\cos 2\alpha\,. \tag{18.2.8}$$

Extreme values of shear forces are calculated as well.

Shells

The stresses on the upper and lower surfaces of the shell are calculated taking into account the membrane stresses and bending forces according to the following relationships:

$$\begin{aligned} \sigma_x^{L/U} &= N_x \pm \frac{6M_x}{h^2}; \\ \sigma_y^{L/U} &= N_y \pm \frac{6M_y}{h^2}; \\ \tau^{L/U} &= T_{xy} \pm \frac{6M_{xy}}{h^2}; \end{aligned} \tag{18.2.9}$$

where h is the thickness of the shell; U and L are indices which indicate belonging to the upper or lower surfaces.

The values of the criteria for shell elements are calculated based on the determined stresses according to the formulas (18.2.5-18.2.6). Calculate moment M and stress N according to the formulas (18.2.5)-(18.2.6). If N is nonzero, the value of the excentricity $\varepsilon=M/N$ is calculated. Extreme values of shear forces are determined as well.

Special Elements

Positive and negative extreme values of forces in elements are taken here as the criteria of determination of DCF .

Solid and Axisymmetric Elements

The extreme values of the stress tensor components and principal stresses are taken as a criterion for determining the DCS.

Numbering of the Criteria of DCF

Numbers of criteria given in Tables18.2-1 and 18.2-2 are used for **bars**.

For **membranes**, **plates** and **shells** the calculations are performed according to the formulas 18.2.3-18.2.9 with a step specified in the initial data, which can take values of 3, 5, 7.5, 9, 15, 22.5, and 45°. The range of the variation of angle α is from 0° to 180° for normal stresses and from 0° to 90° for shear stresses. The default step is 15°. The numbers of the criteria and their corresponding angles are given in Table 18.2-3, where nnn is the angle, and the indices "L" and "U" correspond to the lower and upper surfaces of the shell.

Table 18.2-3

Number of the criterion	0	1nnn	2nnn	3nnn	4nnn	5nnn	6nnn	7nnn	11nnn	12nnn	8001	8002	8003	8004	8005
membranes	σ	$-\sigma$	τ	$-\tau$											
plates	M	-M	$M_к$	$-M_к$							Q_x	$-Q_x$	Q_y	$-Q_y$	$\lvert Q_x\rvert+\lvert Q_y\rvert$
shells	$\sigma_н$	$-\sigma_н$	$\tau_н$	$-\tau_н$	$\sigma_в$	$-\sigma_в$	$\tau_в$	$-\tau_в$	ε	$-\varepsilon$	Q_x	$-Q_x$	Q_y	$-Q_y$	$\lvert Q_x\rvert+\lvert Q_y\rvert$

Thus, for example, the criterion number 6075 means that τ is calculated on the upper surface of the shell for the angle of 75°.

For **solid** and **axisymmetric** elements the numbers of the criteria are given in Table **18.2-4**, where σ_i, i=1,2,3 are principal stresses.

Table 18.2-4

Number of the criterion	1	2	3	4	5	6
Value	σ_{x+}	σ_{x-}	σ_{y+}	σ_{y-}	σ_{z+}	σ_{z-}
Number of the criterion	7	8	9	10	11	12
Value	τ_{xy+}	τ_{xy-}	τ_{xz+}	τ_{xz-}	τ_{yz+}	τ_{yz-}
Number of the criterion	13	14	15	16	17	18
Value	σ_{1+}	σ_{1-}	σ_{2+}	σ_{1-}	σ_{3+}	σ_{3-}

18.3. Criteria of DCD

The extreme values of displacements of the node along the axes X, Y, Z are taken as the criteria for determining the design combinations of displacements. The criteria which enable to find the extreme total displacement and the extreme horizontal displacement are added to them.

18.4. Criteria of Design Combinations of Deflections

Unfavorable combinations of deflections are determined in each design section of the bar on the basis of the following criteria:

max X and *min* X;
max Y and *min* Y;
max Z and *min* Z;
$max\sqrt{X^2+Y^2+Z^2}$ and $max\sqrt{X^2+Y^2}$,

where:
X, Y, Z — values of deflections in the section in the global coordinate system.

18.5. Criteria of Design Combinations of Reactions in Constraints

There are the following criteria for determining the design combinations of reactions in constraints:

max R_X and *min* R_X;
max R_Y and *min* R_Y;
max R_Z and *min* R_Z;
max R_{UX} and *min* R_{UX};
max R_{UY} and *min* R_{UY};
max R_{UZ} and *min* R_{UZ};
$max\sqrt{R_X{}^2+R_Y{}^2+R_Z{}^2}$ and $max\sqrt{R_X{}^2+R_Y{}^2}$;
$max\sqrt{R_{UX}{}^2+R_{UY}{}^2+R_{UZ}{}^2}$ and $max\sqrt{R_{UX}{}^2+R_{UY}{}^2}$,

where:
R_X, R_Y, R_Z — reactions along the axes X, Y, Z of the global coordinate system;
R_{UX}, R_{UY}, R_{UZ} — reactions about the axes X, Y, Z of the global coordinate system.

18.6. Criteria of Design Combinations of Punching

When calculating the design combinations of punching the software determines all nodes which belong to the horizontal plates and vertical (and only vertical) bars. The loads in each such node on the horizontal plate are calculated for each loading and combination of loadings. Then the unfavorable combinations of design loads are determined. The rules of combining given in the selected design codes are used. There are the following criteria for determining the design combinations:

max R_X and *min* R_X;
max R_Y and *min* R_Y;

max R_Z and *min* R_Z;
max R_{UX} and *min* R_{UX};
max R_{UY} and *min* R_{UY};
max R_{UZ} and *min* R_{UZ};
$max\sqrt{R_X{}^2 + R_Y{}^2 + R_Z{}^2}$ and $max\sqrt{R_X{}^2 + R_Y{}^2}$;
$max\sqrt{R_{UX}{}^2 + R_{UY}{}^2 + R_{UZ}{}^2}$ and $max\sqrt{R_{UX}{}^2 + R_{UY}{}^2}$,

where:
R_X, R_Y, R_Z — loads along the axes X, Y, Z of the global coordinate system;
R_{UX}, R_{UY}, R_{UZ} — loads about the axes X, Y, Z of the global coordinate system.

18.7. Data Preparation

The operation of preparing data is invoked from the **Project tree** in the **Special Initial Data** section, the **Design Combinations of Forces** subsection. The data are the same for all the above types of design combinations.

The **Design Combinations of Forces and Displacements** dialog box (Fig. 18.7-1) includes the following controls:

- an icon indicating the code which is used to perform the calculation of combinations (place the mouse over this icon and a pop-up window will appear with the name of the code);
- a table with characteristics of loadings;
- the **Parameters** group of buttons which enable to invoke dialog boxes for specifying the characteristics of the groups of elements;
- the **Relations between loadings** group of buttons which enable to invoke dialog boxes with the description of the relations between loadings;
- the **Activate/Deactivate loading** button for blocking the loadings (temporary exclusion from the list of loadings), for example, it is recommended to use this option for thermal loadings in those cases when the arrangement of expansion joints corresponds to the requirements of the codes, or to exclude the trial loadings from the design combinations;
- a drop-down list for specifying the step (angle) of the analysis of areas in plates;
- control buttons of the dialog box.

When the data is input into the table with the characteristics of loadings for the first time, the number of lines in the table will correspond to the number of loadings in the problem plus the number of combinations of loadings. If when specifying the combinations of loadings the **Do not take into account combinations in DCF** checkbox is checked (see Chapter 17), only the lines of the loadings will be left.

The table includes the following columns:

Active loading — checkboxes of this column enable to exclude certain loadings from the analysis when generating design combinations of forces and displacements. All checkboxes are checked by default, i.e., all loadings are involved in the analysis.

Active loading in DCD — these checkboxes work in the same way as those from the previous column, but they exclude loadings from the analysis only at the generation of design combinations of displacements;

Name — this column is used to specify the name of the loading or the formula of the combination;

Type of loading — a list of types of loadings provided by the codes: permanent, long-term, short-term, crane, special or inactive. If a loading is inactive, it is not involved in the generation of design combinations;

Type of load — a type of load in selected from the list in this column in accordance with the type selected in the previous column. Thus, for example, the following list appears for short-term loads (when the calculations are performed according to SNiP 2.01.07-85*):

Loads from the equipment in the nonstationary mode
Weight of people and repair materials in the service areas
Full loads on floors of residential, public and industrial buildings
Full loads from the mobile handling equipment
Full snow loads in the III snow region
Full snow loads in the IV snow region

Full snow loads in the V and VI snow regions
Full snow loads in other snow regions
Full thermal climatic actions
Wind loads
Ice loads
Repeated
Loads from railway mobile equipment
Loads from highway mobile equipment
Other

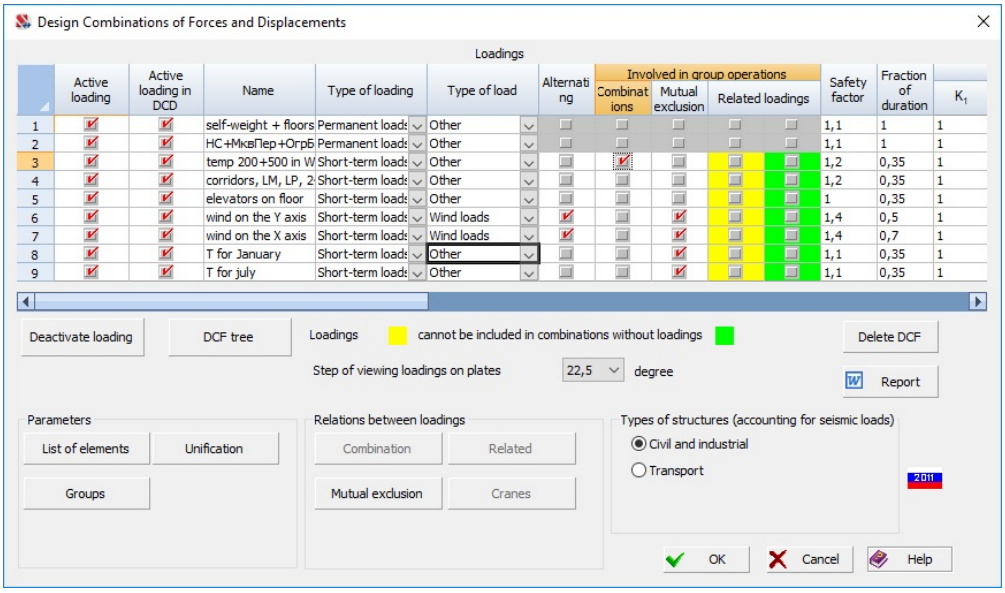


Figure 18.7-1. *The* **Design Combinations of Forces and Displacements** *dialog box*

After selecting the respective type of load for the given loading, the values of the safety factor and the fraction of duration, which can be corrected, will appear in the **Safety factor** and **Fraction of duration** columns. If the values of these coefficients are not defined by the codes, they have to be specified. If EN 1990 is selected for calculating load combinations, these coefficients are not specified.

The **Alternating** column — the property of load (usually dynamic) is specified by checking the checkbox in the respective column of the table.

The group of columns united under the common name **Involved in group operations** includes the following columns **Combinations**, **Mutual exclusions** and **Related loadings**.

The **Combinations** column — checkboxes of this column are used to specify loadings and combinations which in the process of **generation** of combinations can serve as:

a) related;

b) independent, but acting as one load at the simultaneous action (Sec. 1.13 of SNiP 2.01.07-85*).

The checked checkbox in the line of the loading means only that the loading is involved in the operation of combining. Specific "partners" are specified in the table of the **Simultaneous Loadings** dialog box (Fig. 18.7-2). Checkboxes of this dialog box work in the following way: loadings belonging to type *a*) are indicated by the first click ("tick"), and those belonging to type *b)* — by the second click ("cross").

The **Mutual exclusions** column — checkboxes of this column are used to specify loadings which can not be **included** in one combination. Similarly to the previous column the checked checkboxes of this column indicate only the fact that a loading is involved in the operations of mutual exclusion, and the groups of such loadings are specified in the dialog box shown in Fig. 18.7-3. The **All** checkbox enables to check all checkboxes in the table by one click.

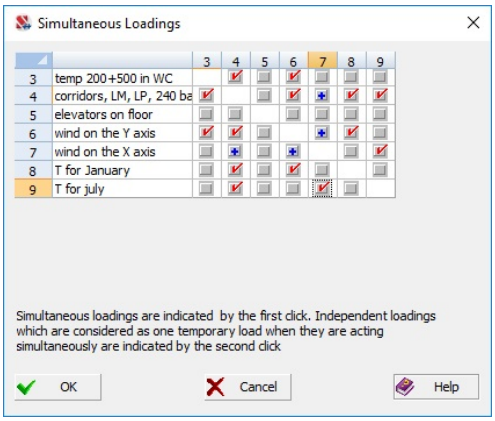


Figure 18.7-2. *The* **Simultaneous Loadings** *dialog box*

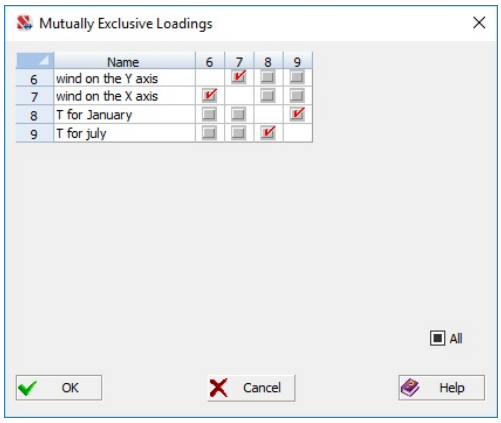
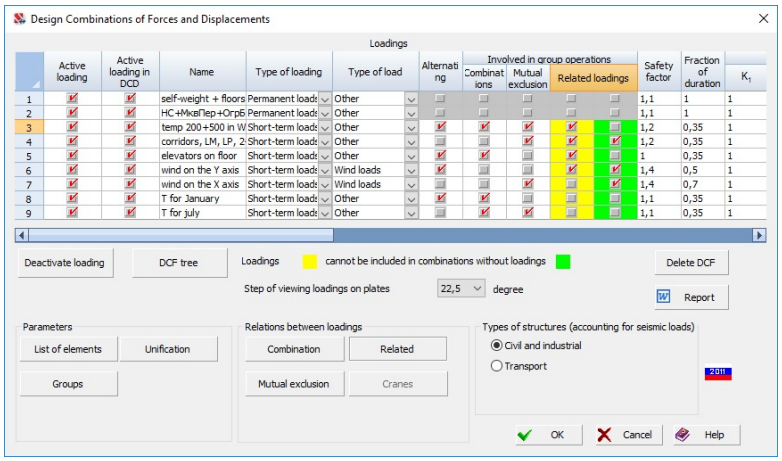
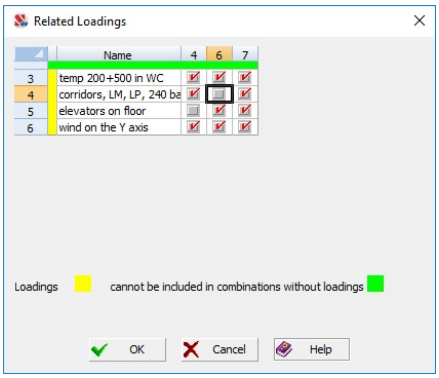


Figure 18.7-3. *The* **Mutually Exclusive Loadings** *dialog box*

In accordance with the requirements of SNiP special loadings (seismic actions belong to them as well) are mutually exclusive. This rule is implemented in the software automatically. However since seismic actions can be specified as three components in the orthogonal directions you can describe them as simultaneous for the same type of seismic dynamic action and thus implement the requirements of SNiP (see, for example, Sec. 5.4 of SP 14.13330.2014) on the simultaneous consideration of the vertical and horizontal seismic loads. The summation of these loadings is performed according to the rules of the respective codes.

The **Related loadings** column — this column is used to describe related loadings, i.e. when some loadings (let's call them of type "s") can not be included in a combination without other loadings (of type "m"), indicated as related. For example, loadings including loads from the seismic action can not be included in a combination without loadings the loads of which were used to specify masses (dead weight, stationary equipment, etc.). Loadings of type "m" can be included in combinations independently. The column is divided into two columns highlighted, respectively, in yellow and in green. Checkboxes of the "yellow" column are used to indicate the loadings of type "s", and those of the "green" are used for loadings of type "m". Similarly to the previous columns of this group checked checkboxes indicate that the loadings are involved in the operations with related loadings. A loading can not be simultaneously selected both in yellow and in green columns, i.e. it must have a certain status. (Fig. 18.7-4).





a) *b*)

Figure 18.7-4. *The* **Design Combinations of Forces and Displacements** *dialog box* (*a*) and its corresponding dialog box **Related Loadings** (*b*)

Relations between loadings are specified in the **Related Loadings** dialog box. Loadings of type "s" are specified in lines (they are highlighted in yellow), and loadings of type "m" are specified in columns

(they are highlighted in green). If in some operations a loading must belong to type “s”, and in others — to type “m”, it has to be duplicated (making the main loading and the duplicate mutually exclusive).

The **Safety factor for load** column — the value of this factor (γ_f) is automatically set in the respective column depending on the type of loading and the type of load and, if necessary, it can be changed by the user. In those cases when this value is not clearly defined by the codes, a question mark appears in the column, which must be replaced by a specific number. This factor is used for the transition from design to characteristic values of loads in the calculation of elements of reinforced concrete structures for the serviceability limit state. It is obvious that a factor with the same value should have been used in the loading when specifying design loads. Moreover, one loading should not contain loads with different values of the factor, for example, the weight of elements of reinforced concrete structures (γ_f = 1,1) can not be included in the loading with the weigh of the leveling courses of floors (γ_f = 1,3).

When the **Design Combinations of Forces and Displacements** dialog box is opened for the first time, this column will contain values of the factors specified when creating loadings. When the user changes the value of the factor in this dialog box, it will be automatically changed in the **Save Loading** dialog box as well.

The **Fraction of duration** column — it is used to specify a factor for sustained load for temporary and short-term loads. The value of the factor is set automatically depending on the type of loading and the type of load and can be changed by the user. When the dialog box is opened for the first time, this column will contain a value of the factor specified when creating the loading. When the user changes the value of the factor, the factor in the loading will be changed as well.

Additional factors for groups of DCF — the column with the value of the factor for sustained load is followed by 14 columns where you have to specify additional factors. The values of the forces/stresses are multiplied by these factors. In this case depending on the number of nonzero additional factors the respective number of independent combinations of forces will be generated. Only the first column of the factors is filled with the value of one by default, since the coefficients of the combination are generated automatically in accordance with the codes. Using the mechanism of **groups of DCF** (see below), these coefficients can be used, for example, to find DCF separately for steel and reinforced concrete elements with different safety requirements.

In those cases when the design combinations of forces have to be calculated not for all elements of the model, click the **List of elements** button and the **List of Elements** dialog box will appear (Fig. 18.7-5).

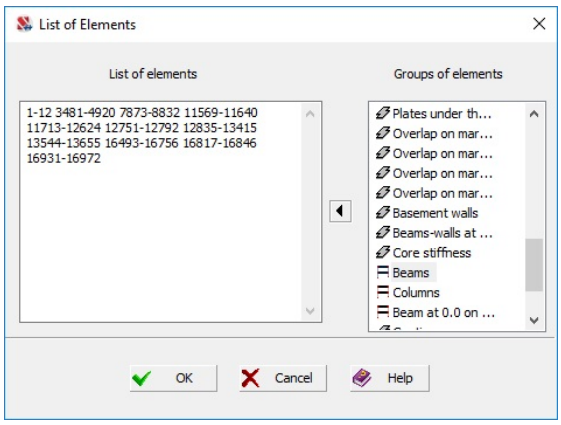


Figure 18.7-5. *The* **List of Elements** *dialog box*

Elements can be created with the help of groups of elements. Groups specified for the selection of reinforcement and other groups of elements are available. If the list is entered without using the groups or adjusted after creating the groups, it is prepared according to the following rules:

- numbers of elements are separated by spaces;
- in the case of consecutively numbered elements the first and the last elements of the series are written with a hyphen (for example: 1-23 34 45 76 87-125).

Peculiarities of Specifying Combinations for EN 1990

If EN 1990 is selected for calculating load combinations, the classification of loadings (type of loading and type of load) corresponding to the given design code is used. Moreover, in this case groups of DCF are not implemented and the following coefficients have to be specified for each loading γ_{Sup}, γ_{Inf}, γ_Q, ψ_0, ψ_1, ψ_2 instead of the coefficients K_1,K_2,...

The **Design Combinations of Forces and Displacements** dialog box contains a button clicking which will add the **EN 1990 Settings** tab (Fig. 18.7-6). This tab enables to select the combination factors in compliance with EN 1990 and with National Annexes to Eurocode, which can be selected from the respective drop-down list.

If the values of the coefficients or the calculation parameters are specified by the user, then you have to select the **Other** item (Other) in the drop-down list. Thus you can use the application for custom, nonstandard situations, and also when any modifications are made in NAD.

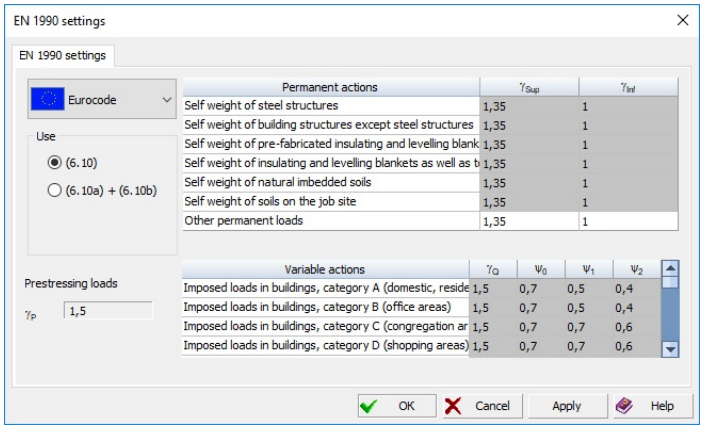
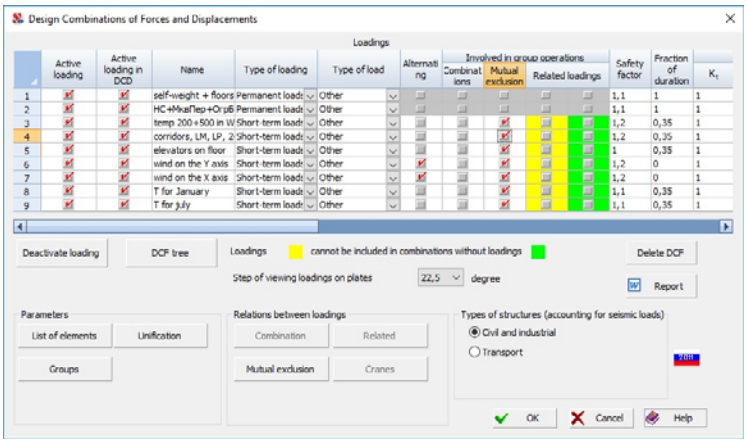


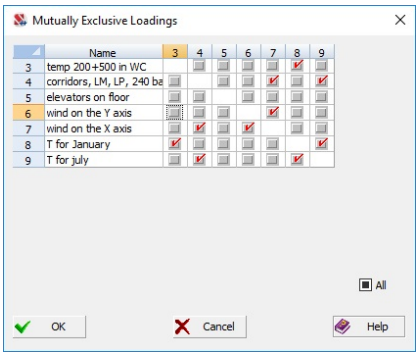
Figure 18.7-6. *The* **EN 1990 Settings** *tab*

Peculiarities of Specifying Combinations for Variations of Models, Erection, and Nonlinear Problems

When calculating variations of models mutual exclusions between loadings which correspond to different models of the package are set automatically. This remark also applies to the loadings of various erection stages. If it is specified that the mutual exclusions in the variations of models will be set automatically, then when generating the data on mutual exclusions of loadings for such problems the “matrix” of loadings will have a “block” structure in the respective dialog box (Fig. 18.7-7).



a)



b)

Figure 18.7-7. *The* **Design Combinations of Forces and Displacements** *dialog box* (*a*) in the **Variation of models** mode and its corresponding dialog box **Mutually Exclusive Loadings** (*b*)

If you want to refuse the automatic mutual exclusion in the calculation of variations of models, it should be done very carefully, because (in the general case) the superposition principle, which is the basis for the calculation of design combinations, is not applicable.

The calculation of combinations of forces is also provided for nonlinear problems. It is necessary so that the results of the nonlinear analysis could be used to select/check steel and/or reinforced concrete structures. The reason is that when calculating according to the codes it is necessary to know, in particular, the characteristic and design values of forces, the long-term part, … The function for preparing the respective data is invoked from the **Project tree** in the **Special Initial Data** section, the **Design Combinations of Forces at the Modeling of Nonlinear Loads** subsection. The dialog box is similar to that described above for the case of standard combinations. The difference is that all loadings are mutually exclusive and you can not specify combinations of loadings and related loadings.

When dealing with nonlinear problems, you should keep in mind that the software converts the characteristic values into the design ones (and vice versa), and determines the sustained loads by multiplying or dividing the result of the nonlinear analysis by a safety factor for load and a factor for sustained load. This approach should be considered as approximate when applied to nonlinear problems, since it implements the requirements of the design codes formulated for linear problems.

The errors of such an approximation are usually small, but it is recommended to refine the solution in critical cases. This can be done by outputting intermediate results corresponding to the sustained load and the characteristic load value when performing the nonlinear analysis for the design load value.

Allowing for the Pulsation Component of the Wind Load

If the user has decided to consider the static and full (statics and dynamics) wind loadings separately, the possibility of their co-occurrence in the DCF will be excluded automatically. If it is necessary to consider only the full dynamic loading, the static loading should be made *inactive*.

Crane Loads

Taking into account the peculiarities of the specification of crane loads, they are considered as a separate type of loadings and they have their "own" list of load types, which includes:

- reduced vertical loads from overhead travelling and underslung cranes;
- full vertical loads from overhead travelling and underslung cranes;
- braking along a crane runway;
- braking “on the left”, “load is on the left”;
- braking “on the left”, “load is on the right”;
- braking “on the right”, “load is on the right”;
- braking “on the right”, “load is on the left”;
- crane load at the seismic action.

Additional information required for the generation of design combinations taking into account the crane loads is specified in the **Crane Loads** dialog box (Fig. 18.7-8), where the respective crane number is specified for each line together with its operation mode.

The maximum number of cranes and brake loadings simultaneously taken into account which can be included in a combination are specified in the dialog box.

Having information on the types of the crane load and knowing the number of the crane a load belongs to, the software can take into account the following logical peculiarities of interaction:

- all vertical loads belonging to the same crane are mutually exclusive (they refer to the different positions of the crane bridge and the crane trolley);
- limitations on the number of cranes simultaneously taken into account and on the coefficient of their combinations are performed in accordance with the specified code (for example, according to Sec.4.14 of SNiP 2.01.07-85* or Sec.9.16 of SP 20.13330.2011);
- horizontal crane load can not appear without the vertical pressure;
- brake loads of different types are mutually exclusive;
- Crane load at the seismic action is a special loading that conforms to the rules of the codes “Construction in seismic regions”, it can not act together with the brake forces. All other crane loads are mutually exclusive with seismic loads;

- when calculating the number of simultaneously acting temporary loads all components of a crane load of the same crane are considered as one temporary load (see, for example, Sec. 1.13 of SNiP 2.01.07-85* or Sec. 6.6 of SP 20.13330.2011).

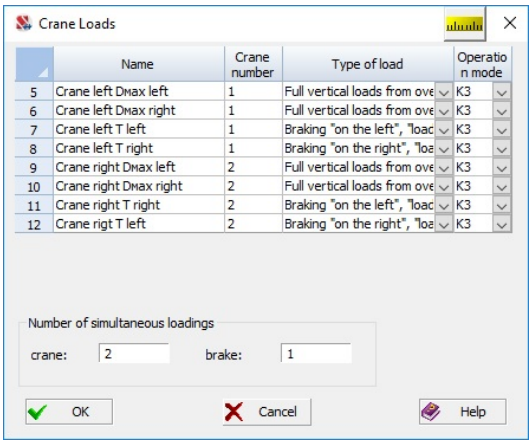


Figure 18.7-8. *The* **Crane Loads** *dialog box*

Any number of loadings of one type can be specified for one crane. When specifying the information on brake loads it is necessary to check the references to the virtual numbers of cranes to ensure that position of the load is specified correctly.

For crane loads all mutual exclusions and related loadings for a single crane are set automatically, and it is not necessary to specify them. In order to specify the interactions of a crane with other loadings of the problem (e.g., cranes) you just have to indicate a relationship with any loading that describes its action.

Specifying the crane number is also important because it enables to distinguish between two fundamentally important cases:

a) different cranes actually work on the structure (different cranes);

b) there are different positions of the location of the crane or its crane trolley (virtually different cranes).

In the first case, different cranes can operate simultaneously, although there may be restrictions on their relative position, which are defined by the specified terms of mutual exclusion; in the second case, all variants of loads from virtually different cranes are mutually exclusive, which should be explicitly specified. Under the conditions of mutual exclusion you just have to specify **any pair** of load types, and all other load types will automatically be mutually exclusive.

Unification

When determining the design combinations of loads the software provides the possibility of unification within the specified lists (of groups) of elements. This means that it is suggested to neglect the differences between the extreme values of stress in various elements and in various sections of these elements, and you should find the worst value for all unified elements and their sections. The following three types of unification are implemented:

- sections are unified in a group of elements in such a way so that all elements of the group have the same section;
- the unification for the respective sections is performed in the group of elements, i.e. elements of the group will have identical first, second, third, and so on sections;
- symmetrical sections are unified in the group of elements, i.e. all elements of the group will have identical first and last sections, second and next to last sections and so on.

It is obvious that only the first type of unification (if another type is selected, the software will use the first type of unification for plates) can be used for plate and shell elements, and also for elements included in a single physical bar (structural member). Only the elements with the same cross-sectional shape and orientation of the local axes can be included in one group.

Despite the fact that the check of sections of bar elements included in one unification group is not performed during the generation of DCF, it is recommended to create groups in such a way so that all elements of one group have the same cross-sectional shape.

Data for the unification of DCF are specified in the **Unification** dialog box (Fig. 18.7-9). The following procedure is recommended for specifying the new data:

- click the **New list (of elements)** button;
- select the type of unification from the respective list;
- select the number of the group of unification in the **Groups of unification** list;
- specify the list of elements or load the groups created earlier;
- click the **Save** button**.**

It is necessary to perform the following actions to modify the list created earlier:

- select the number of the modified group in the **Groups of unification** list;
- change the characteristics of the list;
- save the changes by clicking the **Save** button.

In order to delete the list created earlier, it is necessary to select it and click the **Delete** button.

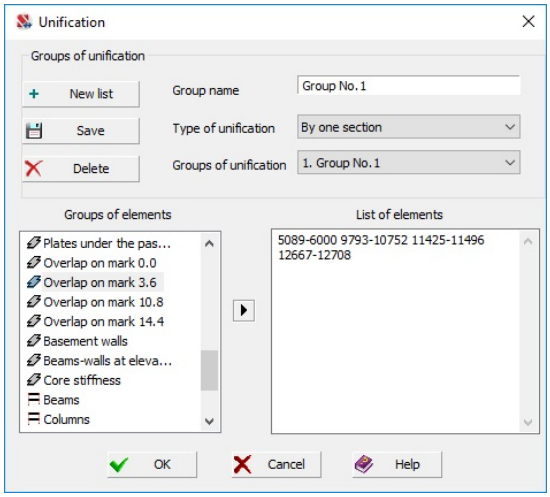


Figure 18.7-9. *The* **Unification** *dialog box*

Unification is performed only during the calculation of DCF and the data on the groups of unification is neglected during the calculation of other types of design combinations.

Groups for DCF

In practice, it might be sometimes necessary to apply different combination factors for different elements of the model which will be used for a single loading in a combination. Such elements are combined into groups Fig. 18.7-10 (not to be confused with the groups of elements created during the generation of the design model and the analysis of results, which are specified in the **Groups of DCF** dialog box).

The procedure for working in this dialog box is basically the same as that described above for the **Unification** dialog box — click the **New list** button, specify the numbers of elements and the numbers of columns the DCF coefficients are selected from, and click the **Save** button.

The numbers of columns from which the DCF coefficients should be taken for the current group of elements are indicated in the table in the upper part of the dialog box. It should be noted that these coefficients are specified in the columns of the table in the **Design Combinations of Forces** dialog box (see Fig. 18.7-1).

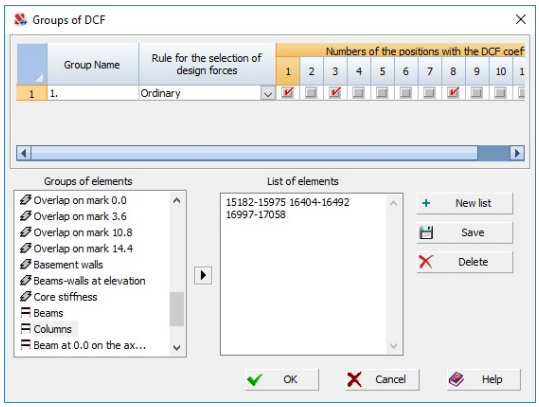


Figure 18.7-10. *The* **Groups of DCF** *dialog box*

When working with groups (Fig. 18.7-10) they have an additional characteristic: "The rule for selecting design forces", where a type of building structures is specified according to Table P.4.1 of "Design codes for earthquake-resistant nuclear plants (NP-031-01)", which establish special rules for combining seismic loads acting in different directions. If you specify three seismic loadings in the directions X, Y and Z as simultaneous, the postprocessor will automatically go through all the combinations given in the code.

Thus, for example, the following combinations will be automatically generated for the structural members of buildings (structures) of the seismic category I:

$$\pm (N_x \pm 0.4\ N_y \pm 0.4\ N_z),$$
$$\pm (N_y \pm 0.4\ N_x \pm 0.4\ N_z),$$
$$\pm (N_z \pm 0.4\ N_x \pm 0.4\ N_y),$$
$$\pm \sqrt{\left(N_x^2 + N_y^2 + N_z^2\right)},$$

where N_x, N_y and N_z are criteria for seismic loadings for the respective directions. In the case of the fourth combination (square root), 10000 will be added to the criteria numbers.

For seismic analysis according to SP RK 2.03-3-2017 in the above formulas, the coefficient is used 0.3 instead of 0.4.

In case of analysis according to SP RK 2.03-3-2017 in the above formulas factor 0.3 is used instead of 0.4.

If EN 1990 is selected for calculating load combinations, groups of DCF are not implemented.

The **DCF tree** button invokes the dialog box (see Fig. 18.7-11), where the logical relationship between loadings is given graphically (in the form of a tree), and which enables to perform an additional visual control of the specified data.

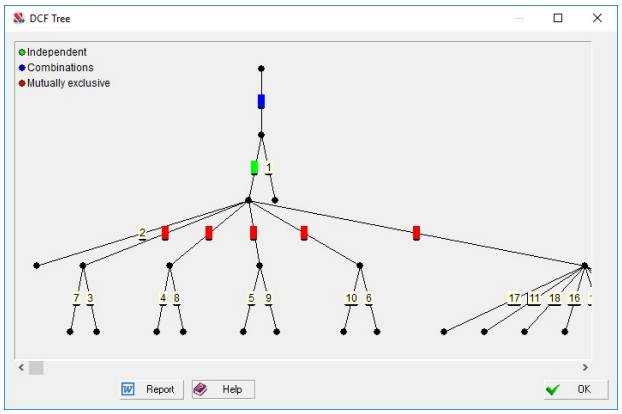


Figure 18.7-11. *The* **DCF Tree** *dialog box*

All the information entered in the mode of specifying the design combinations of forces can be deleted by clicking the **Delete DCF** button (it corresponds to the rejection of the DCF calculation).

18.8. Design Combinations of Displacements

SCAD enables to calculate design combinations of displacements (DCD). Calculation of DCD is performed automatically during the general analysis.

Design codes regulate the limitations of the characteristic values of displacements, therefore only the characteristic and characteristic long-term combinations of displacements are calculated for each node.

18.9. Design Combinations of Reactions in Constraints

If the **Calculate reactions in constraints** option was selected during the calculation, the design combinations of reactions in constraints will be generated as well during the selection of the design combinations of forces.

18.10. Design Combinations of Deflections

The design combinations of deflections are calculated for bar elements simultaneously with DCF. Like DCF combinations of deflections will be generated for each section of the element.

18.11. Design Combinations for the Progressive Collapse

A combination specified by the user is used as an unfavorable combination of loadings in the analysis for the progressive collapse. In fact the "selection" of design combinations in this case comes down to the calculation of design and design long-term combinations of forces for the above combination.

18.12. Analysis of Results (DCF and DCD Postprocessor)

DCF and DCD postprocessor enables to perform the analysis of results of the calculation of DCF and DCD. The control of this postprocessor is performed with the help of a special toolbar (Fig. 18.12-1).

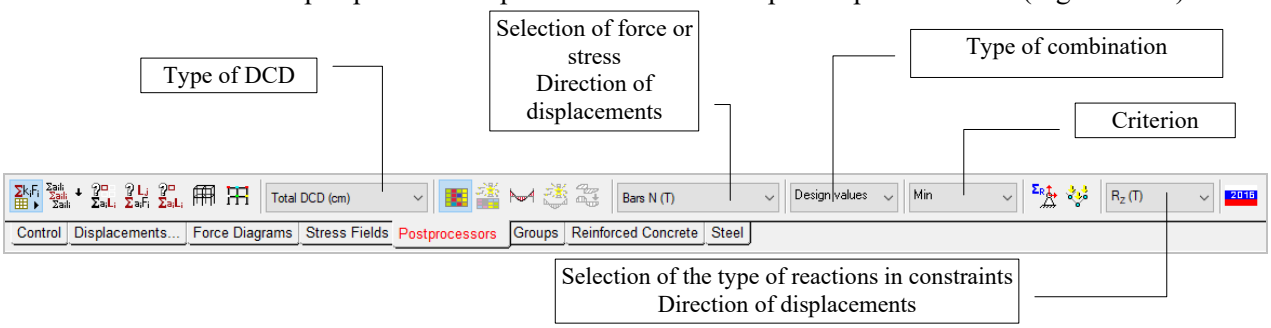


Figure 18.12-1. *The* **DCF and DCD** *toolbar*

The toolbar includes the following operations:

— create combinations of loadings for DCF;

— show elements the DCF of which include the specified loadings;

— show combinations which include the specified loading;

— show elements the DCF of which include the specified combinations;

— color indication of design combinations of displacements;

— color indication of design combinations of forces;

— color indication of design combinations of forces (progressive collapse);

— design combinations of reactions in constraints (color scale);

— design combinations of reactions in constraints;

— envelopes of the diagram of design combinations of forces;

— envelopes of the diagram of design combinations of forces (progressive collapse);

— change the display axis of diagrams N and M_k;

— display design model.

Create Combinations of Loadings for DCF

This operation enables to obtain all combinations of loadings created during the calculation of DCF. Clicking the button invokes the dialog box (Fig. 18.12-2) where you can narrow down your choices of analyzed combinations (for example, select only the design ones). If there are selected elements in the model, you can check the respective checkbox and the software will analyze only the selected elements. You can also click the **Elements** button and specify the list of elements you are interested in or select the elements from the respective groups (Fig. 18.12-3).

A text file with the formulas of combinations will be output after clicking the **OK** button (Fig. 18.12-4). The presence or absence of the numbers of elements in this file depends on whether the **Print the numbers of elements** checkbox is checked.

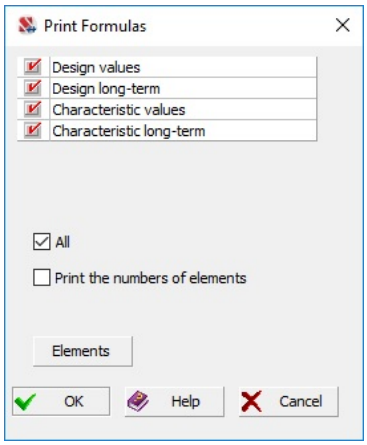


Figure 18.12-2. *The* **Print Formulas** *dialog box*

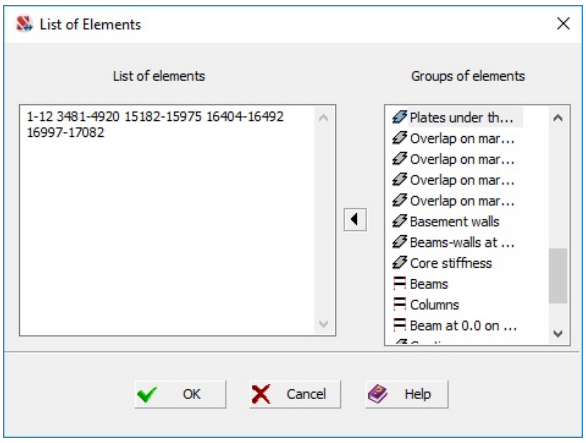


Figure 18.12-3. *The* **List of Elements** *dialog box*

```
1. L1+L2+L7 : 198-230 357 358
2. L1+L2+L3+0.89999*L4-L7 : 198-230
3. L1+L2+L6 : 198 203-205 210 211 214 215 218 227-230 357 358
4. L1+L2+L3+L4-0.89999*L7 : 198-205 207-210 212-217 219-223 225 226 229 230 357
   358
5. L1+L2+L7+0.89999*L8-0.89999*L10 : 198 202-205 212-215 223 230
6. L1+L2+L3+0.89999*L4-L7+0.69999*L9-0.69999*L11 : 198 203-205 212-215 226
7. L1+L2+L7+0.89999*L16 : 198 200 201 203 207 208 210-213 216 224-226 228 358
8. L1+L2+L3+0.89999*L4-L7+0.69999*L8+0.69999*L12 : 198 20
```

Figure 18.12-4. *Text file with the formulas of combinations*

Using the arrow located next to the button you can drop down the menu and select two other possible courses of action: create a report file in the RTF-format, which is compatible with word processors such as MS Word, or create a binary file (with the **rsu2comb** extension), which can be subsequently imported when specifying the initial data for combinations of loadings (see Chapter 17).

Show Elements the DCF of which Include the Specified Loadings

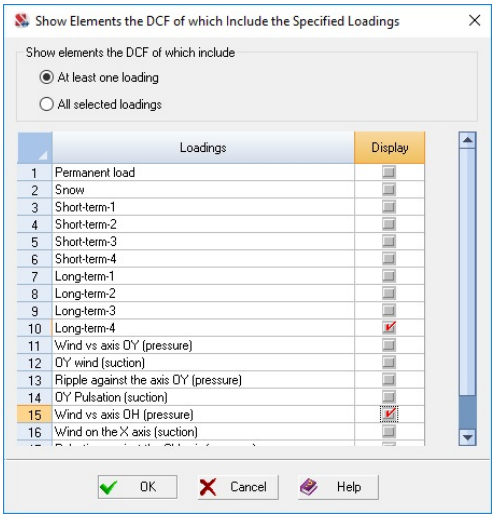


Figure 18.12-5. *Selection of loadings*

This operation invokes the dialog box (Fig. 18.12-5), which lists all loadings (except for those which have *inactive* status in the initial data of DCF). After indicating the respective loadings and clicking the **OK** button, all elements the DCF of which have selected loadings become selected (and are highlighted in red). It is obvious that if you select lines with the loadings of "permanent" and "long-term" type, all elements will be selected in the model.

Show Combinations which Include the Specified Loading

This operation invokes the dialog box (Fig. 18.12-6), which lists all loadings (except for those which have *inactive* status in the initial data of DCF). After selecting the necessary loading we can obtain a set of formulas of the combinations which include it (this loading) in the list on the right. You can select only the combinations you are interested in (all, only the design ones, ...) from the drop-down list in the upper part of the dialog box.

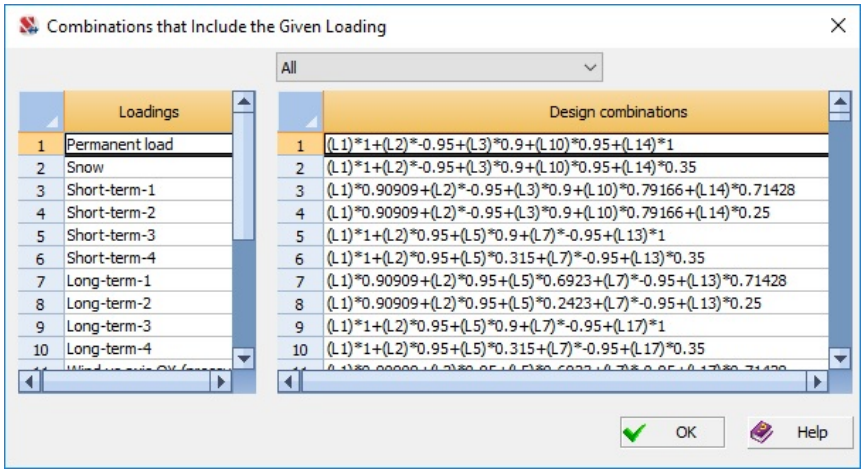


Figure 18.12-6. *The* **Combinations that Include the Given Loading** *dialog box*

Show Elements the DCF of which Include the Specified Combinations

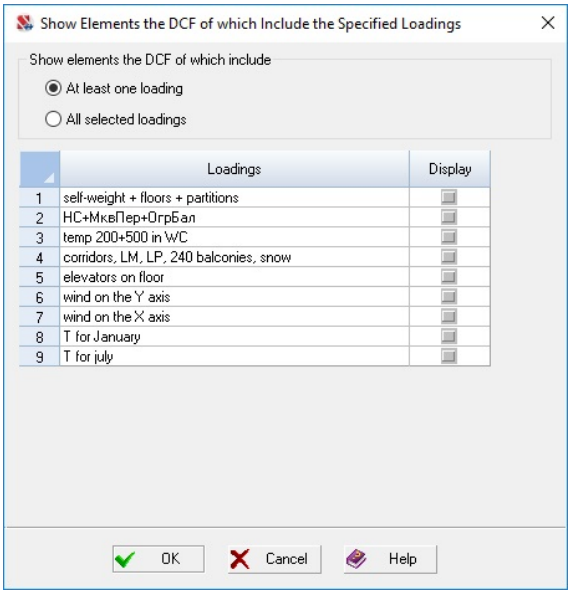


Figure 18.12-7. *Selection of combinations*

This operation invokes the dialog box (Fig. 18.12-7), which lists all combinations of loadings (except for those which have *inactive* status in the initial data of DCF). After indicating the respective combinations and clicking the **OK** button, all elements the DCF of which have selected combinations become selected (and are highlighted in red).

Color Indication of Design Combinations of Forces

You can obtain the color indication of the values of DCF during the graphical analysis of the results in the postprocessor. In order to do this it is necessary to perform the following actions:

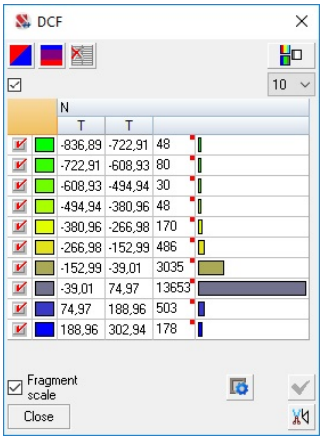


Figure 18.12-8. *Color scale of DCF*

- select the necessary type in the **Select type of force/stress** list in the toolbar;
- select the type of the analyzed combination (design, characteristic, ...) in the **Type of combination** list;
- select the criterion (Min/Max/Max Abs) in the respective list.

After specifying the above parameters the color scales will be generated (Fig. 18.12-8) by selecting minimum, maximum or maximum of absolute values of the force factor from all combinations for each element (in the case of bars it will be done for all sections).

Color Indication of Design Combinations of Displacements

Similarly to DCF you can obtain the color indication for the design combinations of displacements.

Figure 18.12-9. *Color scale of DCD*

In order to do this you have to select the necessary criterion (for example, the maximum total displacement) in the **Type of design combinations of displacements** list in the toolbar and invoke the operation.

Checkboxes in the nodes of the design model and the color scale are used for the color indication of DCD (Fig. 18.12-9).

Color Indication of Design Combinations of Forces (Progressive Collapse)

Displaying the results of the calculation of DCF during the analysis for the progressive collapse enables to obtain a color indication of the design combinations according to the same rules as for the standard combinations (see above).

Envelopes of the Diagram of Design Combinations of Forces

This operation enables to plot the envelopes of the diagrams based on the design combinations of forces. The value of the specified force factor is selected from the combinations of the given type for each section of bar elements based on the criterion of minimum or maximum.

In order to perform this operation select the criterion, force factor and the type of combination from the respective lists in the toolbar and invoke the operation by clicking the button .

Envelopes of the Diagram of Design Combinations of Forces (Progressive Collapse)

This operation is similar to the previous one and it is available during the analysis of DCF for the calculations for the progressive collapse.

Design Combinations of Reactions in Constraints

Color indication of design combinations of reactions in constraints can be obtained if, first of all, the **Reactions in constraints** checkbox was checked when performing the calculation of the problem, secondly, the initial data for generating design combinations of forces were prepared and, thirdly, the design combinations of reactions in constraints were calculated.

Reactions are displayed as color markers in the nodes with constraints in the analyzed direction of reactions. It is necessary to perform the following actions to display the reactions:

- select the necessary type in the **Select type of force/stress** list in the toolbar;
- select the type of the analyzed combination (design, characteristic, ...) in the **Type of combination** list;
- select the criterion (Min/Max/Max Abs) in the respective list.

Design combinations of reactions in constraints can be also displayed by arrows — (similarly to the nodal loads).

Information on the Element/Node

You can obtain a tabular presentation of DCF and DCF (progressive collapse) for elements, as well as design combinations of displacements in a node and design combinations of reactions in constraints for each finite element or node using the tools of **Information on the element** and **Information on the node** operations.

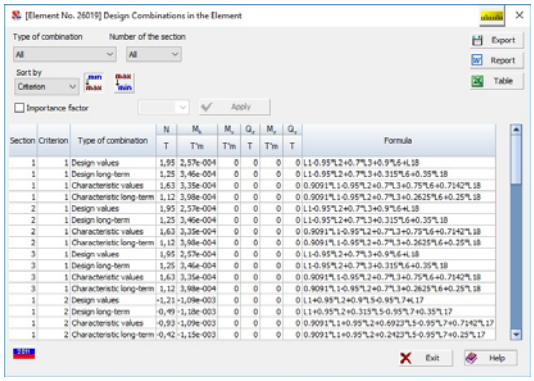


Figure 18.12-10. *The* **Design Combinations in the Element** *dialog box*

The dialog box with the DCF of the element (Fig. 18.12-10) enables to sort the data according to a certain feature, display only the indicated types of combinations (design, characteristic, ...), output the information in the report document or in the spreadsheets. You can select either all sections, or any one section (depending on the specified number of sections for calculating forces) for bar elements in the **Number of the section** list. The **Export** button enables to save the values of design combinations of forces in order to transfer them to the **Resistance of sections** mode of **Kristall**, **ARBAT** or **Decor** in a file with extension **.rsu2**.

If the **Importance factor** checkbox is checked in the **Design Combinations in the Element** dialog box, you can specify the numerical value of this coefficient. In the case when the selected finite element belongs to a structural member or to a group of structural members, the default value for the importance factor will be taken from the data on the respective structural member. In the case when the considered finite element does not belong to any structural member, the default value of the importance factor will be taken as 1.

Depending on the selected safety codes the importance factor will be applied either to design, or to design and characteristic values of forces/stresses.

After clicking the buttons Report or Table the data on DCF will be exported to the Word document (file type *.doc) or to the Excel table (file type *.xls) respectively taking into account the specified (determined) values of the importance factor. This may be needed in the case when the data are generated for example for the “Specification of elements” table of the drawings of the MS stage.

Clicking the button Export exports the data on DCF into the file of type *.rsu2 without taking into account the importance factor, since in those programs where the import of *.rsu2 files is provided this coefficient is specified separately.

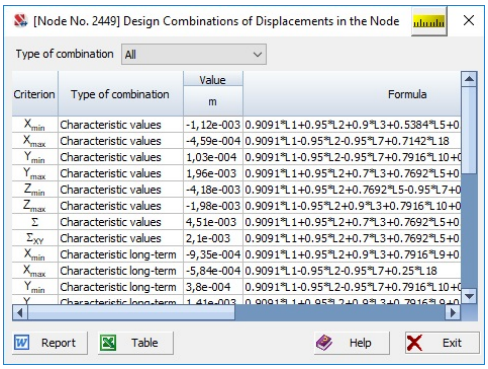


Figure 18.12-11. *The* **Design Combinations of Displacements in the Node** *dialog box*

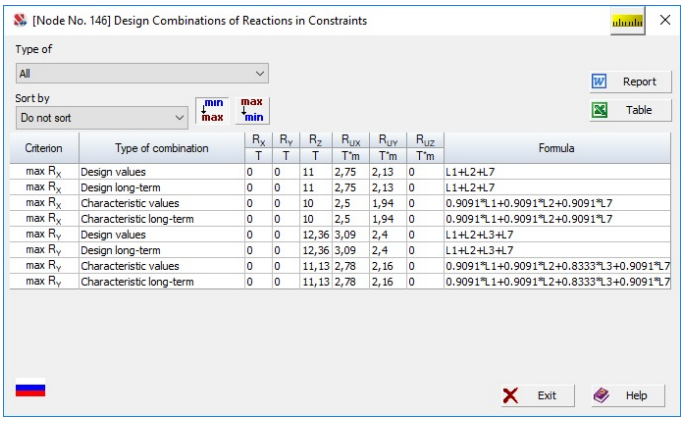


Figure 18.12-12. *The* **Design Combinations of Reactions in Constraints** *dialog box*

Design combinations of displacements (Fig. 18.12-11) and reactions in constraints (Fig. 18.12-12) in the selected node can be also obtained in a tabular form using the DCD and DCR buttons respectively in the **Information on the Node** dialog box (Fig. 18.12-15).

Tables with the results can be generated with the help of the **Generator of tables in the text format** or **Documenter** (see Chapter 13).

19. Reinforcement of Sections of Reinforced Concrete Elements

This mode enables to select reinforcement and check the specified reinforcement in the elements of reinforced concrete structures for the ultimate and serviceability limit states in accordance with the requirements of SNiP 2.03.01-84* "Concrete and Reinforced Concrete Structures ", SNiP 52-01-2003 (SP 52-101-2003) or SP 63.13330.

The analysis is performed for reinforced concrete structures made from heavy-weight, fine-grained and lightweight concrete with the reinforcement steel or the reinforcing wire.

The library of procedures of selecting the reinforcement contains the following modules:

- **Bar 2D** — for the reinforcement of flat bar reinforced concrete elements of rectangular section, T-section, I-section, or ring section for the ultimate and serviceability limit states;
- **Bar 3D** — for the reinforcement of spatial bar reinforced concrete elements of rectangular section, T-section, I-section, or ring section for the ultimate (according to SNiP 2.03.01-84*) and for the ultimate and serviceability limit states (according to SP 52-101-2003 or SP 63.13330);
- **Plate** — for the reinforcement of plate elements for the ultimate and serviceability limit states;
- **Shell** — for the reinforcement of shell elements for the ultimate and serviceability limit states;
- **Deep beam —** for the reinforcement of deep beam elements for the ultimate and serviceability limit states.

The following initial data has to be specified for the work of the postprocessor:

- geometry of the reinforced section ;
- design combinations of forces (DCF);
- information on the class of concrete, class of reinforcement, distance to the center of gravity of reinforcement etc.

The check and selection of reinforcement in bar elements is performed in accordance with the method given in the selected code. Since the procedure for checking the reinforcement in the elements of shells, plates, etc., is not given in the codes (except for SP 63.13330), the method proposed by N.I. Karpenko [3] is used in **SCAD**. Recommendations of SP 63.13330 are also based on this method. The main guidelines of this method are given in Sec. 19.16. It should be noted that they correspond to the general guidelines for calculating planar structures given in Sec. 1.38 of SNiP [6].

Moreover, unlike SP 52-101-03 (more specifically, the Guide to this SP) the condition of formation of oblique cracks is not considered in SP 63.13330. However, the software performs this check if there is a requirement of the absence of cracks. The analysis is implemented in accordance with the recommendations of Sec. 4.28 of the Guide to SP 52-101-03.

Definition of the Main Concepts

Area of the rebar (a_1, a_2, a_3, a_4, aw_x, aw_y) – area of one rebar of the respective type calculated as the geometric area of a circle with the diameter d_i by the formula (cm^2):

$$a_i = \frac{\pi d_i^2}{4}; i = 1,2,3,4;$$

$$aw_{x(y)} = \frac{\pi d_{x(y)}^2}{4}.$$

Reinforcement of Two-dimensional Elements

The concept of the intensity of reinforcement is introduced for plates, shells and deep beams.

Intensity of longitudinal reinforcement (IS_1, IS_2, IS_3, IS_4) is the value which characterizes the area of reinforcement of a certain type (S_1, S_2, S_3, S_4) per unit length of the reinforcement area in the direction perpendicular to the direction of layout of rebars. The intensity is numerically equal to the cross-sectional area of one rebar divided by the spacing of the given type of reinforcement (cm^2/m):

$$IS_i = \frac{a_i}{s}, \quad \begin{cases} s = s_x (i = 3,4), \\ s = s_y (i = 1,2). \end{cases}$$

Intensity (per running meter)[1] **of transverse reinforcement** (***W***$_x$, ***W***$_y$) is the area of transverse reinforcement per unit length of the transverse reinforcement area[2] in the direction of the respective axis, and it is numerically equal to the area of one rebar divided by the spacing of the given type of reinforcement (cm^2/m).

In the result of the reinforcement selection operation implemented in **SCAD** the values of the intensity of reinforcement (cm^2/m) which ensure the satisfaction of requirements are obtained for each type of reinforcement:

$$W_{x(y)} = a_w / s_{x(y)},$$

where s is the spacing of transverse reinforcement, a_w is the area of one rebar of transverse reinforcement.

Intensity of transverse reinforcement (*W*) is the area of transverse reinforcement per unit area, and it is numerically equal to the area of one rebar divided by the products of spacings of the transverse reinforcement in X and Y direction (cm^2/m^2).

$$W = a_w / s_x s_y.$$

Area of transverse reinforcement (*AW*$_x$, *AW*$_y$) is the product of the intensity of reinforcement by the spacing of reinforcement in the orthogonal direction (cm^2):

$$AW_{x(y)} = W_{x(y)} s_{y(x)}.$$

Reinforcement of Bar Elements

Area of longitudinal reinforcement (***AS***$_1$, ***AS***$_2$, ***AS***$_3$, ***AS***$_3$) is the total area of rebars placed along the longitudinal axis of the bar near the certain sides of the element section (cm^2).

$$AS_i = \sum_{k=1}^{M_i} a_k; \quad i = 1, 2, 3, 4,$$

where M_i is the number of rebars of the i-th type.

Area of transverse reinforcement (*AW*$_z$, *AW*$_y$) is the total area (cm^2) of rebars in the section of the element placed parallel to the certain sides of the section (for stirrups — number of intersections of stirrups with the plane passing through the longitudinal axis of the bar).

$$AW_{z(y)} = \sum_{k=1}^{M_{z(y)}} W_{z(y)k}.$$

Intensity of transverse reinforcement (*W*$_z$, *W*$_y$) is the area of transverse reinforcement per unit length of the transverse reinforcement area in the direction parallel to the respective side and numerically equal to the total area of transverse reinforcement $AW_{z(y)}$, lying in the section of the bar parallel to the respective side divided by the spacing of the given type of reinforcement (cm^2/m).

$$W_{z(y)} = AW_{z(y)} / s_{z(y)}.$$

Initial data are specified in the dialog box of the postprocessor. In the result of the work of the postprocessor the areas of "distributed" reinforcement are obtained, as well as the number, cross-sectional area and diameters of rebars. The results can be given in the form of tables and drawings, displayed on the screen or printed.

---

[1] The check and selection of transverse reinforcement in plate elements in SCAD are based on the recommendations of SNiP for the calculation of transverse reinforcement for a bending element with a rectangular cross-section (the program considers a 1 m wide strip with a height equal to the thickness of the plate). Therefore, we use the concept of ***linear*** intensity. When choosing a design on the basis of the results of the selection the spacing of transverse reinforcement should not exceed 1 m in order to provide the correspondence with the approach used for the calculation (especially since a much more strict restriction is given in the design requirements of SNiP). The dimension of type cm^2/mm is used for values IW_x, IW_y everywhere in the program, although it is more accurate to write $\frac{\text{cm}^2}{\text{mm} \times \text{running meter}}$.

[2] The area of transverse reinforcement for Q_x extends along the Y_1 axis, and for Q_y — along the X_1 axis.

19.1. Limitations

The following limitations should be taken into account when using the postprocessor:

- the analysis of members made of cellular, porous, or self-stressing concrete is not implemented;
- the analysis of prestressed reinforced concrete members is not performed;
- the analysis of elements for the limit state for deformations is not performed;
- the set of sections is limited to a rectangle, T-section, I-section, round and ring section;
- the limitation of the diameter of reinforcement for the concrete of low grades is not controlled (the maximum diameter of reinforcement is specified by the user);
- the limit width of flanges of T- and I-sections is not controlled (the design width of flanges is specified by the user in accordance with codes, for example, Sec. 3.16 of SNiP 2.03.01-84*);
- the coefficient γ_{s5} is not taken into account for the high-strength reinforcement of the following classes A-IV, A-V, A-VI, B-II, BP- II, K7, K-19 at the stresses greater than the yield stress (Table 24 of SNiP 2.03.01-84*);
- the analysis for the serviceability limit state does not include the analysis of crack closing;
- the fatigue analysis is not performed;
- SNiP method of strength analysis of oblique sections is focused on the work with a diagram of shear forces, using which it is possible to obtain a "shear force in an oblique section with the length of the projection C on the longitudinal axis of the element determined from all external forces, located on one side of the considered oblique section". It can be easily done in the case of one loading. However, if there are many variants of a loading, we have their envelope which is not a resultant of all external forces. No recommendations are given in the codes for this case. Therefore, the values of shear forces in the design sections are used to increase the reliability;
- the procedure for the calculation of circular and ring cross-sections for the action of a transverse force is not provided in the codes. The formal analysis of the SNiP formulas makes it evident that two values b and h_0 are required. The h_0 value is clearly defined in the codes, but it is not clear what is the width of the section. The calculation in **SCAD** is performed in the following way. When the value $b \times h_0$ is required in the calculation for the action of a transverse force, a value equal to the area of a section with an outer radius R-a is used, where R is the outer radius of the considered cross-section, and a is the distance to the center of gravity of the reinforcement;
- The flexibility limitations are not taken into account during the selection of longitudinal reinforcement;
- The flexibility limitations are not checked during the analysis of the specified reinforcement.

19.2. Selection of Design Codes

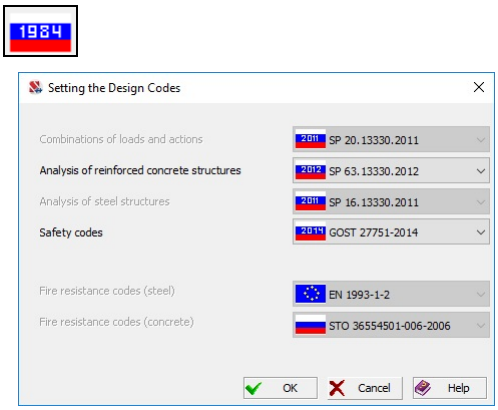


Figure 19.2-1. *The* **Setting the Design Codes** *dialog box*

This operation enables to change the design codes for reinforced concrete structures and safety codes. You should select the necessary codes from the drop-down lists in the dialog box (Fig. 19.2-1) and click the **OK** button. When changing the codes you should remember that the codes according to which the design combinations of forces were selected and the design codes should correspond to each other.

19.3. General Information about the Reinforcement Modules

Bar 2D (Uniaxial Stress State)

It enables to check and select reinforcement in sections of bar reinforced concrete elements for ultimate and serviceability limit states (strength and crack resistance). The module calculates bars of rectangular section, T-section, I-section, or ring section for **bending and eccentric compression (tension) with torsion**. The following force factors can act in a section:

- longitudinal force — N;
- torque — M_k;
- shear force — Q_z;
- bending moment — M_y.

The areas of symmetric and asymmetric longitudinal reinforcement, area and spacing of transverse reinforcement and their corresponding set of rebars are obtained in the result of the work of the module.

The arrangement and identification of asymmetric and symmetric longitudinal reinforcement for different types of sections are given in Fig. 19.3-1, *a–d* and Fig. 19.3-1, *e–h* respectively. It is obvious that symmetric reinforcement can be selected only for sections symmetric with respect to the Y_1 axis.

The arrangement of transverse reinforcement for sections of different types is given in more detail in Section 19.13.

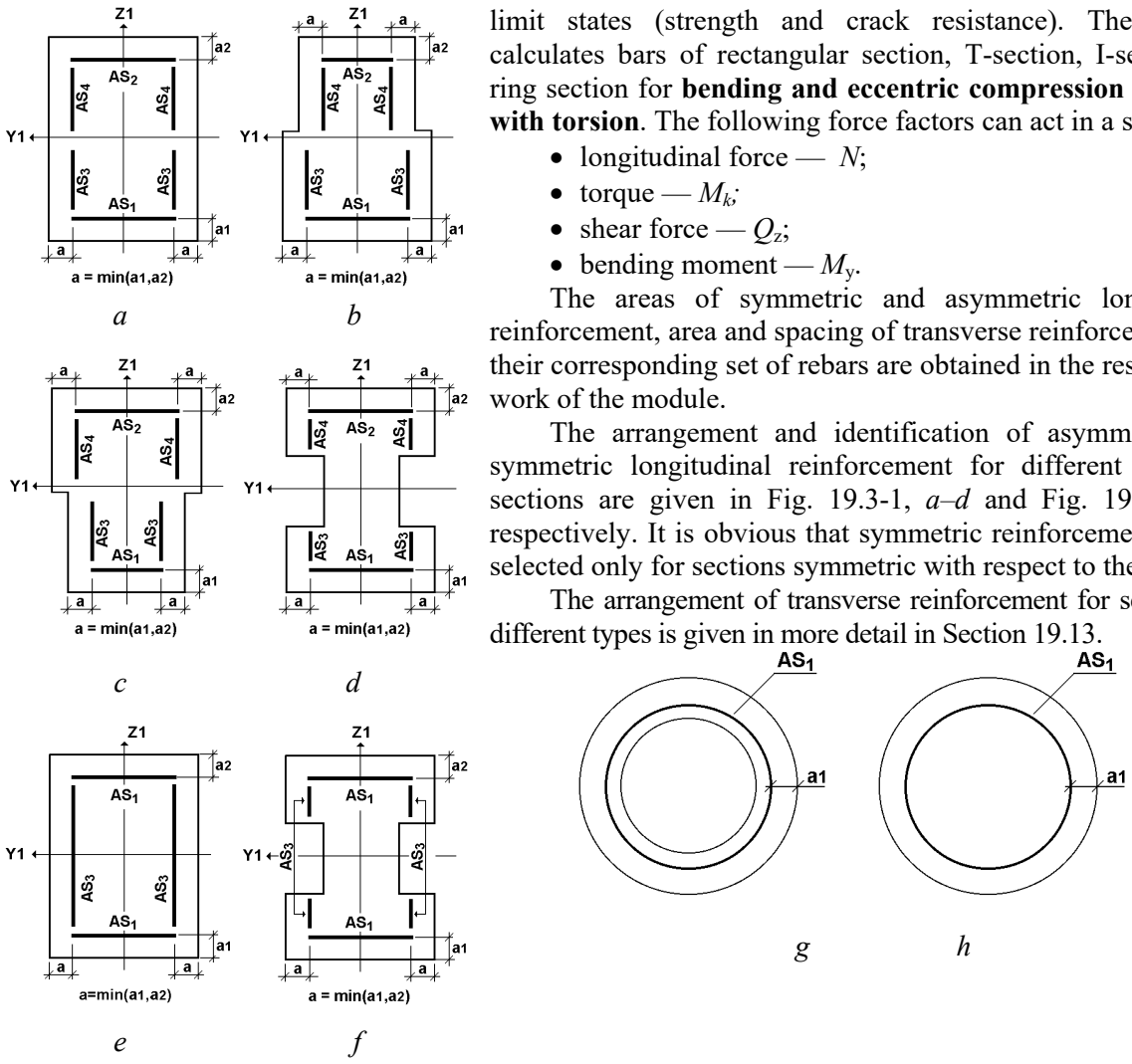


Figure 19.3-1. *Types of sections with asymmetric (a–d) and symmetric (e–h) arrangement of longitudinal reinforcement for elements under bending*

Bar 3D (Biaxial Stress State)

It enables to check and select reinforcement in sections of bar reinforced concrete elements for ultimate limit states (strength) using the recommendations of SNiP 2.03.01-84* and for ultimate and serviceability limit states according to SP 52-101-2003 or SP 63.13330.

The module calculates bars of rectangular section, T-section, I-section, or ring section for **biaxial bending and biaxial eccentric compression (tension) with torsion**. **Spatial** behavior of a bar is considered. The following force factors can act in a section:

- longitudinal force — N;
- torque — M_k;
- shear forces — Q_z, Q_y;
- bending moments — M_y, M_z.

The area of longitudinal reinforcement, area and spacing of transverse reinforcement and their corresponding set of rebars are obtained in the result of the work of the module.

The arrangement and identification of asymmetric and symmetric longitudinal reinforcement for different types of sections are given in Fig. 19.3-2, *a–d* and Fig. 19.3-2, *e–h* respectively. Symmetric reinforcement can be selected only for sections symmetric with respect to the Y_1 axis.

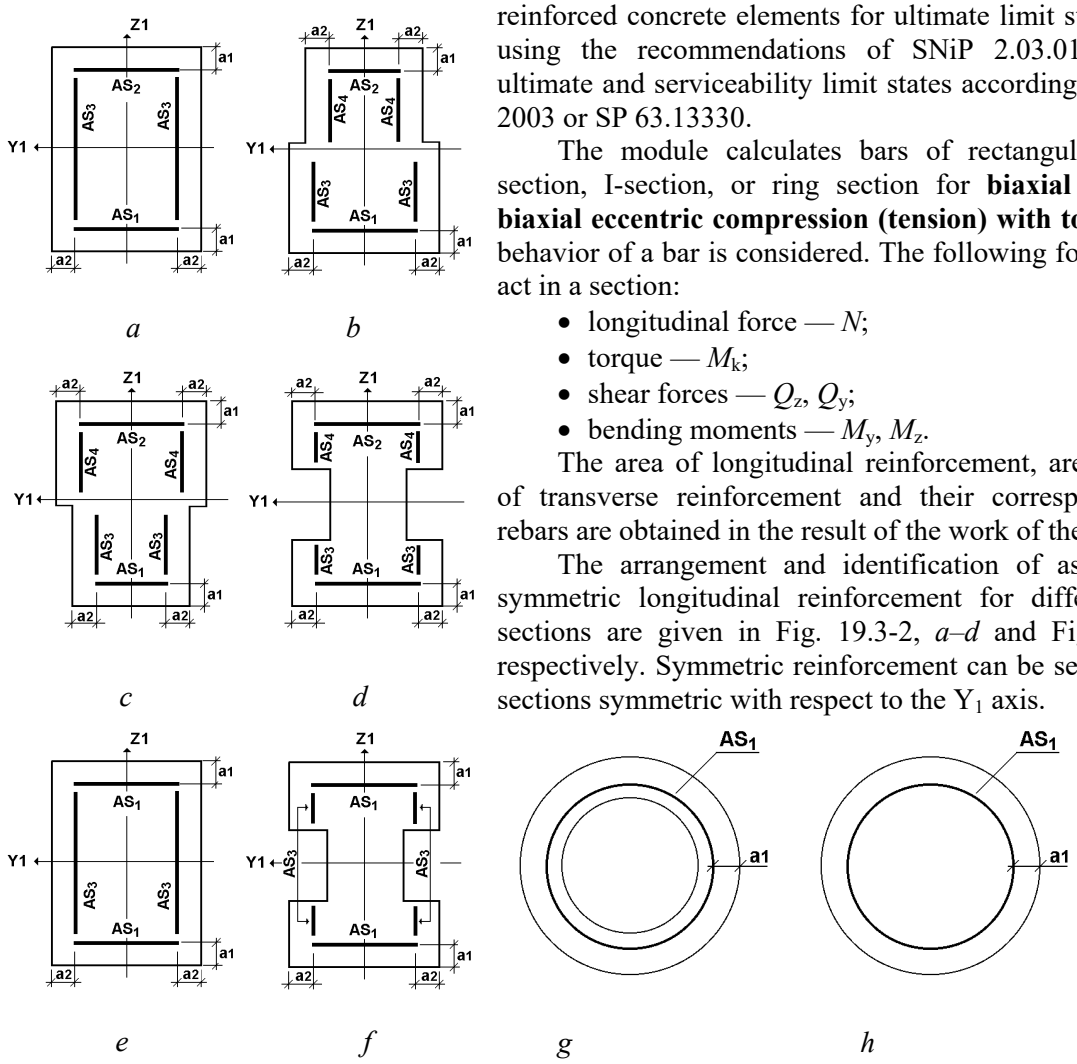


Figure 19.3-2. *Types of sections with asymmetric (a–d) and symmetric (e–h) arrangement of longitudinal reinforcement for elements under compression and bending*

Plate. Shell

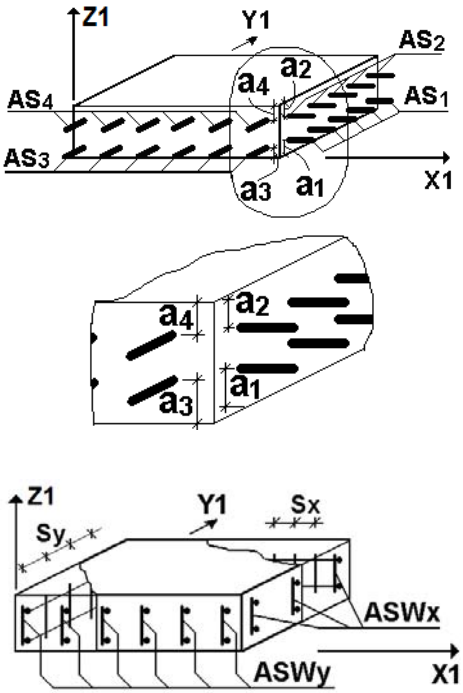


Figure 19.3-3. *Reinforcement of elements of a reinforced concrete shell*

It enables to check and select reinforcement of reinforced concrete plates and shells for ultimate and serviceability limit states (strength and crack resistance).

Plate

The selection is performed taking into account the following force factors calculated in the center of an element:

- torque — M_{xy};
- shear forces — Q_x, Q_y;
- bending moments — M_x, M_y.

Shell

The selection is performed taking into account the following force factors calculated in the center of an element:

- normal stresses — N_x, N_y (only in shells);
- shear stresses — T_{xy} (only in shells);
- torque — M_{xy};
- shear forces — Q_x, Q_y;
- bending moments — M_x, M_y.

The areas of the upper and lower longitudinal reinforcement, areas and spacing of transverse reinforcement are calculated in the result of the work of the module. The arrangement and identification of the upper and lower longitudinal reinforcement, and transverse reinforcement for sections of an element of a reinforced concrete shell are given in Fig. 19.3-3.

It should be noted that the distance to the center of gravity of reinforcement can be defined by two or four numbers. In the first case, the value $\mathbf{a_1}$ corresponds to the distance to the center of gravity of the lower reinforcing mesh, and $\mathbf{a_2}$ corresponds to the distance to the center of gravity of the upper reinforcing mesh. In the second case, $\mathbf{a_1}$ and $\mathbf{a_2}$ are specified for the reinforcement placed respectively below and above along the X_1 axis, and $\mathbf{a_3}$ and $\mathbf{a_4}$ are specified for the reinforcement along the Y_1 axis.

Deep Beam

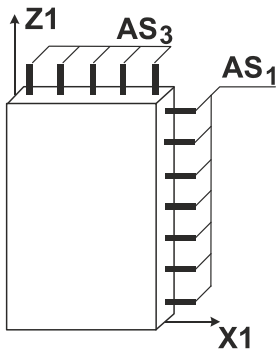


Figure 19.3-4. *Reinforcement of elements of a deep beam*

It enables to check and select reinforcement of reinforced concrete deep beams (plane stress state) for ultimate and serviceability limit states (strength and crack resistance). The module calculates an element of a reinforced concrete deep beam for the action of the following force factors calculated in the center of an element:

- normal stresses — N_x, N_z;
- shear stresses — T_{xz}.

The areas of reinforcement working in sections orthogonal to the local axes of the local coordinate system X_1 and Z_1 are calculated in the result of the work of the module. The arrangement and identification of the selected reinforcement for sections of an element of a reinforced concrete deep beam are given in Fig. 19.3-4.

Before performing the selection of reinforcement in the elements of a plate, shell or deep beam, it is necessary to make sure that the forces in the reinforced elements have been obtained in the right directions. Otherwise, you should use the operation of changing the direction of the output of forces (see Chapter "Specifying Characteristics of Nodes and Elements**") to specify the necessary directions and repeat the calculation of forces and DCF (in the *Continued analysis* mode).**

19.4. Work with the Postprocessor

The control of the operations of the input of initial data, analysis of the load-bearing capacity and selection of reinforcement in the elements of reinforced concrete structures is performed from the **Reinforced concrete** toolbar. The following toolbars contain the controls of these operations:

— **Input data**;

— **Selection of reinforcement**;

— **Check of reinforced concrete**;

— **Punching.**

The design combinations of forces (DCF) must be specified, otherwise the functions of the analysis of reinforced concrete structures will not be available. You should also make sure that the codes specified at the generation of DCF correspond to the codes used in the analysis of elements of reinforced concrete structures.

19.5. Structural Members and Groups of Structural Members

The analysis of the load-bearing capacity of bar elements of reinforced concrete structures is performed for **groups of structural members** (in such a group each finite element is considered as an individual structural member) and for **structural members**, each of which consists of one or several finite elements. A finite element can be included only in one group of structural members. Moreover, all finite elements included in a group will have the same effective length factors.

When generating ***groups of structural members*** you should keep in mind that all bar elements included in a group must have the same type of section (sizes of a section can be different), and the same type of finite elements. Moreover, it is correct when the elements included in one group have the same length, although the program does not check this parameter.

Structural member models a physically homogeneous element of a structure, for example, a column of a frame, a beam, or a rib of a ribbed floor slab.

In cases when the rules for working will apply both to structural members and groups of structural members, the general concept of a “structural group” is introduced.

The geometric length of a structural member is equal to the sum of the lengths of the flexible part of its constituent finite elements.

A continuous chain of bar finite elements with the following properties is considered as a structural member:

- elements included in the chain lie on one straight line;
- all elements of the chain have the same type of rigidity;
- all finite elements of the chain have the same type;
- all elements of the chain have the same orientation of the principal axes of the cross-section;
- elements of a chain the “Slab rib” checkbox is not checked for must not have rigid inserts and hinges (hinges and rigid inserts are allowed only in the first and the last node of the chain, and the rigid inserts must be located only in the direction of the local X_1 axis);
- elements of a chain can not be included in other structural groups (even for the analysis of the load-bearing capacity of elements of steel structures) unless these structural groups are specified as “additional”.

Plate elements included in a group of structural members must have the same finite element type in terms of their stress-strain state (shell, plate, deep beam). It is recommended to specify the same orientation of the local axes Z_1 and the same rules for the orientation of the force output axes. The software does not check these factors and the user is responsible for their correct specification.

19.6. Input the Initial Data

The toolbar of the **Input data** mode includes the buttons for invoking the following operations:

— create groups of reinforcement of plates;

— create groups of reinforcement of bars;

— create structural members of the reinforcement of bars;

— input data and save the active group;

—rejection (reset the current state);

— display the design model;

— select active group;

— show active group;

— control the groups of reinforced concrete structural members;

— information on the groups of reinforcement;

— specify the classes of concrete and reinforcement;

— select the design codes.

The toolbar also contains a drop-down list for selecting a structural group.

Structural groups can be created beforehand in the **Groups** toolbar (the **Elements of reinforced concrete structures** operation —). In this case, unless you want to correct them, you can skip the data input mode, and proceed to the selection of reinforcement at once. Otherwise, the structural groups are created and corrected in the **Input data** section of the **Reinforced Concrete** toolbar.

Creating New Structural Groups

The following procedure is recommended for preparing the new structural groups:

- select elements included in the structural group in the model;
- select the **Input data** operation — in the **Reinforced Concrete** section of the toolbar and the toolbar for the preparation of the initial data for the analysis of elements of reinforced concrete structures will appear;
- click one of the buttons corresponding to the type of the created group:
  - — **Create groups of reinforcement of plates**;
  - — **Create groups of reinforcement of bars**;
  - — **Create structural members of the reinforcement of bars**;
- click the button — **Confirmation**;
- the **Reinforcement of Plates** or **Reinforcement of Bars** dialog box (depending on the type of elements of the group) will appear, where you have to specify the name of the group, enter or select from the lists the necessary data, including: the value of the importance factor, class of reinforcement, service factors and maximum diameters of longitudinal and transverse reinforcement, concrete class etc.;

- check the **Slab rib** checkbox, if the bar structural member is considered as a rib of a ribbed slab and has rigid inserts in the nodes of its elements;
- click the **Add** button and close the dialog box by clicking the **Exit** button.

The Reinforcement of Bars and Reinforcement of Plates Dialog Boxes

The **Reinforcement of Bars** (Fig. 19.6-1) and **Reinforcement of Plates** (Fig. 19.6-2) multi-tab dialog boxes are used for specifying the data for the groups of reinforcement.

The characteristics specified in the **General** tab include:

- **Importance factor** — the value is selected from the drop-down list or is specified by the user;
- **Member type** — a shell, deep beam, or slab can be selected for plate elements, and a member under bending or a member under compression and bending (tension and bending) can be selected for bar elements; regardless of the value and sign of the longitudinal force it is taken as zero for a member under bending, even if this force appeared in the static or dynamic analysis. Similarly, all stresses N_x, N_y, T_{xy} for the members of slab type are taken as zero, and M_x, M_y, M_{xy}, Q_x, Q_y — for the members of deep beam type;
- **Effective length factors** or **effective lengths of elements** (for bar elements under compression and bending). You can specify it either in the form of the effective length factor or the effective length, which can be selected from the drop-down list next to the text field;
- **Stress state** — uniaxial or biaxial bending; in the case of the uniaxial stress state forces M_z and Q_y are taken as zero, except for the ring section, where the analysis is performed for the moment $M = \sqrt{M_y^2 + M_z^2}$ and the shear force $Q = \sqrt{Q_y^2 + Q_z^2}$;
- Characteristics of reinforcement, which include **Class** of longitudinal and transverse reinforcement, **Service factors**, and the maximum **Diameter** of longitudinal reinforcement allowable at the selection. When the reinforcement is selected taking into account the crack resistance, diameters of longitudinal and transverse reinforcement which will be taken into account when determining the crack opening width are specified in the **Diameter** column;
- **Random eccentricity** (for bar elements under compression and bending) — if zero values are specified, the value of eccentricity is taken automatically according to the recommendations of codes;
- **Distances to the center of gravity of reinforcement**;
- **Maximum percentage of reinforcement**, if it is exceeded, it will be impossible to select reinforcement;
- **Analysis on crack resistance** — after checking this checkbox the **Crack Resistance** tab appears, where you can specify the category of crack resistance, and the allowable short-term and long-term crack opening width (diameter of reinforcement considered when determining the distance between cracks is taken from the table in the **General** tab);
- **Coefficients allowing for seismic action for normal and oblique sections** (if zero values are specified, the values are taken by default according to the respective seismic codes); if the calculation is performed according to SP 63.13330, this table also enables to specify additional service factors for concrete and reinforcement (longitudinal and transverse) for combinations including special actions (see Annex B of SP 296.1325800.2017 and Annex A of SP 385.1325800.2018) and an additional service factor which reduces the design strength values and is considered in Sec. 5.11 and Annex C of SP 296.1325800.2017. When the analysis is performed for special loadings, characteristic properties of materials are used instead of the design ones (see Sec. 5.6 of SP 296.1325800.2017). It should be noted that the codes do not provide data on the characteristic resistance of the transverse reinforcement. Therefore, the following value is used in the analysis R_{sw} γ_s,γ_s=1,15 (safety factor for reinforcement). The characteristic resistance of the compressive resistance of the longitudinal reinforcement is determined in a similar way. The user can also specify the reduction factor of limit relative height of the compressed area, which is provided in some seismic codes (see, for example, Sec. 6.7.2 of SP 14.13330). Its default value is 0.85. The user must/can enter the value of this factor due to the fact that the codes provide its value

only for some particular cases (design seismicity of 7, 8 or 9). The codes do not provide any recommendations for the cases when the seismicity is not expressed as an integer or it is expressed as a number greater than 9.

- The **Statically indeterminate system** checkbox enables to specify whether the system is statically indeterminate.

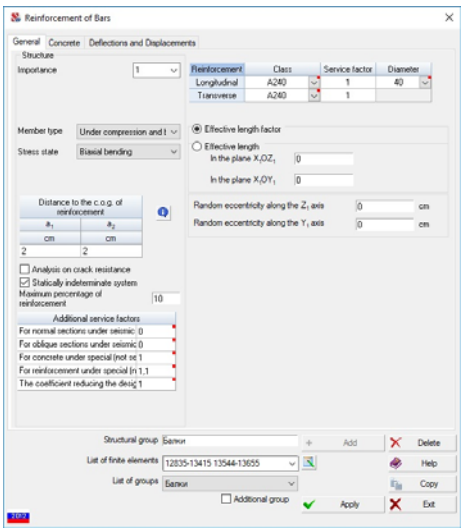


Figure 19.6-1. *The* **General** *tab of the* **Reinforcement of Bars** *dialog box*

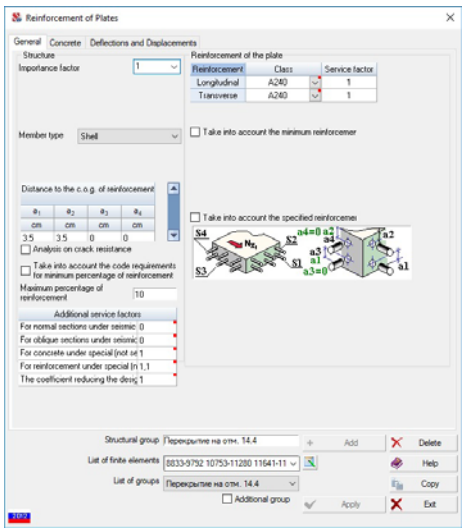


Figure 19.6-2. *The* **General** *tab of the* **Reinforcement of Plates** *dialog box*

Selection with Limitation

When selecting the reinforcement in plates you can take into account the following limitations on the minimum reinforcement:

Take into account the code requirements for minimum percentage of reinforcement —when the respective checkbox is checked the percentage of reinforcement of the plate is determined as for a bar of rectangular cross-section (cut from the plate), one of the sides of which (width) is equal to 1m, and the second one (height) is equal to the thickness of the plate. The area of the reinforcement per running meter provided in the result of this selection is not less than the area necessary to meet the requirements of the codes for the minimum percentage of reinforcement.

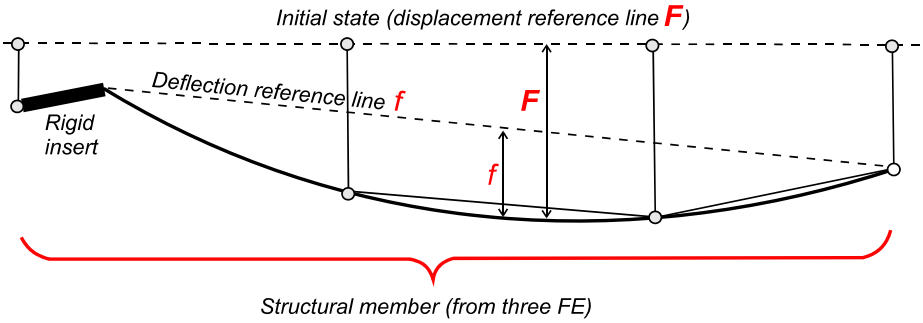
Take into account the specified reinforcement — if the reinforcement is specified for a particular finite element included in a structural group and the respective checkbox is checked, the area of reinforcement per running meter obtained in the result of the selection will be not less than the area of the specified reinforcement per running meter.

Take into account the minimum reinforcement — if the respective checkbox is checked, a table appears where you can specify the arrangement of reinforcement (diameters and spacing) for a structural group. In the result the area of the selected reinforcement per running meter will be not less than the area of reinforcement per running meter specified for a structural group.

Accounting for the Maximum Allowable Deflections and Displacements

The maximum allowable horizontal and vertical deflections and displacements are taken into account in accordance with the code selected for the analysis. For example, if SP 20.13330.2011 is used, the checks are performed in accordance with Section F.2.1 of Annex F, and when you are working with DSTU B.B.1.2-3: 2006 (Deflections and displacements) — according to item 5.1 of Section 5.

The deflection reference line is a straight line that connects the beginning and the end of the flexible part of a structural member. The figure below shows an example where the structural member consists of three finite elements.



The check is performed on the basis of the parameters selected in the respective table in the **Deflections and Displacements** tab.

Maximum allowable displacements and deflections:	Relavite k*L	Absolute mm
vertical deflection for all loads	Automatically	0.7
vertical deflection for permanent and long-term loads	0.00833	0.7
vertical deflection only for live loads	0.00833	0.7
vertical displacements for all loads	0.00667	0.7
vertical displacements for permanent and long-term lc	0.00667	0.7
vertical displacements for live-loads	0.00667	0.7
horizontal deflection for all loads	0.00833	0.7
horizontal deflection for permanent and long-term loac	0.00833	0.7
horizontal deflection only for live loads	0.00833	0.7
horizontal displacements	Unlimited	Unlimited
horizontal displacements for permanent and long-term	Unlimited	Unlimited
horizontal displacements only for live loads	Unlimited	0.7

You can specify both relative and absolute values of the allowable deflections and displacements. In the first case, the following variants of the accounting for displacement are considered (the variants can be selected from the drop-down list of the respective column of the table):

- **Unlimited** — deflections and displacements are not taken into account in the check;
- **Automatically** — relative (with respect to the length of the structural member) deflections and displacements are calculated by the software depending on the length of the member; the range of lengths and corresponding limit displacements are determined by the code;
- the limit value is selected in the list.

The absolute values are specified by the user or are not taken into account if the "**Unlimited**" variant is selected.

If both relative and absolute limitations are simultaneously specified, the value resulting in a greater coefficient is considered when calculating the utilization factor of restrictions.

It should be noted that the automatic check by the relative value makes sense only in those cases where a structural member has no constraints in the span between the end nodes.

Taking into account the fact that there are situations when the designer is interested in deflections and displacements caused only by the variable actions (for example, the check of the instability of the structure, or the case when the displacements from the self-weight are counteracted by the camber), the software enables to perform the checks for all loads, or, separately for only the temporary ones. The type of calculation is selected by checking the checkboxes in the respective lines and activating the design combinations. In any case the checks are performed taking into account the design combinations of deflections and displacements obtained on the basis of the characteristic values of loads.

Maximum allowable displacements and deflections are taken into account only in the check mode and are not used when selecting the sections.

Importance Factors

All design modes require data on the importance factor according to the selected safety codes (GOST 27751-88, GOST 27751-2014, DBN B.1.2-14-2009, …).

The following importance factors are used in the analysis:

- ultimate limit state importance factor γ_n;
- serviceability limit state importance factor;

- importance factor for accidental/special design situations (it is also used in the progressive collapse and fire resistance analyses).

The factor γ_n can be selected from a drop-down list in accordance with the characteristic of a structure (it can be of *special*, *essential* or *limited* importance) or entered by the user (in non-standard situations).

DBN B.1.2-14-2009 provides three importance factors (for the ultimate limit state, for the serviceability limit state and for accidental situations). The user can specify each of these three factors. GOST 27751-88 provides only one factor, which is used in all design situations. If GOST 27751-2014 is selected as a safety code, the user-defined factor γ_n is used in the ultimate limit state analyses. The following value is assumed for the serviceability limit state analyses γ_n=1.0 (but not greater than γ_n for the ultimate limit state in order to avoid a situation in which the characteristic values turn out to be greater than the design ones). And the following value is used for the analysis of special design situations γ_n =1.0 (see Sec. 5.5 of SP 296.1325800.2017).

Advanced Settings

In many modes there is a button next to the icon indicating the selected design codes which invokes a special dialog box for specifying additional calculation parameters.

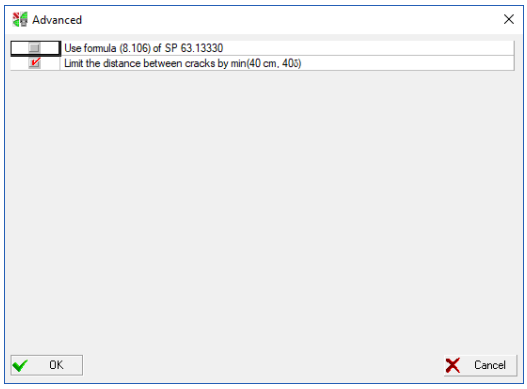


Figure 19.6-3. *The* **Advanced Settings** *tab*

- ***Limit the distance between cracks to min(40 cm, 40∅)***

The program performs the analysis of plate elements based on the theory of N.I. Karpenko (see Sec. 19.16 for more details), since SP 63.13330 provides recommendations for calculating flat elements only for particular cases (slabs and deep beams). The codes do not provide any clear guidelines for the calculation of shell elements with simultaneously acting membrane and bending groups of forces. **SCAD** determines the crack opening width by the formulas recommended by N.I. Karpenko of the type $(\varepsilon_x+\varepsilon_y)l$, where l is the distance between cracks, ε_x,ε_y are strains in the respective directions.

To determine the crack opening width in flat elements SP 63.13330 refers to Section 8.2 (requiring to neglect the torque), which considers the formation of cracks in bar elements and contains the following limitation on the distance between cracks: “not less than $10d_s$ and 10 cm and not more than $40d_s$ and 40 cm”. The authors of the program do not know whether this limitation is reasonable with respect to plate elements, therefore it is not used by default. However, you can activate it using this additional setting.

- ***Use the formula (8.106) from SP 63.13330***

The default algorithm for calculating reinforced concrete flat elements is based on two separate calculations — for the action of Q_x and Q_y. SP 63.13330 contains the following recommendation for the calculation for shear forces — formula (8.106)

$$\frac{Q_x}{Q_{x,ult}}+\frac{Q_y}{Q_{y,ult}}\le 1.$$

This formula sums the stresses at various sites and can hardly seem reasonable. Based on the vector addition rules, it would be more logical to consider the shear force $Q=\sqrt{Q_x^2+Q_y^2}$. The following paper S.N. Karpenko, *Modern design techniques of high buildings made of monolith reinforced concrete,* High Buildings, 2007, No. 3, 34-39, for example, contains a similar recommendation on the analysis of flat elements.

- ***Increase the area of longitudinal reinforcement when implementing Sec. 8.1.34 of SP 63.13330***

Section 8.1.34 which has appeared in SP 63.13330 regulates the consideration of a longitudinal force in the analysis of the behavior of a bar element under shear forces. This section is obviously taken from EN1992-1-1 (see Sec. 6.3.2). However, unlike the Eurocode, where the coefficient φn is used only for prestressed structures, SP 63.13330 requires to use it even when there is no prestress.
When the longitudinal forces are relatively large, there are two methods for selecting transverse reinforcement. If the first (default) one is used, the transverse reinforcement is selected after the longitudinal one. The previously selected longitudinal reinforcement is taken into account when calculating the coefficient φ_n, but the area of longitudinal reinforcement does not change when selecting transverse reinforcement. In the second case having established that the value φ_n is too small, you can try (it is not always possible) to increase the area of longitudinal reinforcement and select the transverse reinforcement. The second method in most cases leads to an unreasonably high reinforcement ratio (it is much more effective to increase the formwork sizes of the section) and is not used by default. However, this additional setting enables to activate it.

- ***Ignore torsion***

This option can be used when the analysis is performed according to EN 1992-1-1. It has been added due to the fact that Sec. 6.3.1(2) EN 1992-1-1 mentions a rather wide range of cases when torsion can be ignored. Moreover, using this option enables to avoid one inconsistency of EN 1992-1-1. The thing is that if a member is subjected to shear, both the resistance of concrete and the resistance of reinforcement are taken into account in the analysis (see Sec. 6.2.3). If a member is simultaneously subjected to both torsion and shear, then Sec. 6.3.2(4) limits its resistance by the capacity of the concrete struts.

- ***Implementation of Sec. 8.1.34 of SP 63.13330***

The new Sec. 8.1.34 of SP 63.13330 regulates the shear analysis during eccentric compression/tension. It should be noted that the rules for calculating the coefficient φ_n in this section are almost literally taken from EN 1992-1-1, but in EN 1992-1-1 these rules apply only to prestressed elements. In addition, Sec. 8.1.34 of SP 63.13330 does not provide any rules for calculating the coefficient φ_n in the case when the average stress from the tensile force σ_t exceeds R_{bt}. The default value is $\varphi_n=10^{-6}$, but it is also possible (and quite logical) to set the following value $\varphi_n=0.5$ (which corresponds to $\sigma_t = R_{bt}$). In order to avoid the above problems, the software allows to set the lower limit of the coefficient φ_n for tension, as well as the upper and lower limits of φ_n for compression. In particular, by setting all limit values to 1.0, you can perform the analysis corresponding to the recommendations of EN 1992-1-1, and by setting the lower limit φ_n equal to 0.5, you can determine φ_n for $\sigma_t > R_{bt}$.

Change No. 2 to SP 63.13330.2018 changes the rules for calculating the coefficient φ_n. When choosing SP 63.13330.2018 as the design code, this change will be taken into account, but a special option allows you to refuse to use this change.

- ***Slenderness Control***

SCAD does not perform the slenderness analysis by default. You can use this checkbox to perform this analysis for eccentrically compressed bar elements and to specify the value of the limit slenderness. The slenderness control is not performed when selecting the reinforcement.

- ***Normalized Normal Force***

In the case of the seismic analysis of ductile reinforced concrete walls Sec. 5.4.3.4.1(2) of EN 1998-1 requires that the value of the normalized axial load should not exceed 0.4. You can use this checkbox to perform an additional calculation of the factor equal to $N/(0.4 \cdot A \cdot f_{cd})$ during the check in order to control the fulfillment of this requirement.

- ***Increase Design Shear Forces for Seismic Combinations***

You can use this checkbox to increase the values of shear forces by 50% for the seismic analysis and thereby satisfy the requirement of Sec. 5.4.2.4(7) of EN 1998-1.

Additional Information on the Initial Data

When preparing the initial data some values do not have to be specified — they are taken by default according to the recommendations of the codes. In particular, the default values are provided for the following parameters:

- **service factors for longitudinal and transverse reinforcement.** These factors can be used to correct the values of design resistances R_s, R_{sc}, R_{sw}, which are taken as 1 by default;
- **coefficients allowing for seismic action**. When performing the analysis for seismic action, two coefficients are introduced in the **Reinforcement of Plates/Bars** dialog box: one is used in the strength analysis of normal sections, and the other one is used in the strength analysis of oblique sections of reinforced concrete elements. These coefficients are taken into account for those DCF which include a seismic loading, and their values are specified by default as 1.2 and 0.9 respectively. If the given structure has not been calculated for the seismic action, the values of coefficients in these positions are not taken into account;
- **random eccentricity** in the X_1OZ_1 and X_1OY_1 planes is specified only for bars. If the value of random eccentricity is specified as zero, the value calculated according to Sec. 1.21 of SNiP 2.03.01-84* (Sec. 4.2.6 of SP 52-101-2003, Sec. 7.1.7, 8.1.7 of SP 63.13330) will be used in the analysis. If the specified random eccentricity is not equal to zero, the greater of the two values: calculated according to the codes and the specified value, will be used (according to Sec. 3.50 of Guide to SNiP 2.03.01-84* and Sec. 3.6 of Guide to SP 52-101-2003 and Sec. 7.1.7, 8.1.7 of SP 63.13330);

- if the calculation is performed according to SP 16.13330, then you will be able to specify a service factor for concrete and reinforcement (longitudinal and transverse) in accordance with Annex B of SP 296.1325800.2017 in the special table (moreover, the importance factor $\gamma_n = 1.0$ in accordance with Sec. 5.5 of SP 296.1325800.2017 will be used for such loadings). Furthermore, an additional service factor which reduces the design strength values and is considered in Sec. 5.11 and Annex C of SP 296.1325800.2017 can be taken into account. When the analysis is performed for special loadings, characteristic properties of materials are used instead of the design ones (see Sec. 5.6 of SP 296.1325800.2017).

If a structural member (e.g., a column) consists of a number of finite elements and has a longitudinal force constant along the length, each finite element has to be assigned an effective length corresponding to the effective length of the structural member. In other cases values obtained in the stability analysis mode should be used when specifying the effective lengths.

If the effective lengths are specified as equal to zero, the value of the longitudinal force is neglected, and the bar will be reinforced as that under bending (even if it is specified as that under compression and bending in the group of reinforcement).

Coefficients specified in the initial data and their consideration in the characteristics of concrete and reinforcement are given in Table 19.6-1.

Table 19.6-1

Coefficient		Characteristics according to SNiP				
		E_b	R_b	R_{bt}	R_s, R_{sc}	R_{sw}
Hardening factor		+				
γ_{b2} = 1.1*			v	v		
γ_{b2}**			v	v		
Resulting coefficient γ_b with no γ_{b2}			+	+		
For longitudinal reinforcement					+	
For transverse reinforcement						+
Coefficients allowing for seismic action	normal sections		v		w	
	oblique sections			v		w

Designations to Table 19.6-1:

* — coefficient γ_{b2} from item 2b Table 15 of SNiP 2.03.01-84*;

** — coefficient γ_{b2} from item 2a of Table 15 of SNiP 2.03.01-84* (it is taken from the initial data);

+ — coefficient is always taken into account;

v — one coefficient from the respective column is always taken into account;

w — it is taken into account when there are seismic loadings.

When the calculations are performed according to SP 52-101-2003 and SP 63.13330, the user has to specify several service factors for concrete. Their usage is similar to that described in the Table 19.6-1 given above. The only peculiarity is that according to these codes only the value R_b is multiplied by coefficients γ_{b2} (allowance for the failure behavior) and γ_{b3} (allowance for the vertical position during concreting).

The following information is specified in the **Concrete** tab (Fig. 19.6-4): concrete type, concrete class, service factors for concrete, conditions of hardening and hardening factor.

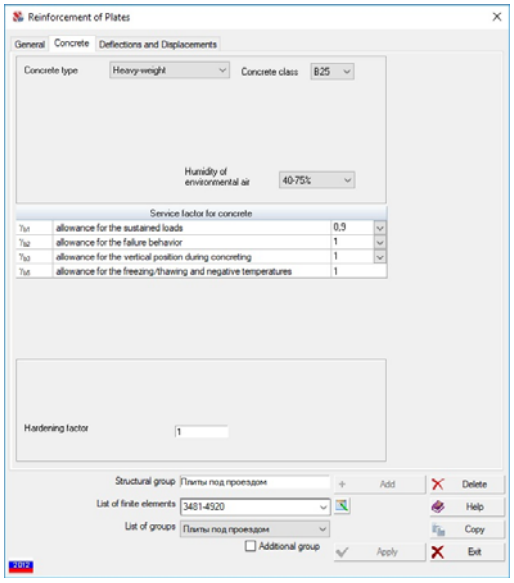


Figure 19.6-4. *The* **Concrete** *tab*

The class of heavy-weight and fine-grained concrete is selected from the **Concrete class** list. If lightweight concrete is selected, it is necessary to select the grade by average density first, and then the concrete class and aggregate.

If the analysis is performed according to SP 52-101-2003, only the heavy-weight concrete is available, because the rules for designing from other types of concrete are not provided in this document. Service factors for concrete are specified in the table in this tab.

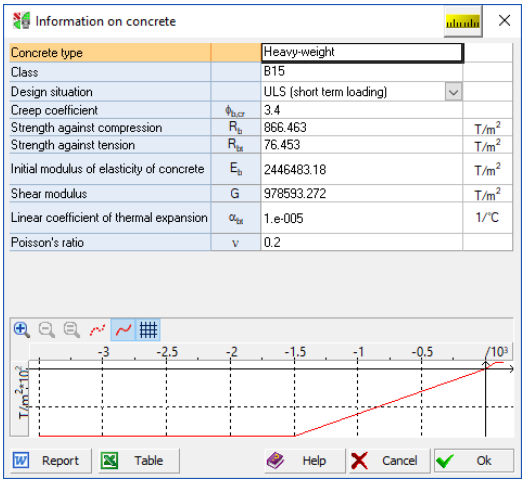
Service factor for concrete γ_{b2} (γ_{b1} in the case of analysis according to SP 52-101-2003 and SP 63.13330) allows for the sustained loads. The value of this factor is specified as equal to 1 or 0,9 (item 2 a of Table 15 of SNiP 2.03.01-84* and Sec. 5.1.10 a of SP 52-101-2003 and Sec. 6.1.12 of SP 63.13330)

and is taken as one by default. In those cases when according to the conditions of the analysis it is necessary to take another value of this factor, it can be specified directly in the list.

If a combination of loadings includes short-term loads, it is assumed that γ_{b2} = 1,1 in the calculation (regardless of the specified value) according to the recommendations of item 2 *b* of Table 15 of SNiP 2.03.01-84* (and similar sections of SP).

If the value of the initial elastic modulus of concrete is different from the table value, the hardening factor is specified, which is used to correct this value (it is assigned only in case of the natural conditions of hardening of concrete).

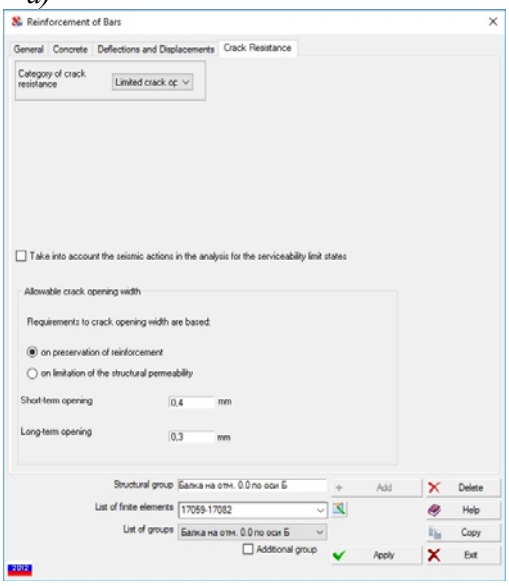
The button invokes the dialog box, which provides the design properties of the material.



The **Design situation** drop-down list enables to select the type of analysis (ultimate limit state, serviceability limit state,...) in order to display the respective parameters.

In those cases when the analysis on crack resistance is required, information on the category of crack resistance, allowable crack opening width etc. has to be specified. All these data are specified in the **Crack Resistance** tab (Fig. 19.6-4).

a)



b)

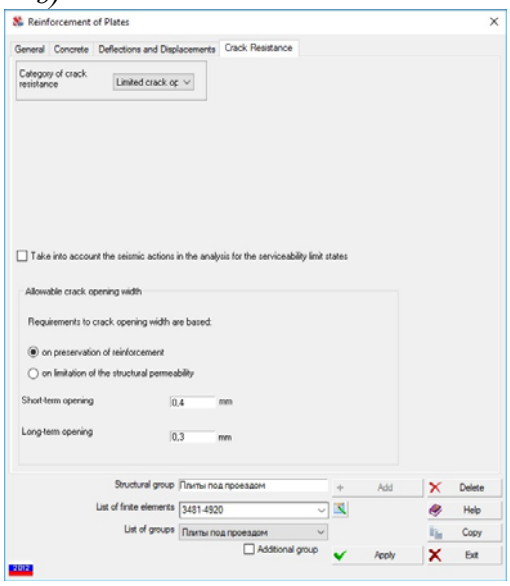


Figure 19.6-5. *Data on crack resistance*

If the 1-st category of crack resistance is selected (no cracks), no additional information has to be specified in this tab.

If the analysis is performed according to SNiP 2.03.01-84* and the 3-rd category of crack resistance is selected (Fig. 19.6-5, *a*), it is necessary to select the conditions of operation of the structure, mode of concrete humidity, and humidity of environmental air in the respective lists, and then specify the allowable short-term and long-term crack opening width (the values recommended by SNiP 2.03.01-84* based on the conditions of operation of the structure and parameters of humidity are taken by default).

If the analysis is performed according to SP 52-101-2003 or SP 63.13330 and the **Limited crack opening width** item is selected in the drop-down list (Fig. 19.6-4, *b*), it is necessary to specify the requirements to crack opening width (*based on preservation of reinforcement* or *based on limitation of the structural permeability*). The values of maximum allowable long-term and short-term crack opening width recommended by Sec. 7.2.3 of SP 52-101-2003 or Sec. 8.2.6 of SP 63.13330 will be automatically specified. The user can correct these values if necessary.

It should be noted that there is an inaccuracy in SP 52-101-2003 concerning the use of the safety factor for load in the calculation for the serviceability limit states. Sec. 7.1.3 of this document states that "When the calculation is performed for the serviceability limit states, the loads are taken with a safety factor for load $\gamma_f = 1{,}0$.". Such concepts as the analysis on crack formation and the analysis of their opening and closing are not clearly distinguished. As a result all types of calculations are performed taking into account the characteristic loads. SNiP 2.03.01-84 * and SP 63.13330 do not have this inaccuracy, and when the analysis on crack formation is performed, design values of loads are used, i.e. $\gamma_f > 1{,}0$.

Design values of loads are used in SCAD++ for SP 52-101-2003 in the analysis on crack formation, if "no cracks" option was selected.

When the analysis is performed according to SP 52-101-03, Sec. 3.52 of the Guide specifies how to take into account the effect of a longitudinal force in the calculations for transverse forces. It should be noted that the longitudinal reinforcement is not taken into account in Sec. 3.52 in the formula for the value $N_b=1{,}3R_bA$. There is a letter from the authors of SP which describes an alternative method for calculating the value N_b taking into account the longitudinal reinforcement ($N_b=R_bA+R_sA_{s,tot}$). There is a special checkbox prompting the application to use this formula to determine N_b. Since there are two methods for calculating N_b (according to SP and according to the letter), the program calculates N_b in two different ways and uses the highest value.

The name of a structural member or group must be specified in the **Structural group** text field.

List of groups includes a list of structural groups created earlier.

The lower part of the dialog box (Fig. 19.6-6) contains a list of *finite elements* with numbers of elements included in a structural member or group. This list can be edited directly in the window. If there are selected elements in the model, you can perform the following actions with the help of a set of operations which can be opened by clicking the button on the right from the list:

- **Create list of elements from selected elements**, i. e. actually replace the list of elements created earlier by the list of elements selected in the model;
- **Create list of elements from group's data and selected elements**, i. e. add the elements selected in the model to the list of elements created earlier;
- **Create list of elements from group's data excluding selected elements**, i. e. exclude the elements selected in the model from the list of the group created earlier.

The respective button is used to edit the existing list of the group, clicking which invokes a dialog box with a list of elements.

If after performing a series of operations with the list of elements (before clicking the **Apply** button), it is necessary to return to its initial state, you should use the first operation in the considered list — **Create list of elements from group's data**, and the initial list of the group will be restored.

In order to delete a structural member or a group of structural members it is necessary to select a name from the list in the **Input data** toolbar and click the button . Click the **Delete** button in the dialog box that will appear.

If you have to delete several objects, it is necessary to perform the above operations for a single object, and then consecutively select the objects you want to delete from the list of structural members (or groups) and click the **Delete** button without closing the dialog box.

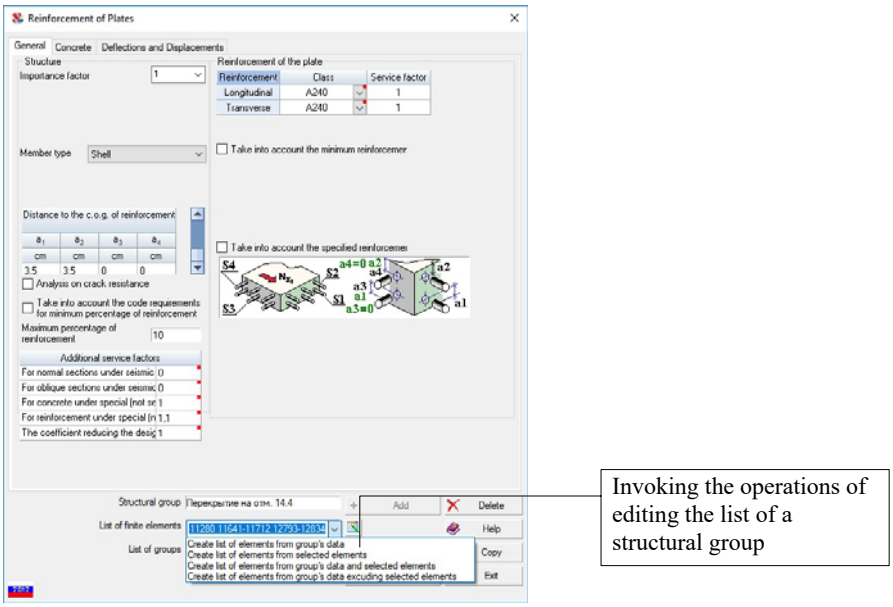


Figure 19.6-6. *Editing the list of elements included in a structural group*

The **Copy** button invokes a dialog box with a list of groups. After selecting a group from the list and clicking the **OK** button, you can copy the data from the group created earlier. If this button is pressed when the **General** tab is active, all the data on all tabs are copied (the lists of finite elements and the name of the group are obviously not copied). If this button is clicked when the **Concrete** or **Crack Resistance** tab is active, only the data in the active tab is copied.

Correcting the Initial Data of the Structural Groups

It is necessary to perform the following actions to change the parameters of a group of structural members created earlier:

- select the **Input data** operation — in the **Reinforced Concrete** toolbar and the respective toolbar will appear;
- select the name of the structural group you want to correct in the **Groups of reinforcement** list (the respective button is pressed in the toolbar);
- click the button — **Confirmation**;
- the **Reinforcement of Plates** or **Reinforcement of Bars** dialog box will appear (depending on the type of elements of the group), where you have to make necessary changes to the parameters of the group and/or list of elements and click the **Apply** button;
- close the dialog box by clicking the **Exit** button.

If it is necessary to change the parameters of other groups, the adjustments can be performed without closing the dialog box. You just have to select the name of the group you want to correct in the list of groups and perform the above actions.

Dividing a Group of Reinforcement into Two Groups or More

If based on the results of the selection of reinforcement a group of structural members has to be divided into two groups or more, it is necessary to perform the following actions:

- select the **Input data** operation — in the **Reinforced Concrete** toolbar and the respective toolbar will appear;

- select the name of the group in the **Groups of reinforcement** list (the respective button is pressed in the toolbar);
- click the button — **Confirmation**;
- the **Reinforcement of Plates** or **Reinforcement of Bars** dialog box will appear (depending on the type of elements of the group), where you have to specify the name of the new group in the **Structural group** field and click the **Add** button;
- correct the list of elements of the new group removing the numbers of elements which "stay" in the initial group;
- correct the parameters of the new group;
- select the name of the initial group in the list of groups and correct the list of elements removing the elements that were transferred to the new group;
- correct the parameters of the initial group if necessary;
- click the **Apply** button;
- close the dialog box by clicking the **Exit** button.

You can repeat the above actions to create new groups by dividing the initial group.

19.7. Main and Additional Groups

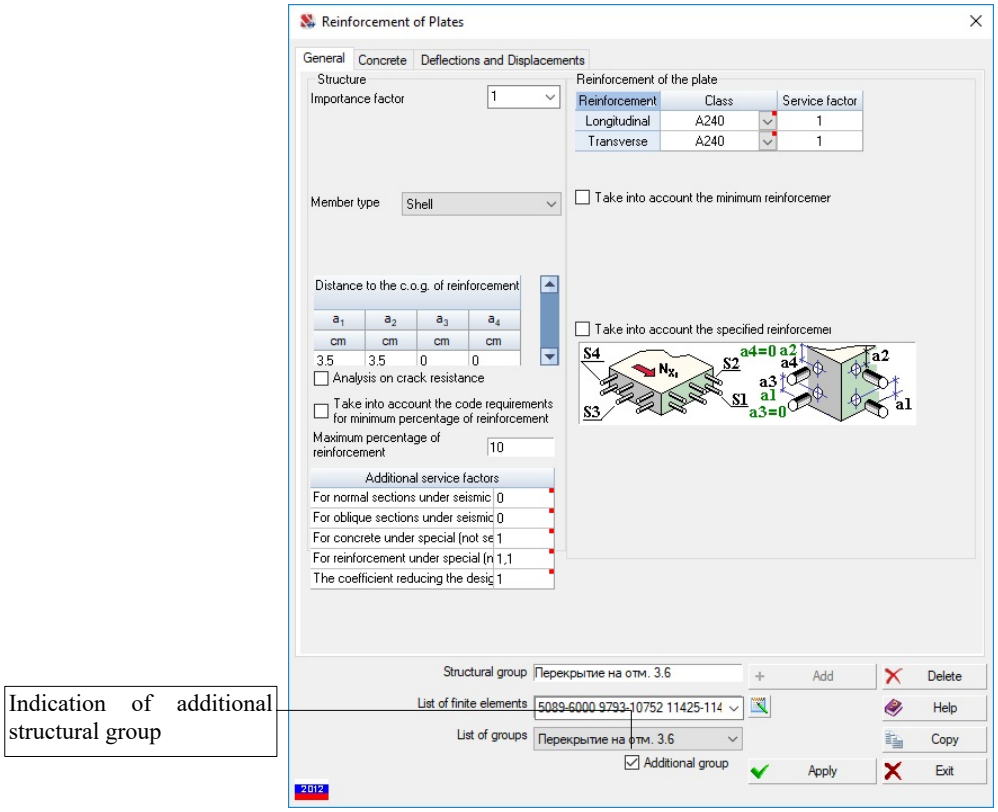


Figure 19.7-1. *The* **Reinforcement of Plates** *dialog box (additional group)*

If a finite element simultaneously belongs to several structural members (or groups of structural members), it causes many problems and questions. For example, if you perform a selection of reinforcement, the results of which group should be considered as the result of selection? Which results should be used when transforming the selected reinforcement into the specified reinforcement? Which data should be used to calculate the value of K_{max} after performing the check when displaying the utilization factor of restrictions for the whole model and not for an individual structural member?

You can create additional structural groups (Fig. 19.7-1). This enables to work with a set of structural groups with matching finite elements. There are the following limitations when using the main and additional groups:

- a finite element can not belong simultaneously to two main groups;
- additional groups are not used when determining K_{max} for the whole model;
- additional groups are not used when transforming the results of the selection of reinforcement into the specified reinforcement;
- if a finite element simultaneously belongs to the main and additional group (groups), after clicking the buttons **Reinforced concrete. Factors**, **Reinforcement**, ... in the information on the element, a menu appears with the list of groups and you have to select a group the results of which you want to be displayed;
- additional groups are not used in the analysis for the progressive collapse.

Additional groups are distinguished by the green background of icons in the lists of structural groups in toolbars and dialog boxes. Unique names should be given when creating the main and additional groups. The **Control the groups of reinforced concrete structural members** operation considered below is used to change the status of the group.

19.8. Selection of Elements of Groups

If a group of structural members or a structural member is selected in the list of structural groups in the **Input data** toolbar, clicking this button will select all finite elements of the group. Red color is assumed for the selected nodes and elements by default. You can change it in the **Color Palette Settings** section of the ***Settings*** menu (the **Model** tab, the **Select nodes and elements** item).

19.9. Active Group Visualization Mode

In those cases when you do not need to perform any operations with the selected structural group, and you only want to highlight it in the model, an operation of visualization of a structural group is used. A group selected in the list of structural groups in the **Input data** toolbar will be "highlighted" in purple in the model.

19.10. Control of the Groups of Reinforced Concrete Structural Members

The operation is used to control the attributes of structural groups and enables to perform the following actions:

- rename a group;
- sort groups by a specified criterion;
- delete a group;
- combine identical groups;
- change the status of structural groups (transform a group into an additional or main one);
- change the place of the selected group in the list of groups (buttons **Up**, **Down**).

This operation invokes the **Control the Structural Groups of Reinforced Concrete Elements** dialog box (see Fig. 19.10-1), with a list of structural groups. Columns of the table of structural groups include:

- current number of the structural group;
- **No.** — number of the group in the order of creation;
- **Type** — icon of the type of the structural group (— bar structural member, — group of bar structural members; — group of plate structural members). If a group is additional, the respective icon has a green background;
- **Name** — name of the structural group;
- **Number of elements** in the group;
- **Additional —** this checkbox indicates additional structural groups.

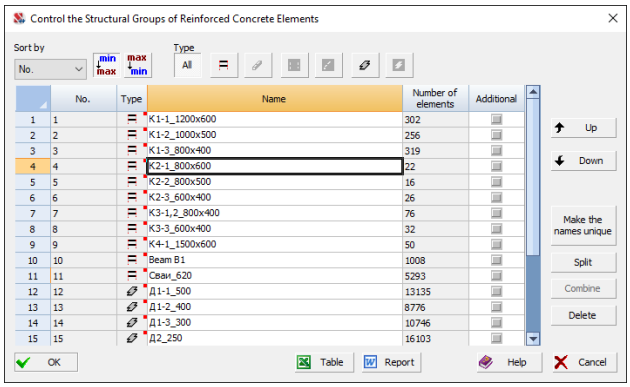


Figure 19.10-1. *The* **Control the Structural Groups of Reinforced Concrete Elements** *dialog box*

In order to rename a group place the cursor over the name of the respective group and double left click. The line will become available for editing.

The characteristic of the group with respect to which the sorting is performed is selected from the **Sort by** list. The following types of sorting are provided:

- by the number (in the order of creation of groups — when the **Min/Max** button is pressed; and vice verse — when the **Max/Min** button is pressed);
- by the type of group (when the **Min/Max** button is pressed structural members will be first in the list and they will be followed by the structural members, and vice verse — when the **Max/Min** button is pressed);
- by the name of the group (in alphabetical order — when the **Min/Max** button is pressed, and vice verse — when the **Max/Min** button is pressed);
- by the number of elements in the group (in the ascending order of the number of elements in the list of structural groups — when the **Min/Max** button is pressed, and vice verse — when the **Max/Min** button is pressed);
- by the checked "Additional" checkbox.

In order to delete structural groups select them in the table, place the cursor over the current number of the group and left click. Then click the **Delete** button. If the groups you want to delete go in the successive order in the list, place the cursor over the first group in the list to select them, click and hold the left mouse button, and drag the pointer across the numbers of the groups you want to delete.

To change the position of one or several structural groups in the list it is necessary to select these groups and move them in the list by clicking the **Up** and **Down** buttons.

If when specifying the data the user has created several groups with the same names, clicking the **Make the names unique** button enables to rename them automatically (by adding the suffix № XXX to the name) and obtain unique names.

In order to combine groups of structural members it is necessary to select these groups in the list while pressing the **Ctrl** key in the keyboard and then click the **Combine** button. The operation will be available only if all parameters of the combined groups of structural members are matching. The dialog box will appear after invoking the operation, where you can specify the name of the combined group.

The **Split** button enables to automate the process of correcting errors in the data on structural members and automatically create structural members and groups of structural members. Clicking this button will perform the following operations for each selected group.

For groups of plate structural members:

- All the elements other than plates are removed from the list of elements.
- Structural members are created for the remaining elements by combining plate finite elements with the same rigidity that lie in one plane, form a connected region, contacting by common nodes.
- All other parameters (material, concrete cover, ...) are inherited from the properties of the original group.

For groups of bar structural members:

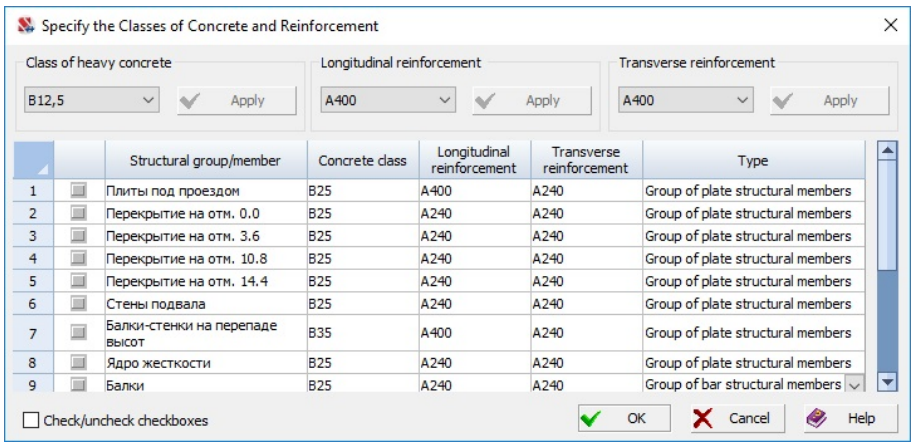
- All the elements other than bars are removed from the list of elements.
- All elements that cannot be interpreted as reinforced concrete are removed from the list of elements.
- Structural members are created for the remaining elements by combining bars with the same rigidity that lie on one line and are successively adjacent to each other. In addition, the absence of hinges in intermediate nodes is controlled. Allowed the presence of rigid inserts, provided that these inserts are the same for all elements.
- All other parameters (material, effective lengths, ...) are inherited from the properties of the original group.
- Groups of structural members are created for all the remaining finite elements, each group containing elements with the same type of rigidity.

Buttons in the upper part of the dialog box enable to filter groups which should be displayed (for example, to display only the additional groups of plate structural members).

All specifications made in the dialog box will be saved and performed after closing the dialog box by clicking the **OK** button. The position of the structural groups in the list of the toolbar will correspond to their position in the list of the dialog box.

Specify the Classes of Concrete and Reinforcement

This operation is used in those cases when several structural groups have the same class of concrete and class of reinforcement, i.e. you can specify these parameters simultaneously for several or even all structural groups. Moreover, this operation is useful when changing the design codes. Invoking the **Specify the Classes of Concrete and Reinforcement** operation opens the respective dialog box (Fig. 19.10-2) with a table listing all structural groups with classes of concrete and reinforcement specified for them.



	Structural group/member	Concrete class	Longitudinal reinforcement	Transverse reinforcement	Type
1	Плиты под проездом	B25	A400	A240	Group of plate structural members
2	Перекрытие на отм. 0.0	B25	A240	A240	Group of plate structural members
3	Перекрытие на отм. 3.6	B25	A240	A240	Group of plate structural members
4	Перекрытие на отм. 10.8	B25	A240	A240	Group of plate structural members
5	Перекрытие на отм. 14.4	B25	A240	A240	Group of plate structural members
6	Стены подвала	B25	A240	A240	Group of plate structural members
7	Балки-стенки на перепаде высот	B35	A400	A240	Group of plate structural members
8	Ядро жесткости	B25	A240	A240	Group of plate structural members
9	Балки	B25	A240	A240	Group of bar structural members

Figure 19.10-2. *The* **Specify the Classes of Concrete and Reinforcement** *dialog box*

It is necessary to perform the following actions to specify these parameters for the groups:

- specify the class of concrete, and classes of longitudinal and transverse reinforcement by selecting them from the respective lists;
- check the checkboxes on the left from the name of a structural group for those groups the selected class of concrete is specified for;
- click the **Apply** button related to the class of concrete;
- perform similar actions for the class of longitudinal reinforcement, and then for the transverse reinforcement by checking the checkboxes of the necessary structural groups and clicking the respective **Apply** buttons;
- after specifying the parameters for the necessary groups, close the dialog box by clicking the **OK** button.

Similar specifications for each structural group separately can be also performed in the dialog boxes for creating groups, and also in the **Reinforcement of Plates** and **Reinforcement of Bars** dialog boxes when creating or editing structural groups directly during the selection of reinforcement. It makes sense to use this operation in the case of specifying parameters for a large number of groups simultaneously.

Moreover, a type of group is specified in the **Type** column and for bar groups you can replace the *Structural member* by *Group of structural members* and vice versa.

Information on the Groups of Reinforcement

This operation enables to obtain the dialog box (Fig. 19.10-3) with the information in all groups of reinforcement. The **Report** button invokes the generation of the report.

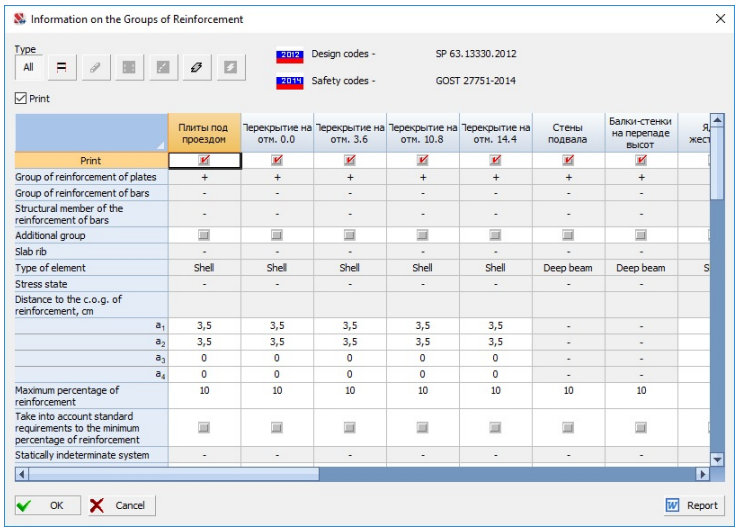
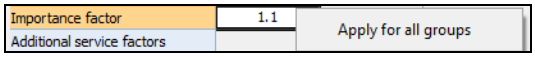


Figure 19.10-3. *The* **Information of the Groups of Reinforcement** *dialog box*

Buttons in the upper part of the dialog box enable to filter groups which should be displayed (for example, to display only the additional groups of plate structural members).

Checkboxes in the **Print** line enable to select only the groups information on which must be given in the report document.

Moreover, this dialog box also enables to change some data (effective length factors, service factors, …). In the case of editable data cells, right-clicking in this dialog box will invoke a context menu, where you can select the **Apply for all groups** option, and the data in the active cell will be copied to all groups:



19.11. Setting the List of Reinforcement Spacing

Data on the reinforcement (selected or specified) in plate elements can be given as intensity (areas per running meter) or as **Diameter/Spacing**. The spacing of reinforcement is specified by the user. List of such spacings can be specified (or changed) with the help of a special dialog box (Fig. 19.11-1), which can be invoked from the window of the project tree using the **Set Reinforcement Spacing** item of the ***Settings*** menu.

This dialog box contains a table of spacings (in *mm*) which can be changed by correcting values, adding spacings (the **Add** button), deleting spacings which (in the user's opinion) should not be used (the **Delete** button). The values in the table can be sorted in the ascending order (the **Sort** button). If the sorting is not performed by the user, it will be done automatically when closing the dialog box. Moreover, the **Standard** button enables to load a set of standard values of spacings into the table.

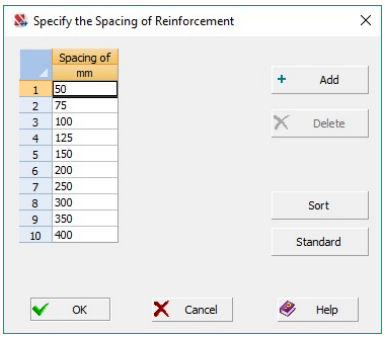


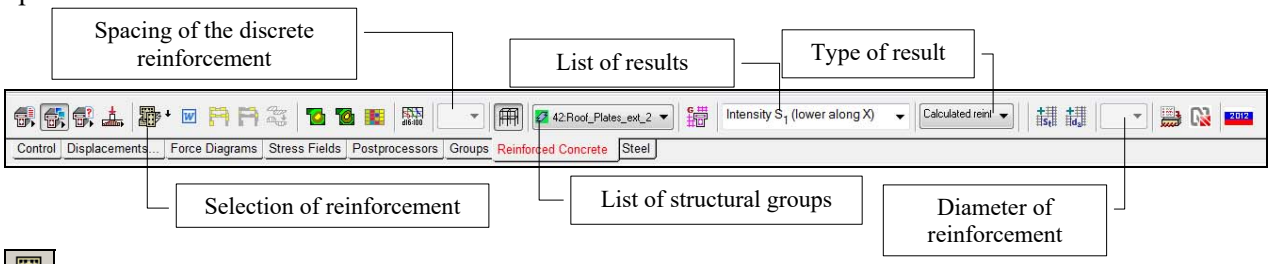
Figure 19.11-1. *The* **Specify the Spacing of Reinforcement** *dialog box*

19.12. Selection of Reinforcement

Selection of reinforcement and display of the results of the selection are performed in the **Selection of reinforcement** section of the **Reinforced Concrete** toolbar. In order to perform the selection it is necessary to create structural members and/or their groups. Moreover, design combinations of forces (DCF) have to be calculated for all finite elements included in these structural members and groups.

When selecting the reinforcement the software does not check the structural limitations on the arrangement of reinforcement, assuming that the user can perform this check in an informal way. Moreover, after obtaining the results of the selection and making sure that the design requirements can not be met, the user may decide to correct the sizes of the formwork.

The toolbar of the **Selection of reinforcement** mode includes buttons for invoking the following operations:



— select reinforcement;

— generate report;

— reinforcement diagrams of bars;

— reinforcement diagrams of bars (color display);

— change the display axis of diagrams;

— switch the output form of the reinforcement results;

— isofields of the reinforcement of plates;

— isofields and isolines of the reinforcement of plates;

— color map of the reinforcement results;

— display the design model;

— show active group;

— additional reinforcement with meshes;

— additional reinforcement with bars;

— select the design codes;

— specify reinforcement based on the results of the reinforcement selection;

— export the reinforcement results to ALLPLAN.

The **Select reinforcement** button — in the respective section of the toolbar is used to invoke the analysis. This button has an arrow which besides the complete selection for all structural groups enables to perform the selection only for those groups that have been modified, to perform the calculation for user' selected groups only (in this case dialog box will appear and user must select groups for calculation), to invoke the operations of the data control (without performing the calculation), view messages that were output during the previous calculation or control, and delete results. Regardless of whether changes have been made in all structural groups or only in some of them, the control of the initial data is performed for all groups and diagnostic messages will be generated for all groups, and not just for those which were recalculated.

The course of the selection is displayed in the **Performing the Calculation** window. Messages which appear in the selection process are output after completing the calculation in the information window and in the log. A report can be generated in the result of the selection (the button).

It should be noted that when selecting the reinforcement in plate elements the area of the selected reinforcement is increased by 5% which enables to allow for the possible increase in the force factors at the edges of the element (values of forces in the center of the finite element are used when selecting reinforcement).

The selection of reinforcement in bar elements is performed on the basis of the design combinations of forces in several sections of the bar (the number of the sections is specified by the user). The selected reinforcement area is increased by 5% to allow for the possible increase of the force factors in the intermediate sections. Moreover, such an increase provides a "protection" from the fact that the forces in SCAD are determined without the consideration of the possible crack formation and the consequent weakening of the section.

Output the Results of the Selection

The results of the selection *for bar elements* are displayed as diagrams, and color scales can be used as well. These are areas of symmetric and asymmetric reinforcement for the upper, lower and side reinforcement — S_1, S_2, S_3, S_4 (for ring and round sections — only asymmetric reinforcement S_1). Moreover, the percentage of reinforcement and the crack opening width are displayed as well. Presentation of the results of selection in the form of a set of bars is provided only in the information on the element.

When displaying the diagrams of reinforcement a bar is divided into n segments, where n is the number of sections in the bar. The lengths of the first and last (along the length of the bar L) segments are equal to $\frac{L}{2(n-1)}$, and the lengths of all other (intermediate) segments are equal to $\frac{L}{(n-1)}$. The value of the factor in each segment of the diagram is taken as the maximum value for all sections of the bar in this segment.

The output of intensity (area per running meter) of reinforcement in the form of isolines, isofields, or color maps is provided *for all types of reinforcement of plate elements*. The results of the selection of longitudinal reinforcement can be given as a set of bars in the directions coinciding with the directions of the force output. This operation can be invoked by clicking the button — **Switch the output form of the reinforcement results**, and the spacing of reinforcement is selected from the respective list in the toolbar.

The percentage of reinforcement of plates is determined according to the following rules — a bar of rectangular section is considered with the sizes $b \times H$, where b = 1m, and H is equal to the thickness of the plate. The percentage of reinforcement in the direction X and Y is determined by the formulas:

$$\mu_x = (S_1 + S_2)/(H_x \times b) \times 100,$$
$$\mu_y = (S_3 + S_4)/(H_y \times b) \times 100,$$
$$S_1 = s_1 \times b / step_1,$$
$$S_2 = s_2 \times b / step_2,$$
$$S_3 = s_3 \times b / step_3,$$
$$S_4 = s_4 \times b / step_4,$$

where

s_1, s_2, s_3, s_4 — cross-sectional area of one bar of the respective reinforcement,

$step_1$, $step_2$, $step_3$, $step_4$ — spacing of reinforcement.

Values H_x and H_y are determined as follows:

$$\begin{cases} H_x = H - a_1, \\ a_2 > 0, H_x = \max(H_x, H - a_2), \end{cases}$$
$$\begin{cases} H_y = H - a_3, \\ a_4 > 0, H_y = \max(H_y, H - a_4), \end{cases}$$

where a_1, a_2, a_3, a_4 — distance to the center of gravity of the reinforcement.

The value $H/2$ is used for H_x and H_y for deep beams.

The software enables to output the results of selection of longitudinal reinforcement in the form of additional reinforcement with meshes or individual bars. When meshes are used (button), it is necessary to specify the spacing of bars in a mesh, which is selected from the **Select the spacing of reinforcement** list. If bars are used in the additional reinforcement (button), the diameter of bars is selected from the **Diameter of bars of the additional reinforcement** list.

When outputting the results of the reinforcement, one of the several options can be selected in a separate list.

There are four options for plate elements

1. Calculated reinforcement
2. SNiP
3. Calculated reinforcement (according to the assortment)
4. SNiP (according to the assortment)

The second option differs from the first one in that it takes into account the standard requirements to the minimum percentage of reinforcement (the selection of reinforcement for bar elements is always performed taking into account the minimum percentage of reinforcement, but it is not clearly stated whether it should be done for plate elements).

Variants 3, 4 are similar to 1, 2, but the required intensity of reinforcement is calculated taking into account the fact that rebar diameters and reinforcement spacing can take only discrete sets of values.

Variants 1-2 and 3-4 are identical for bar elements.

Transformation of the Results of the Reinforcement Selection

When performing a check of plate elements of reinforced concrete structures (for example, in the case of analysis for progressive collapse) the longitudinal and transverse reinforcement is defined by the diameter of rebars and their spacing. This information can be prepared by the user or generated automatically by transforming the data obtained by the program of the reinforcement selection. The number of ranges of distribution of reinforcement (diameter – spacing) specified by the user for each type of longitudinal and transverse reinforcement is taken into account.

The transformation is performed with the help of the **Specify reinforcement based on the results of the reinforcement selection** operation, which can be invoked by clicking the button — in the **Selection of reinforcement** section. This operation invokes the **Specify Reinforcement Based on the Results of the Reinforcement Selection** dialog box (see Fig. 19.12-1) containing a table with the names of structural groups, maximum value of the area of each type of reinforcement, and the number of ranges the area of reinforcement of the given type has to be divided into at the transition from theoretical (selected) reinforcement to the discrete reinforcement.

The transformation can be performed for all structural groups or only for those selected from the list. Moreover, the standard requirements to the minimum percentage of reinforcement can be taken into account in the process of transformation. These settings are performed by checking the respective checkboxes.

The number of ranges is specified separately for each type of reinforcement and each structural group. The specification can be performed simultaneously for all groups (the **Number of ranges for all groups should be taken as**... checkbox is checked).

The area of longitudinal reinforcement ΔS_i of the considered group of reinforcement for each range is equal to

$$\Delta S_i = (S_{imax} - S_{imin})/n_i,$$

where

i – type of reinforcement (i = 1, 2, 3, 4);

S_{imax} and S_{imin} – maximal and minimal area of reinforcement of the i-th type, respectively;

n_i – number of ranges specified for the reinforcement of the i-th type of the considered group.

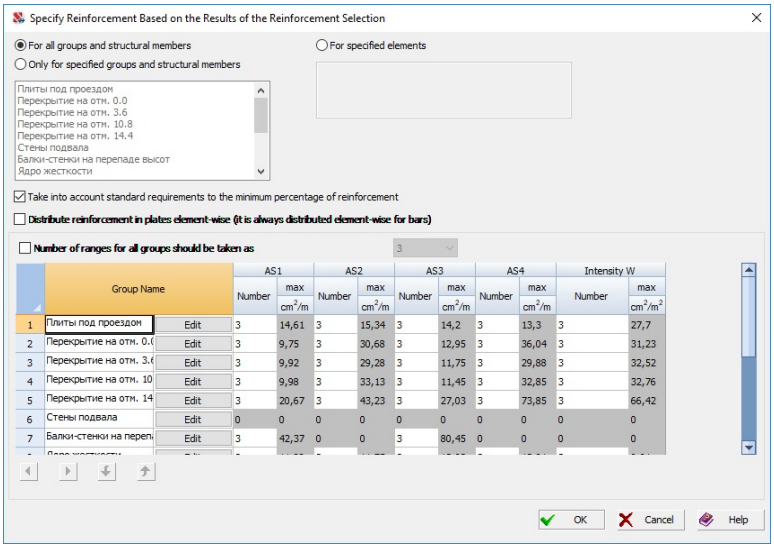


Figure 19.12-1. *The* **Specify Reinforcement Based on the Results of the Reinforcement Selection** *dialog box*

The area of transverse reinforcement (Wx, Wy) of the considered group of reinforcement for each range is determined in a similar way.

Since equal ranges do not always provide rational reinforcement (e.g., several ranges can have the same discrete reinforcement), the software enables to change the number of ranges of reinforcement of *i*-th type within one group and the area of ranges. This can be done in the dialog box (see Fig. 19.12-2), which is invoked by clicking the button on the right from the name of the considered group.

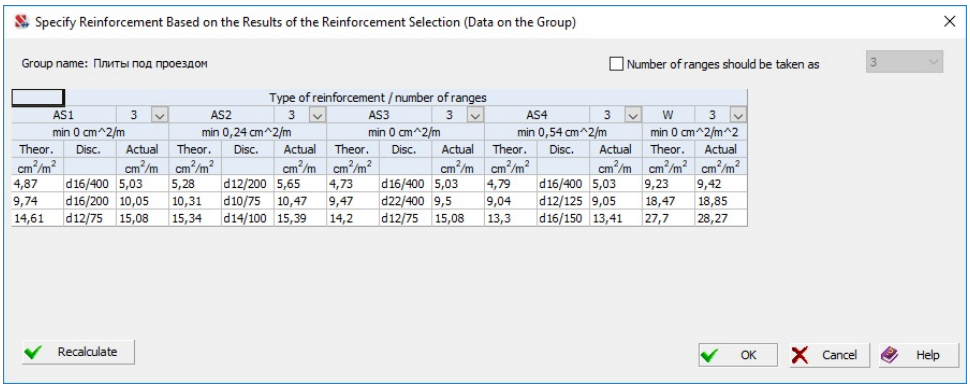


Figure 19.12-2. *The* **Specify Reinforcement Based on the Results of the Reinforcement Selection (Data on the Group)** *dialog box*

The following information is given in this dialog box on each type of reinforcement of the considered structural group:

- number of ranges;
- minimal area of theoretical (selected) reinforcement;
- maximal value of the selected reinforcement for each range;
- discrete (diameter/spacing) reinforcement for each range;
- actual area of discrete reinforcement for each range.

In those cases when discrete reinforcement (diameter/spacing) could not be selected for the maximum value of the reinforcement area of one or several ranges, the combination diameter/spacing with the maximal area and the actual (corresponding to this combination) reinforcement area are output in the respective columns of the table. The search of the diameter/spacing combination is performed on the basis of lists of allowable reinforcement spacing and diameters of reinforcement of the given class according to the criterion of minimum area. Maximum allowable diameter specified in the parameters of the considered structural group is taken into account.

If the user changes the data given in the table of the dialog box (for example, the limits of ranges), the changes are saved and the recalculation is performed after clicking the **Recalculate** button.

The value of the area the discrete reinforcement could not be selected for is highlighted in red in the table.

When transforming the reinforcement of a bar, it divided into n segments, where n is the number of sections in the bar. The lengths of the first and last (along the length of the bar L) segments are equal to $\frac{L}{2(n-1)}$, and the lengths of all other (intermediate) segments are equal to $\frac{L}{(n-1)}$. If the number of sections $n > 5$, the program does not perform the transformation (this is a limitation of the implementation). The division into ranges is not performed for bar elements and segments (in the general case) with different reinforcement are created for each finite element.

The display of the color map of the specified reinforcement of plates is performed with the help of the respective operation in the **Check of reinforced concrete** section — .

Graphical Representation of Results of the Selection of Reinforcement in Bar Elements

The results of the selection of reinforcement in bar elements are graphically displayed as diagrams of areas of longitudinal and transverse reinforcement. Diagrams can be “wire” (when they are not color filled) and they can be invoked by clicking the **Display the reinforcement diagrams of bars** button — , and color scales can be used as well — the **Color display of the reinforcement diagrams** button — .

Diagrams are output for one structural group of reinforcement, which is selected from the list in the **Selection of reinforcement** section of the toolbar.

Figure 19.12-3 shows a drop-down list with the types of information which can be graphically visualized after performing the selection of reinforcement in bar elements.

The main rules for plotting the reinforcement diagrams are as follows:

- a bar is divided into *n* segments, where *n* is equal to the number of sections in which the forces were calculated (the rule for dividing a bar into segments is described above);
- if a bar has rigid inserts, they are displayed automatically in the mode of analysis of results of the reinforcement selection;
- diagrams are plotted for a bar segment only if the reinforcement is selected in this segment (here "selected" means that the area of reinforcement $S_i \geq 0$);
- if the reinforcement (at least in one direction) could not be selected, the respective segment of the bar is highlighted in red;
- the position of reinforcement in the section which enables to meet the standard requirements to the load-bearing capacity can be viewed for each bar separately in the **Information on the element** mode (the **Reinforcement** button);
- the value of the area of reinforcement is displayed on the diagram after clicking the **Digitization** button in the **Visualization of Results** toolbar. It is recommended to perform the digitization on the "wire" diagrams.

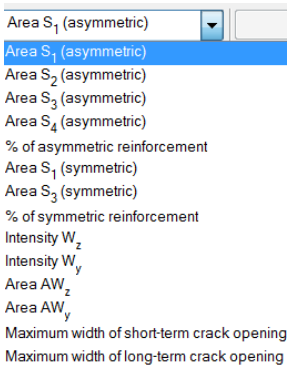


Figure 19.12-3. *List of results of the selection of reinforcement in bars*

Reinforcement diagrams are output in the $\mathbf{x_1 0 z_1}$ plane of the local coordinate system of the bar by default. If the angle of orientation of structural axes has been changed for a bar, the degeneration of the display of the diagrams on the screen can take place. In this case you can use the **Change the display axis of diagrams** operation (diagrams will be output in the $\mathbf{x_1 0 y_1}$ plane).

Graphical Representation of Results of the Selection of Reinforcement in Plate Elements

As noted above, the output of intensity (area per running meter) of reinforcement in the form of isofields or color maps (mosaics) is provided by default for plate elements. You can also output the area of transverse reinforcement for the specified spacing, which is selected from the drop-down list of the toolbar. The following operations using color scales are provided:

— **Display isofields of the reinforcement of plates**. Isofields belonging to the adjacent intervals of the considered factor are not divided by isolines. A standard color scale with the possibility of specifying the number and limits of the intervals is used (limits of the intervals can be changed in the **Settings** dialog box which is invoked by clicking the **Color scale settings** button —). Moreover, the dialog box with a color scale includes a table with the characteristics of concrete and reinforcement of the considered group.

— **Display isolines and isofields of the reinforcement of plates**. The display is performed according to the same rules as in the previous case, but the isofields belonging to the adjacent intervals of the considered factor are divided by isolines, the color of which is specified in the **Color Palette Settings** section of the ***Settings*** menu.

— **Display the color map of the reinforcement results**. This type of display differs in that each finite element is fully displayed in the color corresponding to the color of the interval the value of the considered factor on the color scale belongs to. The color scale and its control principles are similar to those considered above.

Figure 19.12-4 shows a drop-down list with the types of information which can be graphically visualized after performing the selection of reinforcement in plate elements.

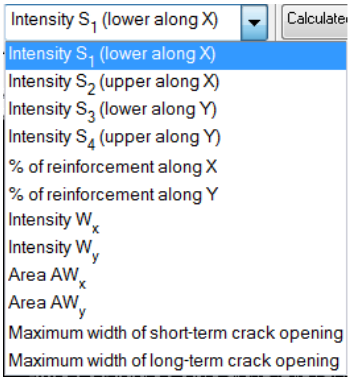


Figure 19.12-4. *List of results of the selection of reinforcement in plates*

In addition to the intensity of reinforcement the above operations and color scales are used for the visualization of the results of the selection of longitudinal reinforcement given as sets of rebars with the specified spacing in the directions coinciding with the directions of the force output. This operation is invoked with the button — **Switch the output form of the reinforcement results**, and the reinforcement spacing is selected from the respective list in the toolbar. In this case at the digitization the data will be output in the format of type *d12/100* (i.e. it is recommended to use the reinforcement with a diameter of 12 mm and a spacing of 100 mm).

The results of the selection of longitudinal reinforcement can be given in the form of additional reinforcement with meshes or bars. In this case the left column of the table of the color scale contains the following data:

- the first line of the table contains the diameter and spacing of the background reinforcement;
- the subsequent lines contain the diameter and spacing of the additional reinforcement marked with "+" for each interval; these values of additional reinforcement for each interval should be considered as a ***supplement to the background reinforcement***;
- if the additional reinforcement could not be selected for the specified diameter or spacing of reinforcement, the current and subsequent lines of the table will contain the values of intensity of the additional reinforcement.

The right column of each interval contains a value of intensity of reinforcement corresponding to the larger limit of the current interval (this value was taken into account in the determination of the discrete reinforcement).

Depending on the type of additional reinforcement (with meshes or with bars), the spacing or the diameter of reinforcement is saved. The selected value is output in the dialog box of the color scale above the table.

The setting of the scale of additional reinforcement is performed by clicking the button of the dialog box of the scale. The **Settings** dialog box will appear (Fig. 19.12-5).

The **Structural reinforcement** button invokes the dialog box (Fig. 19.12-6), where you can specify the intensity of structural (background) reinforcement in the text field or select the necessary combination

Diameter/spacing from the drop-down list (the corresponding intensity will be automatically transferred into the text field).

Drop-down lists for selecting a value of supplements to the background reinforcement are also provided for each interval.

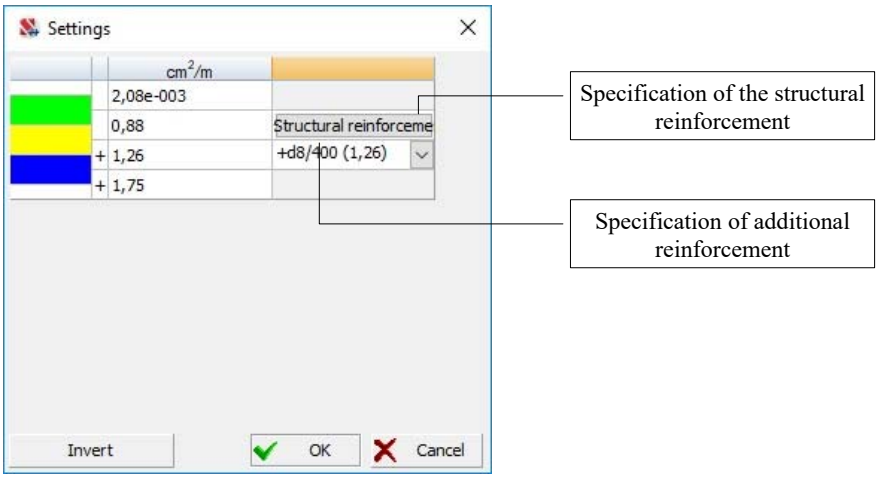


Figure 19.12-5. *The* **Settings** *dialog box for the additional reinforcement*

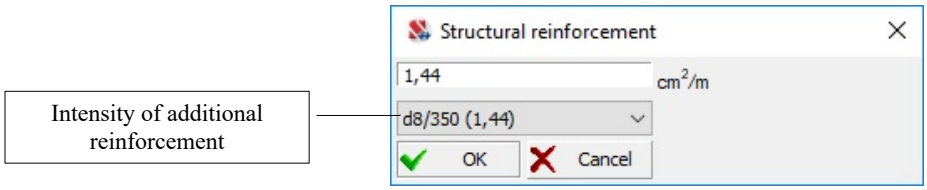


Figure 19.12-6. *The* **Structural Reinforcement** *dialog box*

Export of the Results of Selection of Reinforcement in a Plate to the Design Subsystem of ALLPLAN

This operation enables to transfer the data on the results of selection of reinforcement in the elements of foundation slabs or floor slabs to the design subsystem of **ALLPLAN**. A file with the extension **.asf** is generated in the result of the export. This file contains information on the geometry of the reinforced segment, and also on the areas of the upper and lower reinforcement (according to the assortment). It also contains the characteristics of the elements included in the active group of reinforcement (if the group has selected elements, the export of information will be performed only for them). Clicking the arrow will open a menu where you can select type of areas - calculated or calculated (according to the assortment).

Generating a Report

The operation of printing tables with the results of the selection of reinforcement invokes the dialog box (Fig. 19.12-7). The tables are generated in the **.rtf** format and are automatically loaded into a text editor associated with this format.

Groups of radio buttons and checkboxes in this dialog box enable to control the generation of tables by specifying the following options:

- **For all groups and structural members** — output the results for all groups;
- **Only for specified groups and structural members** — results will be output only for groups selected in the list of groups;
- **For specified elements** — results will be output only for finite elements the numbers of which are listed in the text field;

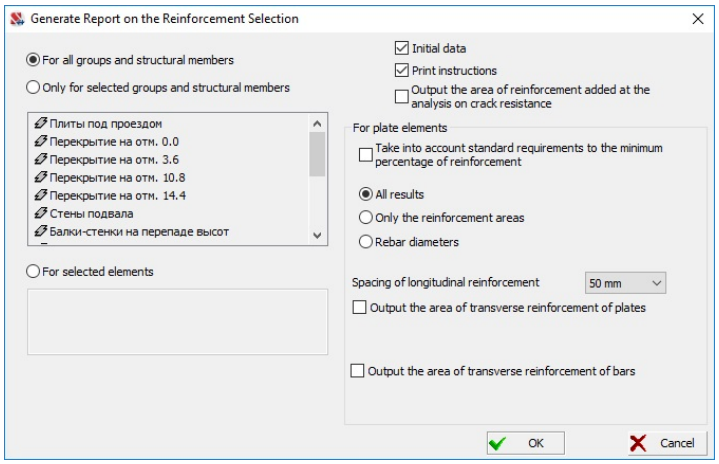


Figure 19.12-7. *The* **Generate Report on the Reinforcement Selection** *dialog box*

- **Print instructions** — output the information on the rules for reading the results after the tables;
- **Output the area of reinforcement added at the analysis on crack resistance**;
- **Take into account standard requirements to the minimum percentage of reinforcement** — if this checkbox is checked, the area per running meter of reinforcement in plates will be (when necessary) increased to satisfy the standard requirement to the minimum percentage of reinforcement;
- **Only the reinforcement areas** — print tables only with the values of the total reinforcement area;
- **Rebar diameters** — print tables with the data on diameters, spacing and number of rebars;
- **All results** — print tables with the values of the reinforcement areas, and data on the diameters, spacing and number of rebars;
- **Spacing of longitudinal reinforcement of plates** — if the report on the results of the reinforcement selection has to include not only the information on the areas per running meter of the longitudinal reinforcement of plates, but also the presentation of this information in the **Diameter/spacing** format, it is necessary to select the spacing of longitudinal reinforcement the diameters of longitudinal rebars will be output for;
- **Output the area of transverse reinforcement of plates** — if this checkbox is not checked, the data on the selected transverse reinforcement in plates will be output as the intensity (cm^2/m); otherwise it will be necessary to select a spacing from the **Spacing of transverse reinforcement of plates (out-of-plane)** drop-down list, and the report will contain the required area (cm^2) of the transverse reinforcement at the selected spacing;
- **Output the area of transverse reinforcement of bars** — if this checkbox is not checked, the data on the selected transverse reinforcement in bars will be output as the intensity (cm^2/m); otherwise it will be necessary to select a spacing from the **Spacing of transverse reinforcement of bars** list, and the report will contain the required area (cm^2) of the transverse reinforcement at the selected spacing.

19.13. Reading the Results of the Reinforcement Selection

Bar Elements

Information for each section of the element is output in several lines in the table with the results of the calculation (Table 19.13-1).

Results of the selection of reinforcement in bar elements **Table 19.13-1**

No. of the element	Section	Type	Longitudinal reinforcement area in cm^2 diameters (∅) in mm spacing (S) in mm								Crack opening width (mm)		Transverse reinforcement area in cm^2	
			Asymmetric					Symmetric					Spacing 50 mm	
			S_1	S_2	S_3	S_4	%	S_1	S_3	%	short-term	long-term	AW_z	AW_y
	b = 40 cm h = 60 cm a_1 = 5 cm a_2 = 5 cm													
11	1	Σ	2,56	22,22	0,21	0,21	1,145	21,61	0,41	2,002	0,298	0,293	0,23	0,1
		C		6,36				5,74						
		∅	2∅14	3∅32				3∅32						
	2	Σ	2,56	12,54	0,21	0,21	0,705	10,66	0,41	1,006			0,1	0,1
		C		2,9				1,01						
		∅	2∅14	3∅25				2∅28						
	3	Σ	2,56	5,09	0,21	0,21	0,366	5,09	0,41	0,5			0,1	0,1
		∅	2∅14	3∅16				3∅16						

For groups of reinforcement of bar elements the information for each element (or a unified group of elements) is output in several lines in the table with the results of the calculation. Each line of the ***Type*** column contains the following symbols indicating the type of data in this line:

Σ — area of reinforcement;

T — area of reinforcement required to resist the torque (included in **Σ**);

C — area of reinforcement added to provide crack resistance (included in **Σ**);

∅ — presentation of the selected area of reinforcement in the discrete form.

In lines the icons of which include symbol **∅**, the results are given as N**∅**D, where N is the number of bars, D is diameter of one bar. If the assortment of the rebar diameters is exhausted, the respective cells of the table will contain N**∅**A, where N is the number of bars, A is the cross-sectional area of one bar.

If there was no analysis on crack resistance, or the reinforcement selected according to the strength requirements provides the crack resistance of the section, the lines indicated by the icon **C** are not output.

If a section has such a shape that it can be reinforced both symmetrically and asymmetrically, the report contains information on two variants of the calculation.

The maximum spacing of the clamps is also given for the transverse reinforcement.

In the calculation results the area of the transverse reinforcement resisting the torque is output as a value calculated for two bars in the section of the element. Thus, the area of one bar can be determined as AW/2.

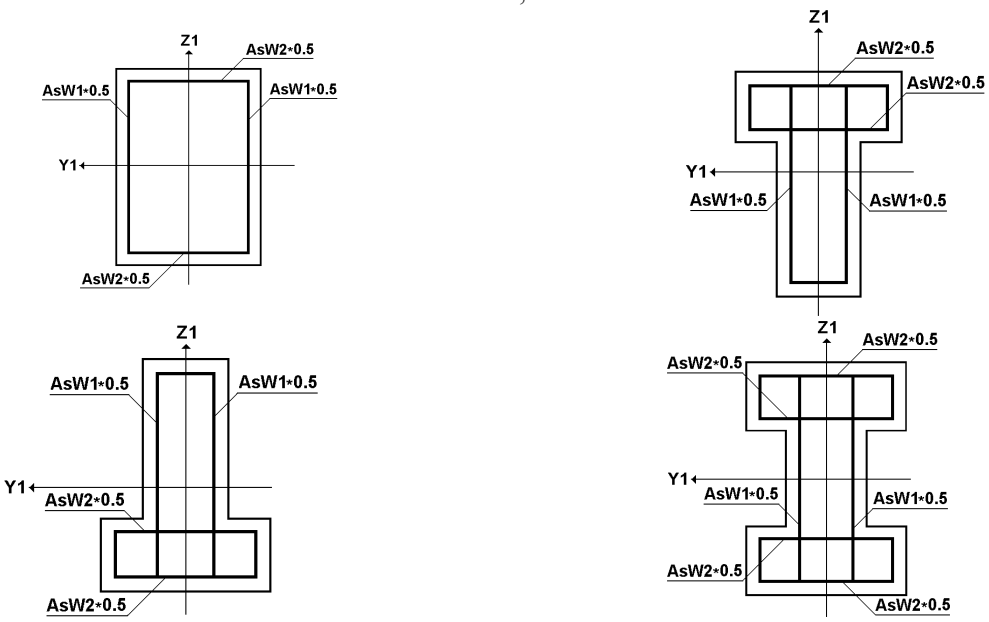


Рис. 19.13-1. *Results for transverse reinforcement for the bars*

Plate Elements

For groups of reinforcement of plate elements the information for each element (or a unified group of elements) is output in several lines in the table with the results of the calculation. Each line of the ***Type*** column contains the following symbols indicating the type of data in this line:

∑ — reinforcement area per one running meter;

C — reinforcement area per one running meter added to provide crack resistance (included in **∑**);

∅/S — presentation of the selected reinforcement as a set of diameters and spacing of reinforcement.

If the analysis on crack resistance is not performed, or the reinforcement selected for the ultimate limit state provides the required crack resistance, the lines with the **C** type are not output.

Cross-sectional area of reinforcement for each plate finite element (or a unified group of finite elements) is determined for sections with the width of 1 m at the specified thickness of the element in accordance with the design combinations of forces.

In lines with type **∅/S** the results are given as ***∅D/S***, where ***D*** is the diameter of one bar, ***S*** is the spacing of bars in millimetres. If the assortment of the rebar diameters is exhausted for the given spacing, the respective cells of the table will contain the value of the area of reinforcement.

The intensity per running meter of the selected transverse reinforcement is output at the out-of-plane spacing of 100 mm.

Table 19.13-2

Results of the selection of reinforcement in plate elements

No. of the element	Type	Longitudinal reinforcement intensity in cm²/m diameters (∅) in mm spacing (S) in mm						Crack opening width (mm)		Transverse reinforcement area in cm² diameters (∅) in mm	
		Along X			Along Y					Out-of-plane spacing 100 mm	
		S_1	S_2	%	S_3	S_4	%	short-term	long-term	AW_x	AW_y
9250	∑	12,04	12,72	0,583	3,4	21,55	0,554	0,298	0,298		1,01
	∅/S	12,04	12,72	0,599	∅7/100	21,55	0,586				∅10/75
	C	0,36									

19.14. Check of the Specified Reinforcement

The check of the load-bearing capacity of structural members and/or groups of structural members is performed by clicking the **Calculation** button — . This button has an arrow which besides the full calculation for all structural groups enables to perform the check only for those groups that have been modified, to perform the calculation for user' selected groups only (in this case dialog box will appear and user must select groups for calculation), to invoke the operations of the data control (without performing the calculation), view messages that were output during the previous calculation or control, and delete results. Regardless of whether changes have been made in all structural groups or only in some of them, the control of the initial data is performed for all groups and diagnostic messages will be generated for all groups, and not just for those which were recalculated. The check is performed for all sections of finite elements included in a structural member or a group of structural members on the basis of the *specified* reinforcement. The specified reinforcement can be obtained by transforming the reinforcement selected with the help of the program (see the description of the operation above), or defined by the user. In the latter case the **Specify the reinforcement layout of plate elements** and **Specify the reinforcement layout of bars** operations are used. In order to perform them the elements must belong to structural groups, since all the other data used when performing a check (class of concrete and reinforcement, distance to the center of gravity of reinforcement etc.) are taken from the description of structural groups the elements belong to.

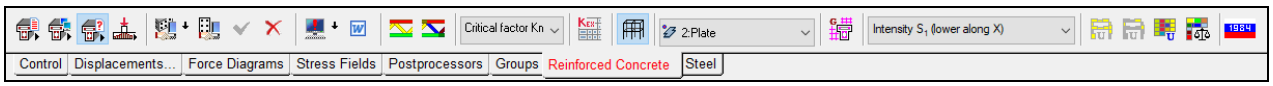


Figure 19.14-1. *The toolbar for the calculation of the load-bearing capacity of reinforced concrete elements and the analysis of results*

The toolbar (Fig. 19.14-1) includes the buttons for invoking the following operations:

— specify the reinforcement layout of plate elements;

— specify the reinforcement layout of bars;

— input data and save the active group;

— reset the selected group;

— invoke the calculation (check);

— generate report;

— display results of the check on the model (3-color scale);

— display results of the check on the model (full color scale);

— display the design model;

— extreme values of factors;

— show active group (here pink color is used both in the image of the icon and in the model unlike the group selection button where the red color is used);

— display diagrams of the specified reinforcement of bars;

— color display of diagrams of the specified reinforcement;

— display the color map of the specified reinforcement of plates;

— weight of the specified reinforcement;

— select the design codes.

The maximum value of each factor defining the load-bearing capacity of the element is obtained in the result of the check.

Specifying the Reinforcement Layout of Plate Elements

This operation invokes the **Reinforcement of Plates** dialog box (Fig. 19.14-2) where you can specify the reinforcement layout of plate elements. The diameter of reinforcement is selected from the drop-down list, the spacing of longitudinal and transverse reinforcement is specified in the respective line. If there is no reinforcement, check the checkbox in the respective line.

If the reinforcement of some elements has been specified earlier, or obtained by an automatic transformation of the results of the selection, the table with the variants of reinforcement will contain all the used layouts. In this case, the specification can be performed by selecting a reinforcement layout in the table (the data in the selected line are transferred to the respective fields of the table in the upper part of the dialog box). Working with these data is similar to, for example, working with the characteristics of rigidity, i.e. if you close the dialog box by clicking the **OK** button without changing the contents of the table, the selected elements in the model will be assigned the reinforcement layout used earlier. If the table has been modified and the **Replace and exit** or **Replace and continue** buttons have been used, the changes will affect only the given layouts and their corresponding elements. Finally, if you close the dialog box by clicking the **OK** button after modifying a previously used reinforcement layout, the modified reinforcement layout will be assigned to the selected elements.

In order to assign the specified reinforcement to the elements of the model, close the dialog box by clicking the **OK** button, select the elements in the model and click the **OK** button — in the toolbar.

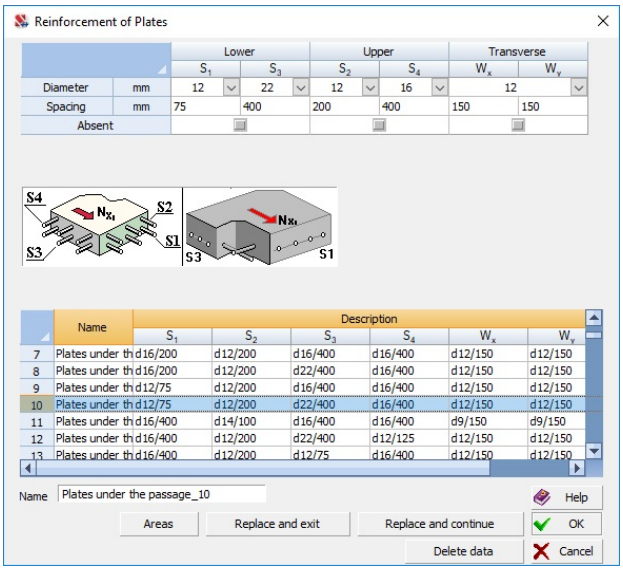


Figure 19.14-2. *The* **Reinforcement of Plates** *dialog box*

The **Delete data** button allows to delete information about reinforcement for the selected elements.

The **Areas** button in the **Reinforcement of Plates** dialog box enables to obtain information on the areas of reinforcement per running meter which correspond to the specified diameters and spacing (Fig. 19.14-3).

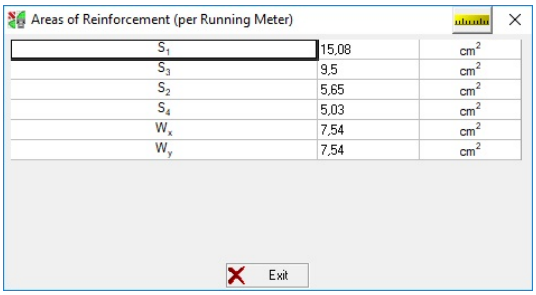


Figure 19.14-3. *The* **Areas of Reinforcement** *dialog box*

The **Specify the reinforcement layout of plate elements** operation provides two type of actions — the specification of the layout and its editing. The action can be selected in the menu which is dropped down by clicking the arrow on the right from the **Specify the reinforcement layout of plate elements** button.

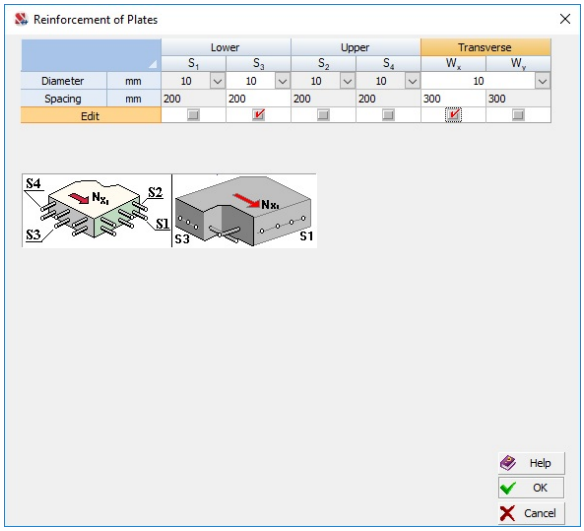


Figure 19.14-4. *Editing of the reinforcement layout*

The specified values of the diameter and/or spacing can be changed for the selected elements only for the specified types of reinforcement. Other parameters of the reinforcement remain unchanged.

In order to perform this operation you have to:

- drop down the menu of the **Specify the reinforcement layout of plate elements** operation by clicking the arrow and select the **Edit the reinforcement layout …** item;
- the **Reinforcement of Plates** dialog box will appear (Fig. 19.14-2) where you can specify the types of reinforcement which have to be edited by checking the respective checkboxes;
- specify the necessary data in the respective columns of the table of reinforcement and click the **OK** button;
- select the elements in the model for which the reinforcement layout is modified, and click the **Confirmation** button in the toolbar.

The specified reinforcement will be replaced in the selected elements.

Specifying the Reinforcement Layout of Bar Elements

This operation invokes the dialog box (Fig. 19.14-5), where you can specify the reinforcement of segments of bar elements. The number of segments is selected in the respective drop-down list and can be not equal to the number of sections DCF were generated for. In any case the check of reinforcement for each element will be performed taking into account all DCF obtained for it.

The lengths of segments are specified as a percentage of the length of the element. The reinforcement layout of each segment is defined by the diameter of reinforcement and the number of rebars of this diameter for the lower, upper and side reinforcement.

The presence of a certain type of reinforcement in the element is specified by checking the respective checkboxes and it is assumed to be the same for all segments. In those cases when a certain type of reinforcement is absent, for example, the second row of the upper reinforcement in the central segment of the element, the number of rebars of this type is taken as zero.

If the **Reinforcement in two rows** checkbox is checked, the **Longitudinal reinforcement (second row)** table appears in the window. It is used to specify the diameter and the number of rebars in the second row of the lower and upper longitudinal reinforcement, and the clear distance between the first and second rows of the lower (Δ_1) and upper (Δ_2) reinforcement. If the clear distance is set to zero, then the software takes it as the minimum design spacing between reinforcing bars which are placed vertically during concreting.

Limitations

For circular and annular cross-sections, the second row of reinforcement is ignored.

If the reinforcement of some elements has been specified earlier, or obtained by an automatic transformation of the results of the selection, the table with the variants of reinforcement will contain all the used layouts. In this case, the specification can be performed by selecting a reinforcement layout in the table (the data in the selected line are transferred to the respective fields of the table in the upper part of the dialog box). Working with these data is similar to, for example, working with the characteristics of rigidity, i.e. if you close the dialog box by clicking the **OK** button without changing the contents of the table, the selected elements in the model will be assigned the reinforcement layout used earlier. If the table has been modified and the **Replace and exit** or **Replace and continue** buttons have been used, the changes will affect only the given layouts and their corresponding elements. Finally, if you close the dialog box by clicking the **OK** button after modifying a previously used reinforcement layout, the modified reinforcement layout will be assigned to the selected elements.

Record in the column of the table, for example, S1 — **1: 2d20 2:3d22 3:2d14** means that for this bar the lower reinforcement S1 is specified for segment 1 as 2 rebars with the diameter of 20 mm, for segment 2 — as 3 rebars with the diameter of 22 mm and for segment 3 — as 2 rebars with the diameter of 14 mm.

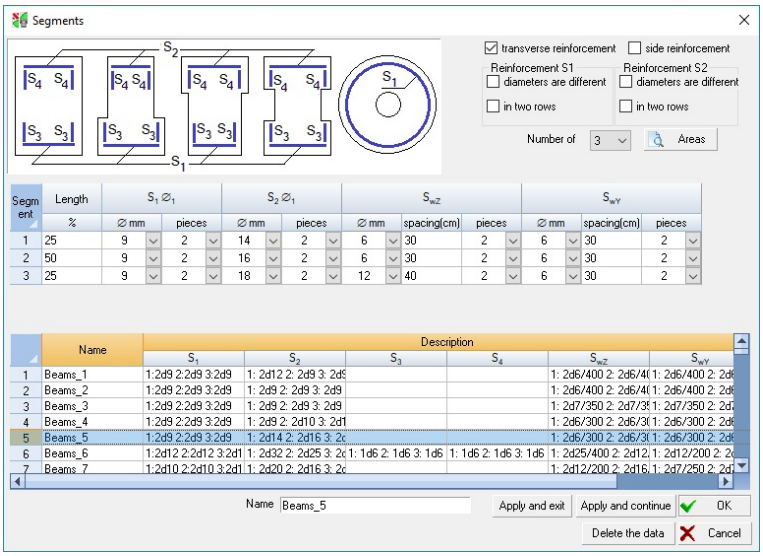


Figure 19.14-5. *The* **Segments** *dialog box*

In order to assign the specified reinforcement to the elements of the model, close the dialog box by clicking the **OK** button, select the elements in the model and click the **OK** button — in the toolbar.

Clicking the **Areas** button enables to obtain the information on the areas of reinforcement of different type, which is specified for each of the segments of the bar. The **Areas of Reinforcement** dialog box (Fig. 19.14-6) will appear which contains the data on the areas of reinforcement, as well as the reinforcement layout for each segment of the bar. Since the variant of reinforcement selected in the table of the **Segments** dialog box can belong to the bars with different sections, the distribution of the rebars in the sections of the element is given for a section with conventional sizes and shape selected from the set of section shapes in the lower part of the dialog box. Similar information but taking into account the real section of the element is available from the information on the element.

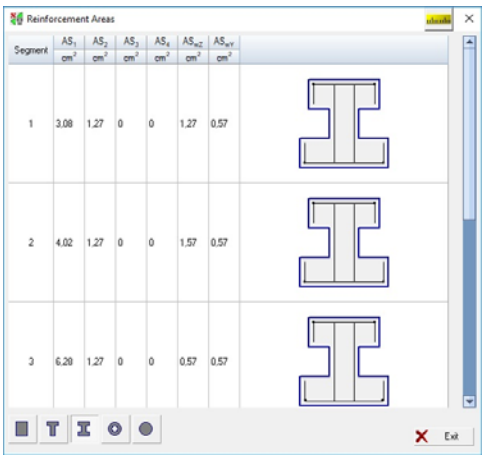


Figure 19.14-6. *The* **Areas of Reinforcement** *dialog box*

Graphical Representation of the Specified Reinforcement in Bar Elements

The specified reinforcement in bar elements is graphically displayed as diagrams of areas of longitudinal and transverse reinforcement. Diagrams can be “wire” (when they are not color filled) and they can be invoked by clicking the **Display diagrams of the specified reinforcement of bars** button —

, and color scales can be used as well — the **Color display of diagrams of the specified reinforcement** button — . Diagrams are output for the active structural group selected from the list in the toolbar, and the type of reinforcement selected by the user. The display is similar to the display of diagrams according to the results of the reinforcement selection described above.

Display of the Color Map of the Specified Reinforcement in Plate Elements

A color map of the display of the specified reinforcement of plates is output in a similar way as a map with the results of the reinforcement selection.

Display Results of the Check

The results of the check of the load-bearing capacity are given as the utilization factors of restrictions and can be displayed on the model simultaneously for all structural groups.

There are two types of the display of the results of the check — as a 3-color and a full color scale. In the first case the **Display results of the check on the model (3-color scale)** operation — is used for the display. When it is performed, the elements included in structural groups are displayed in three colors:

- green — the load-bearing capacity is provided;
- red — the load-bearing capacity is insufficient;
- yellow — the value of the factor of restrictions is close to the critical value (the factor is close to 1,0).

The critical range is specified by setting the color scale in the **Settings** dialog box which is invoked by clicking the respective button — .

For example, if you specify the value 0,03 in the dialog box (Fig. 19.14-7), all elements with the value of the factor less than 0.97 will be displayed in green, if the factor is greater than 1,03 — in red, and if the factor is in the range [0,97;1,03] — yellow will be used.

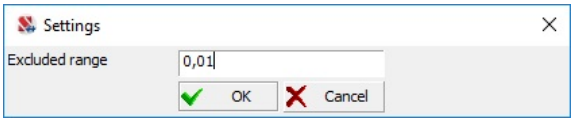


Figure 19.14-7. *The* **Settings** *dialog box*

If a display mode in the form of a full color scale is used (the **Display results of the check on the model (full color scale)** button —), an ordinary color scale with a custom number of colors will appear.

If not a single group of structural members is selected in the drop-down list, the visualization of the critical factor K_{max} is performed. If a particular group is selected, the list of factors will contain the full list of performed checks and you will be able to display on the model the results of the check for each factor defining the load-bearing capacity of the considered structures (for example, strength or crack opening width). The values of the selected factor can be output on the model using the filters button **Digitization** — .

Extreme Factor Values

This operation enables to generate a table (Fig. 19.14-8) for the selected group of structural members or structural members which includes the following information:

- list of factors;
- minimum and maximum value of each factor;

- numbers of finite elements where these minimum and maximum values took place.

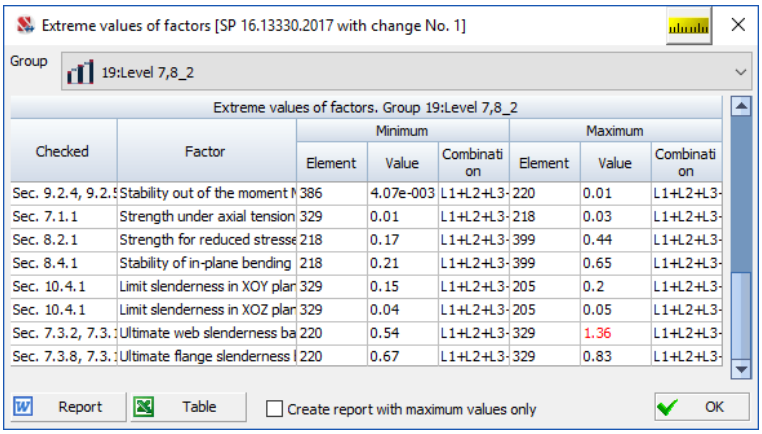


Figure 19.14-8. *The* **Extreme Factor Values** *dialog box*

The groups or structural members can be selected from the list in the upper part of the dialog box. The **Report** and **Table** buttons enable to obtain report documents for the selected group in the respective formats. Special checkbox allow to create report with maximum values only.

Generating a Report

A report can be generated on the basis of the results of the check. It is generated in the **RTF** format and is automatically loaded into an application associated with this format (for example, MS Word or WordPad). The dialog box (Fig. 19.14-9) appears before generating the report, where you have to select structural members and/or groups of structural members, information on which has to be included in the report.

The dialog box provides the following options of the output control:

- **For all groups and structural members** — the report will include all structural groups from the list;
- **Only for specified groups and structural members** — the report will contain only the information on the groups selected in the list;
- **For specified elements** — in this case you have to specify the numbers of finite elements for which the report will be generated in the text field.

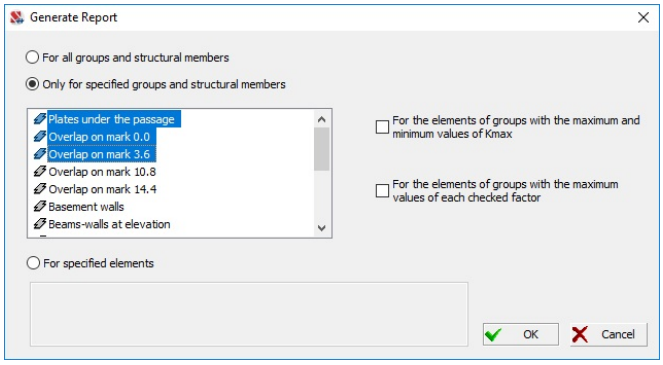


Figure 19.14-9. *The* **Generate Report** *dialog box*

Moreover, the **For the elements of groups with the maximum and minimum values of K_{max}** and **For the elements of groups with the maximum values of each checked factor** checkboxes enable to obtain a report only for the elements with the maximum values of the factors.

Information on the Element – Analysis of Reinforced Concrete Structures

The **Specified reinforcement**, **Reinforcement**, **Reinforced concrete. Check** and **Reinforced concrete. Select** operations, which can be invoked from the **Information on the Element** dialog box (button in the filters toolbar), provide additional possibilities for the analysis of the load-bearing capacity of the elements of reinforced concrete structures. These operations are implemented according to the same rules as the respective operations of **ARBAT** [9];

Specified reinforcement — reinforcement of the considered element is specified in the dialog box according to the rules described in the current chapter in sections **Specifying the Reinforcement Layout of Plate Elements** and **Specifying the Reinforcement Layout of Bar Elements**. When performing the check these data will be taken into account in the analysis of the load-bearing capacity of this element;

Reinforcement — the results of the selection of reinforcement in the considered element for plates (Fig. 19.14-10) and bars (Fig. 19.14-11) are given in the **Selection of Reinforcement** dialog boxes;

Reinforced concrete. Check — the operation similar to the **Strength of Sections** operation of **ARBAT** is performed. The initial data for performing the check include the information on the finite element (geometric parameters etc.), and the parameters of a structural group the considered element belongs to. Forces and stresses in the respective tab correspond to DCF of this element (if the combination includes a special (non-seismic) loading, then the characteristic and characteristic long-term forces/stresses will automatically be assumed equal to zero, since the serviceability limit state analysis does not have to be performed in this case). If a bar element is a part of a structural member, the length of the structural member will be used. If a bar element is selected with the specified reinforcement different in different segments, clicking the button will drop down a menu where you have to select the necessary segment. You can change different parameters, for example, the service factor, class of concrete and reinforcement, etc., during the check, which enables to analyze their effect on the load-bearing capacity of the element. When working with bar elements, the button in the *Reinforcement* tab enables to save the reinforcement given in this tab as *specified* for the selected element (you will have to specify the name of this reinforcement). The specified reinforcement will be assigned to the entire element (i.e., it is assumed that there is only one section and the reinforcement is the same for the entire element). Another button enables to select previously specified reinforcement (the list is limited to the set that was used for bars with one section and with the same type of rigidity as the selected element);

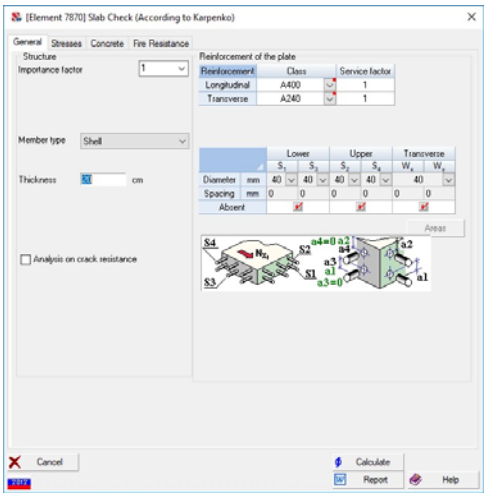


Figure 19.14-10. *The* **Selection of Reinforcement** *dialog box* for plate elements

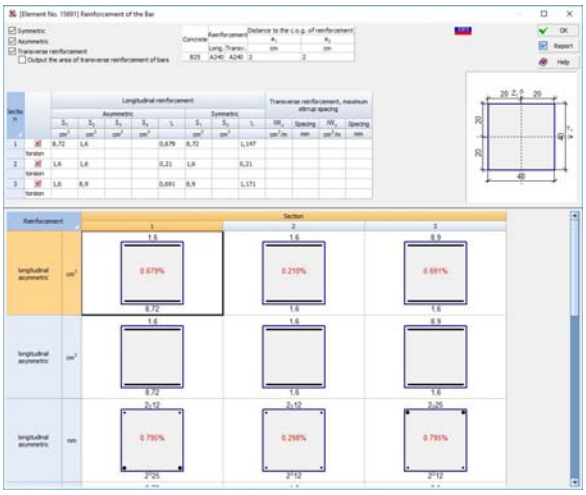


Figure 19.14-11. *The* **Reinforcement of the Bar** *dialog box* for bar elements

Reinforced concrete. Select — selection of reinforcement in the considered element by invoking the programs of the reinforcement selection. Here, like in the case of the check, changes of different parameters are allowed;

Reinforced concrete. Factors — a diagram of factors including all values of the utilization factor of restrictions obtained in the result of the check is plotted for the considered element. The **Show forces** checkbox enables to get an alternative presentation of information showing forces/stresses in an element.

Specified Reinforcement Filter

The respective filter in the **Display Filters** toolbar is used for the color display of the specified reinforcement in the form of discrete reinforcement. Information in the color scale is presented in the following way:

- longitudinal reinforcement of bars — $S_k = nd$ or $n_1d_1 + n_2d_2$ — in the case of the reinforcement arranged in two rows (only for the reinforcement S_1 and S_2),
  where: S_k – type of reinforcement in the section (k = 1, 2, 3, 4),
  n — number of rebars of the given type for the reinforcement arranged in one row,
  n_1 and n_2 — number of rebars of the given type in the first (nearest to the concrete cover) and the second rows of reinforcement respectively,
  d — diameter of rebars for the reinforcement arranged in one row,
  d_1 and d_2 — diameter of rebars for the reinforcement arranged in two rows;
- transverse reinforcement of bars — $Swz/y = nd/s$,

where: Swz/y – transverse reinforcement along the respective axis,
n – number of rebars,
d – diameter of rebars,
s – spacing of stirrups.

For example, the record S1 = 3d16, S2 = 3d32, S3 = 2d12, S4 = 2d12, Swz = 2d12/125, Swy = 2d12/200 for a rectangular section means that in the considered section:
lower reinforcement — 3 bars with the diameter of 16 mm,
upper reinforcement — 3 bars with the diameter of 32 mm,
side reinforcement — 2 bars with the diameter of 12 mm along each side,
transverse reinforcement (stirrups) along the Z_1 axis — two bars with the diameter of 12 mm with the spacing of stirrups of 125 mm,
transverse reinforcement (stirrups) along the Y_1 axis — two bars with the diameter of 12 mm with the spacing of stirrups of 200 mm.

- longitudinal reinforcement of plates — $S_k = nd$,

where: S_k – type of reinforcement in the section (k = 1, 2, 3, 4),
n – number of rebars of the given type,
d – diameter of rebars,

- transverse reinforcement of plates — $W_{x(y)} = d/s$,

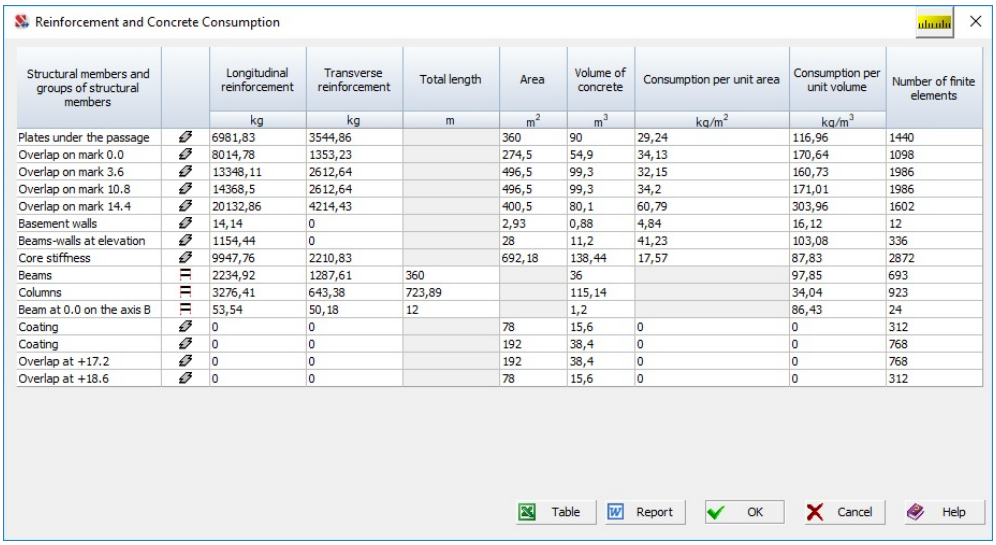
where: $W_{x(y)}$ – transverse reinforcement along the respective axis,
d – diameter of rebars,
s – spacing in the respective direction.

For example, the record S1 = d16/100, S2 = d32/150, S3 = d12/100, S4 = d12/200, Wx = d12/100, Wy = d12/150 means that:
lower reinforcement along the X axis of the direction of the force output — bars with the diameter of 16 mm with the spacing of 100 mm,
upper reinforcement along the X axis of the direction of the force output — bars with the diameter of 32 mm with the spacing of 150 mm,
lower reinforcement along the Y axis of the direction of the force output — bars with the diameter of 12 mm with the spacing of 100 mm,
upper reinforcement along the Y axis of the direction of the force output — bars with the diameter of 12 mm with the spacing of 200 mm,
transverse reinforcement along the X axis of the direction of the force output — bars with the diameter of 12 mm with the spacing of 100 mm,

transverse reinforcement along the Y axis of the direction of the force output — bars with the diameter of 12 mm with the spacing of 150 mm.

Switching the color scale for the display of the specified reinforcement of bars or plates is performed with the help of the respective buttons.

19.15. Weight of the Specified Reinforcement



Reinforcement and Concrete Consumption

Structural members and groups of structural members		Longitudinal reinforcement	Transverse reinforcement	Total length	Area	Volume of concrete	Consumption per unit area	Consumption per unit volume	Number of finite elements
		kg	kg	m	m^2	m^3	kg/m^2	kg/m^3	
Plates under the passage		6981,83	3544,86		360	90	29,24	116,96	1440
Overlap on mark 0.0		8014,78	1353,23		274,5	54,9	34,13	170,64	1098
Overlap on mark 3.6		13348,11	2612,64		496,5	99,3	32,15	160,73	1986
Overlap on mark 10.8		14368,5	2612,64		496,5	99,3	34,2	171,01	1986
Overlap on mark 14.4		20132,86	4214,43		400,5	80,1	60,79	303,96	1602
Basement walls		14,14	0		2,93	0,88	4,84	16,12	12
Beams-walls at elevation		1154,44	0		28	11,2	41,23	103,08	336
Core stiffness		9947,76	2210,83		692,18	138,44	17,57	87,83	2872
Beams		2234,92	1287,61	360		36		97,85	693
Columns		3276,41	643,38	723,89		115,14		34,04	923
Beam at 0.0 on the axis B		53,54	50,18	12		1,2		86,43	24
Coating		0	0		78	15,6	0	0	312
Coating		0	0		192	38,4	0	0	768
Overlap at +17.2		0	0		192	38,4	0	0	768
Overlap at +18.6		0	0		78	15,6	0	0	312

Table | Report | OK | Cancel | Help

Figure 19.15-1. *The* **Reinforcement and Concrete Consumption** *dialog box*

This operation enables to obtain information on the consumption of concrete and the specified reinforcement (longitudinal and transverse) for each structural group (Fig. 19.15-1). Moreover, it also provides data on the reinforcement consumption per unit volume and per unit area (for groups of plate elements), total length of the flexible parts of bar finite elements and the number of elements in the groups. The **Report** button invokes the generation of a report document.

19.16. Basic Principles of the Selection of Reinforcement in the Finite Elements of a Slab and Shell

Selection of the Type of Reinforcement

Following [3], first of all consider the conditions of the possible formation of cracks in the slab, which the need for reinforcement. The analysis is performed on an example of studying the behavior of some small in plan characteristic elements with cracks extracted from the structure. Since there are no fundamental differences between the performance of a small element of the slab and that of the shell (the curvature of the shell has no effect within the small area), we will consider flat plates taking both bending and membrane groups of forces, i.e. the combined action of all three moments (bending and torque) and three forces (normal and shear) on the elements is considered.

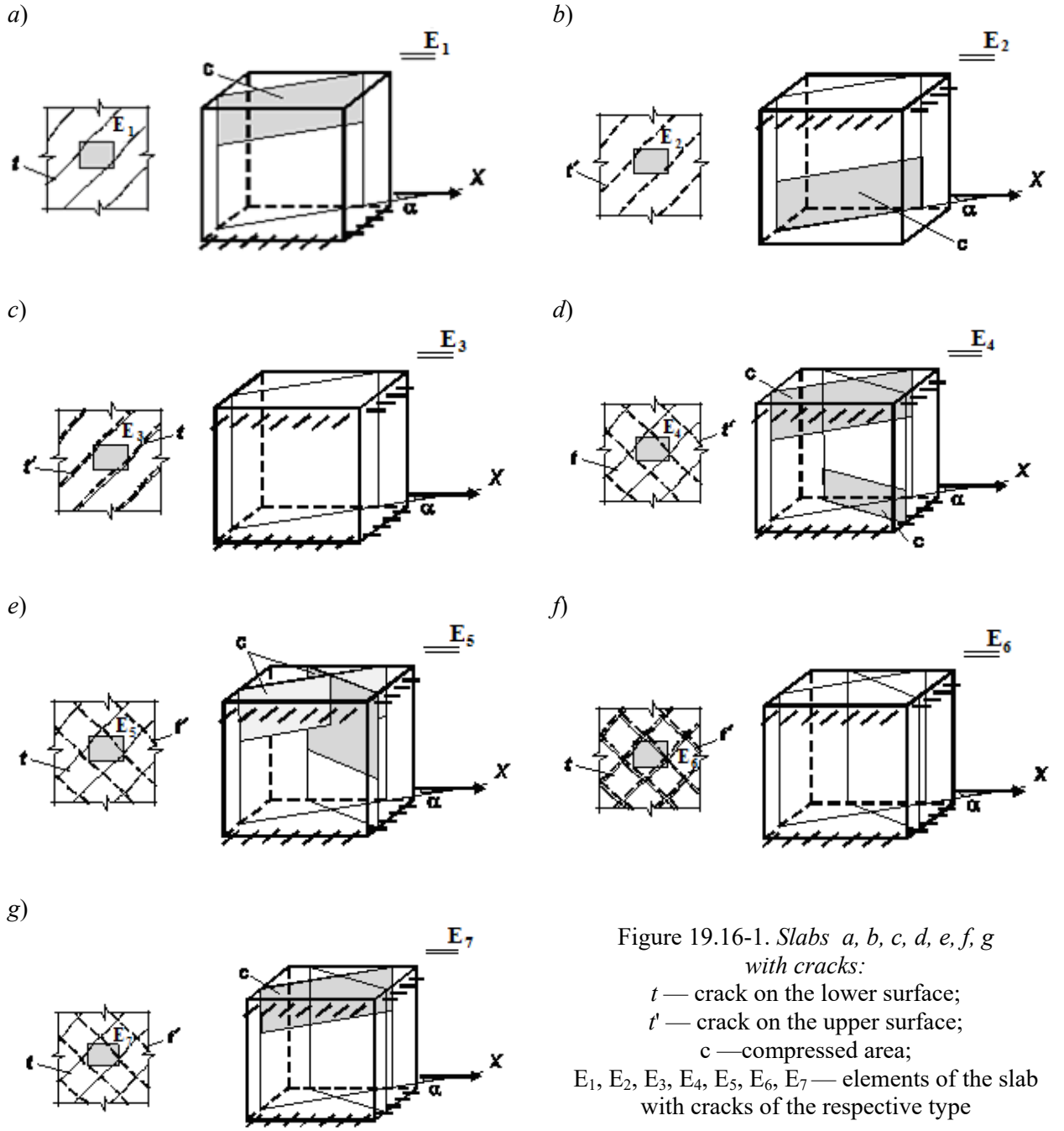


Figure 19.16-1. *Slabs a, b, c, d, e, f, g with cracks:*
t — crack on the lower surface;
t' — crack on the upper surface;
c —compressed area;
E_1, E_2, E_3, E_4, E_5, E_6, E_7 — elements of the slab with cracks of the respective type

The considered elements are assumed to be such that we can neglect the effect of shear forces on their deformability and strength both before and after the appearance of cracks. Due to this hypothesis the considered slabs belong to the category of thin slabs. If it has been somehow established that it is possible to neglect the action of shear forces, the obtained conclusions can be applied to plates of medium thickness as well.

The character of deformation of reinforced concrete spatial structures in areas with cracks depends on the arrangement of cracks, i.e. orientation of cracks with respect to the directions of reinforcement, mutual intersection of cracks, formation of cracks on one or both surfaces of the element, whether the cracks are through or non-through. Through cracks occur in a membrane stress state (in elements in the plane stress-strain state such as deep beams) or under the additional action of small moments. It is assumed that cracks are normal to the middle surface of elements.

The variety of cracks in elements can be reduced to seven arrangements (Fig. 19.16-1):

1. Nonintersecting cracks are located on one of the surfaces: lower (Fig. 19.16-1, *a*) or upper (Fig. 19.16-1, *b*); the elements have a tension area with cracks and a compressed area without cracks (elements E_1 and E_2).
2. The element is laced with nonintersecting through cracks (Fig. 19.16-1, *c*).
3. There are cracks in different directions both on the lower and on the upper surfaces of the element (Fig. 19.16-1, *d*). They often meet in the corner areas of slabs with large torques. The upper and lower areas have concrete strips between the cracks taking the compression forces (they are a kind of "compressed areas" laced with cracks).
4. One of the surfaces of the element — lower or upper (Fig. 19.16-1, *e*) has intersecting cracks in two directions.
5. The element is laced with intersecting through cracks (Fig. 19.16-1, *f*).
6. One of the surfaces has nonintersecting cracks, and the other one has intersecting cracks (Fig. 19.16-1, *g*).

All arrangements can be divided in two large groups: group *N* — nonintersecting cracks (see Fig. 19.16-1) (arrangements *a*, *b*, *c*), and group *I* — intersecting cracks (arrangements *d*, *e*, *f*, *g*).

Conditions of crack formation for elements subjected to the combined action of moments and normal forces can be formulated as a generalization of the theory of core moments of the crack formation of bar systems. The suggestion comes down to applying the theory of core moments of Gvozdev A.A. and Dmitriev S.A. [1] to more complex concrete structures of slabs and shells.

Two surfaces in the element at the distance of $\pm r_я$ from the middle surface are called the upper and lower surfaces of core points respectively. The value $r_я = h/6$ is determined as for a strip of slab by the formula of the strength of materials.

Let's introduce two new tensors of core moments: one with component M_{cx}, M_{cy} and M_{cxy}, where

$$M_{cx} = M_x + N_x r_c; \quad M_{cy} = M_y + N_y r_c; \quad M_{cxy} = M_{xy} + N_{xy} r_c, \tag{19.16.1}$$

and the other with components

$$M'_{cx} = - M_x + N_x r_c; \quad M'_{cy} = - M_y + N_y r_c; \quad M'_{cxy} = - M_{xy} + N_{xy} r_c. \tag{19.16.2}$$

From the very definition of core sizes which are calculated based on the condition of zero stresses on the edge of a cross-section [3], it follows that the appearance of the positive core moment means that there is tension on the respective edge. If we assume that tension has to be taken by reinforcement, the decision on whether the reinforcement is necessary is made based on the analysis of the following inequalities:

for the lower area of the element:

$$M_{c.max} \le 0; \qquad M_{c.min} \le 0, \tag{19.16.3}$$

for the upper area of the element:

$$M'_{c.max} \le 0; \qquad M'_{c.min} \le 0. \tag{19.16.4}$$

If the first condition is violated (19.16.3), a crack is formed according to the arrangement *a* (Fig. 19.16-1) on the area subjected to the main core moment $M_{c.max}$. The angle α between this area (crack) and the X axis at the moment of violation of the first inequality (19.16.3) can be determined using the following expression

$$\mathrm{tg}\alpha = (M_{c.max} - M_{cy})/ M_{cxy}. \tag{19.16.5}$$

When the first inequality is violated (19.16.4), the cracks are formed according to the arrangement *b* (Fig. 19.16-1). The angle between the upper crack and the X axis can be determined using the following expression

$$\mathrm{tg}\alpha = (M'_{c.max} - M'_{cy})/ M'_{cxy}. \tag{19.16.6}$$

Simultaneous violation of the first condition (19.16.3) or the first condition (19.16.4) and of the additional inequality $N_{cl} \le 0{,}75R_p h$ indicates that cracks are formed according to the arrangement *c* (Fig. 19.16-1). Additional inequality is established on the basis of the assumption that the through cracks are

formed at the moment when the cross section has an unambiguous trapezoidal diagram of tensile normal stresses equal on one of the extreme and middle surface to R_p, and on the other extreme surface — to zero.

Simultaneous violation of the first inequality (19.16.3) and of the first inequality (19.16.4) indicates that cracks are formed according to the arrangement *d* (Fig. 19.16-1).

Violation of two inequalities (19.16.3) indicates that cracks are formed in the lower area according to the arrangement *e* (Fig. 19.16-1), and violation of two inequalities (19.16.4) indicates that cracks are formed in the upper area according to the arrangement *e*.

It is assumed that simultaneous violation of conditions (19.16.3) and of inequalities $N_{c1} \le 0{,}75R_p h$ and $N_{c2} \le 0{,}75R_p h$ or (19.16.4) and inequalities $N'_{c1} \le 0{,}75R_p h$ and $N'_{c2} \le 0{,}75R_p h$ leads to the formation of cracks according to the arrangement *f* (Fig. 19.16-1).

Here N_{c1}, N_{c2}, N'_{c1}, N'_{c2} are the values of normal stresses on the respective areas subjected to the core moments.

Simultaneous violation of the condition (19.16.3) and of only one of the inequalities $N_{c1} \le 0{,}75R_p h$ and $N_{c2} \le 0{,}75R_p h$ or of the conditions (19.16.4) and of only one of the inequalities $N'_{c1} \le 0{,}75R_p h$ and $N'_{c2} \le 0{,}75R_p h$ indicates that cracks are formed according to the arrangement *g* (Fig. 19.16-1).

Strength Check

It is known that the angle of principal planes may not coincide with the direction of crack propagation [3]. Therefore, when checking the reinforcement **SCAD** also performs the search of areas where the conditions of strength may be violated the most. Suppose, for example, this area is located at an angle α to the X axis of reinforcement and it is assumed that the reinforcement is parallel to the X and Y axes of the Cartesian coordinate system. We will consider only the case when there is a non-through crack in the concrete and only the lower reinforcement is required (Fig. 19.16-2).

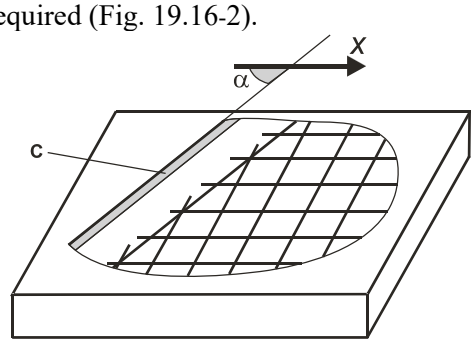


Figure 19.16-2. *To the strength analysis*

If we extract an elementary triangular prism (Fig. 19.16-3) and assume that the reinforcement transferring the forces in a crack takes only the longitudinal forces (neglect the dowel action), then when writing the equilibrium conditions with respect to axes 1 and 2 passing through the center of the compressed area of concrete parallel to axes X and Y, we will obtain:

$$R_a F_{a,x} Z \sin\alpha = (M_x + N_x Z_b)\sin\alpha + (M_{yx} + N_{yx} Z_b)\cos\alpha;$$
$$R_a F_{a,y} Z \cos\alpha = (M_y + N_y Z_b)\cos\alpha + (M_{xy} + N_{xy} Z_b)\sin\alpha, \qquad (19.16.7)$$

where Z is an arm of the tensile reinforcement with respect to the center of gravity of the compressed area of concrete, $F_{a,x}$ and $F_{a,y}$ are reinforcement areas per running meter in the X and Y direction respectively. Since the prism has a unit length of the oblique plane and all internal forces are given by their values per unit length, the equations of equilibrium will include the lengths of planes 1×sinα and 1×cosα the total force is taken from.

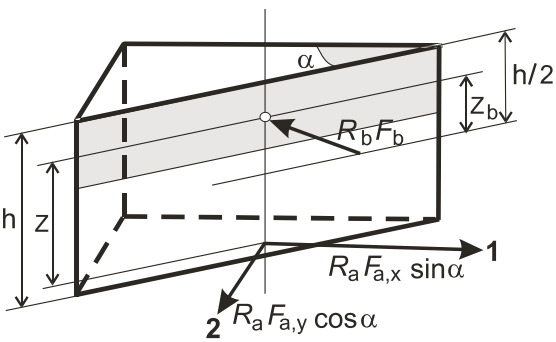


Figure 19.16-3. *To the determination of stresses in reinforcement and concrete in the area of an oblique crack:*

arrangement of forces in the area of the crack

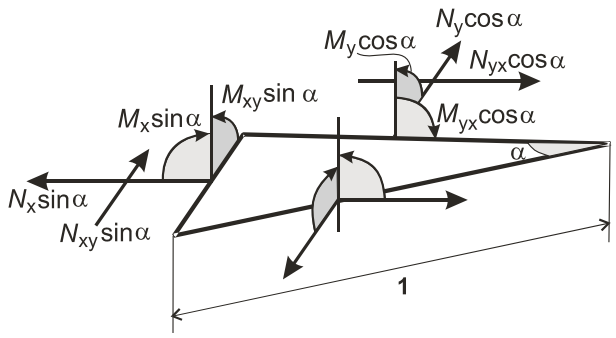


Figure 19.16-4. *To the determination of stresses in reinforcement and concrete in the area of an oblique crack:*

arrangement of moments and longitudinal forces applied to the sides of the elementary triangular prism

It is not difficult to adjust these equations for the case of compressed reinforcement. Then for example for the X axis we will obtain the following formula

$$M_{x1} + \sigma' A'_{sx1} Z' - M_{bx1} \le 0, \qquad (19.16.8)$$

where M_{x1} is the projection of moments of all forces on the plane perpendicular to the X axis;

$$M_{x1} = (M_x + N_x Z_b) + (M_{yx} + N_{yx} Z_b)\mathrm{ctg}\alpha\ , \qquad (19.16.9)$$

$\sigma' A'_{sx1} Z'$ is the moment of forces in reinforcement located in the compressed area of concrete and working in compression together with concrete with respect to the axis of the tensile reinforcement; M_{bx1} is the moment of forces in the compressed area of concrete with respect to the axis of the tensile reinforcement. Naturally, if there is compressed reinforcement, the value of the compressed area of concrete is clarified (this is done in an iterative cycle) and the value of Z is adjusted accordingly.

The condition of strength for the Y axis is determined in a similar way (by a simple replacement of values with the subscript *x* with the corresponding values with the subscript *y*).

For other cases of crack formation and reinforcement arrangements all considerations are performed similarly to those given above. They are described in detail in [3] and are used in **SCAD** when checking the elements of reinforced concrete structures and selecting reinforcement.

The aforementioned selection of the location of areas where the conditions of strength can be violated the most is implemented by the direct enumeration of angles α with a step of 7.5°. The performed comparative calculations have shown that the use of a smaller step of scanning does not lead to noticeable changes in the results, but significantly slows down the computation.

Analysis for Lateral Forces

When plates are calculated for the action of lateral forces, **SCAD** performs two separate calculations – for the action of Qx and for the action of Qy. In each case a strip of unit width is "cut out" from the plate in the respective direction. Then this strip is calculated for the action of the lateral force as a flexural element according to the recommendations of the codes for bar elements.

If we analyze the method of calculating plates for the action of shear forces, it can be seen that the strength is determined by a single value of the intensity $W = \dfrac{a_w}{s_x s_y}$ (a_w is the cross-sectional area of one transverse rebar), i.e. the amount of transverse reinforcement per unit area.

The values of the diameter of reinforcement and the spacing do not matter. It is only important that the value of W is greater than a certain value that ensures that the inequality of the following type is satisfied $\dfrac{Q}{0.5 \cdot R_{bt} \cdot h_0 + W \cdot R_{sw} \cdot h_0} \le 1$.

When determining the required transverse *assortment* reinforcement, the software selects the diameter (based on the set of diameters of the given class of reinforcement and the limitations on the maximum diameter) and two spacings (from the lists of allowable spacing), so that the value of W is greater than the *design* value and gives the minimum intensity (minimizes the steel consumption). The intensities per running meter in the directions W_x,W_y can be either less than the design values, or considerably greater.

Determination of the Crack Opening Width

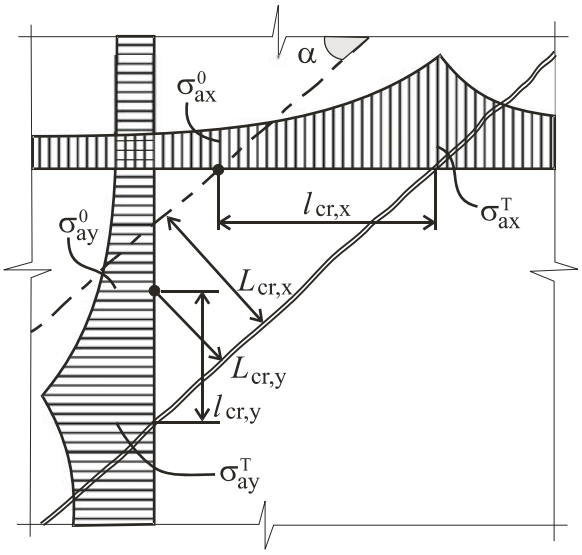


Figure 19.16-5. *Fragment of the surface of a plate with a crack.*
Diagrams of the variation of tensile stresses in reinforcement from the values σ^T_{ax} and σ^T_{ay} near the crack to the values σ^0_{ax} and σ^0_{ay} at the place of formation of a new crack

Crack opening calculation is performed in the same way as the strength analysis for one of the variants of their formation.

Two assumptions are used:

- the change in the stress state in the area between adjacent cracks can be neglected;
- adjacent cracks can be considered as parallel.

In accordance with the guidelines of V.I.Murashev [4], which are used as the basis for the current SNiP, it is assumed that the adjacent crack is formed at such a distance l_{cr}, that the stress in reinforcement due to the adhesion forces of concrete decrease from the value σ^T_a to σ^0_a, and the forces in concrete increase from zero near the crack to the values causing the formation of a new crack. The following values are predetermined:

$$L_{crc,x} = (bh^2/3{,}5)(d_x/4)\eta\ \varphi_l / (A_{sx}Z)\cos\alpha;$$
$$L_{crc,y} = (bh^2/3{,}5)(d_y/4)\eta\ \varphi_l / (A_{sy}Z)\sin\alpha\ , \qquad (19.16.10)$$

where d_x and d_y are rebar diameters;

η is the coefficient accounting for the profile of reinforcement;

φ_l is the coefficient accounting for the duration of the load action for different types of concrete.

The average distance between cracks L_{crc} is determined as the maximal of the two values (19.16.10), since this value defines the area of the plate where the adhesion between concrete and reinforcement is fully restored in two directions.

A value for the lower or upper reinforcement (depending on the situation) in the respective direction is used as the area of reinforcement per unit length A_{sx} and A_{sy}.

Stresses in reinforcement are determined by the formula:

$$\sigma_{sx} = M_x / (A_{sx}Z);\ \ \sigma_{sy} = M_y / (A_{sy}Z), \qquad (19.16.11)$$

where M_i is the projection of moments of all forces acting on the section with respect to the axis of the respective tensile reinforcement (i = x, y);

Z is an arm of the tensile reinforcement with respect to the center of gravity of the compressed area of concrete;

A_{sx} and A_{sy} is the area of tensile reinforcement per unit length.

The crack opening width is determined by the formula:

$$a_{crc} = (\varepsilon_{sx} + \varepsilon_{sy}) L_{crc}, \qquad (19.16.12)$$

where $\varepsilon_{sx} = (\sigma_{sx} * \psi_s) / E_s$ and $\varepsilon_{sy} = (\sigma_{sy} * \psi_s) / E_s$. Here ψ_s is the coefficient accounting for the behavior of concrete in tension within the area with cracks and determined according to the recommendations of Sec. 4.29 SNiP.

Thus the crack opening width is determined by the average distance between them, and the stresses in reinforcement are determined without using the semi-empirical relationship given in SNiP for bar elements.

General Algorithm

In the general case it is necessary to determine the values for six types of reinforcement (Fig. 19.16-6).

A set of unfavorable combinations of loadings is predetermined. This set includes internal forces creating extreme values of normal stresses on the inner and outer surfaces, and also extreme values of shear stresses in the direction of the plate thickness. The values relating to the midpoints of finite elements are used.

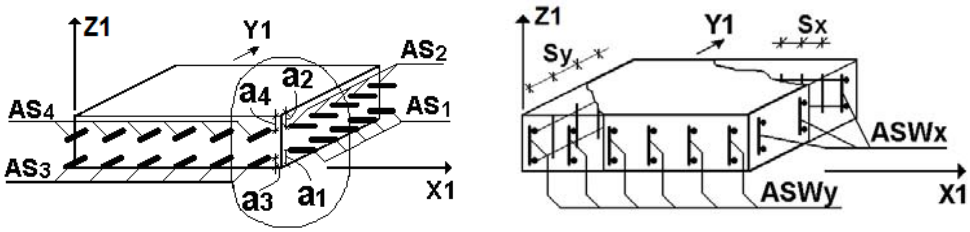


Figure 19.16-6. *Reinforcement layout in sections of a finite element of a slab and shell*

The strength of the plate is checked by the enumeration of variants of combinations for each finite element (core moments, angles α, the possible arrangement of cracks etc. are determined) and if the conditions of strength are not satisfied, the value of the intensity of reinforcement is increased. This is done until the conditions of strength are satisfied or it turns out that it is impossible to provide the conditions of strength without exceeding the specified maximum percentage of reinforcement, and the respective message is generated in this case.

Then the crack opening width is checked for the selected reinforcement (if necessary). If the crack opening width exceeds the allowable value, the area of reinforcement is increased until the crack opening width becomes satisfactory. When selecting the longitudinal and transverse reinforcement from the action of each subsequent combination the reinforcement selected from the action of previous combinations is taken into account.

19.17. Implementation of SNiP 52-101-2003 and SP 63.13330

The selection and check of reinforcement in the elements of reinforced concrete structures in accordance with the recommendations of SNiP 52-01-2003 (SP 52-101-2003) and SP 63.13330 are implemented in **SCAD**. It should be noted that SP 52-101-2003 does not provide any recommendations on the analysis of behavior of reinforced concrete structures under dynamic actions. **SCAD** does not prohibit the calculation of this type, but the responsibility for the use of the results of the calculation in this case should be taken by the user.

The recommendations of SNiP and SP relating to the nonlinear deformation model of the behavior of reinforced concrete are used in the process of analysis of the normal cross-sections. A bilinear diagram of the state of concrete recommended by SNiP is used in the strength analysis for a nonlinear deformation

model, and a trilinear diagram of the state of concrete recommended by SNiP is used in the analysis on crack formation.

Besides the recommendations of SNiP and SP materials given in Guide to SP 52-101-2003 (Sec. 3.29-3.35, 3.50, 3.52, 3.71, 4.28) are used in the analysis of transverse reinforcement.

The following main checks of sections are performed in the process of the reinforcement selection:

	SP 52-101-2003	SP 63.13330
Deformation of concrete in the normal section	Sec. 6.2.25	Sec. 8.1.24
Deformation of reinforcement in the normal section	Sec. 6.2.25	Sec. 8.1.24
Deformation of concrete in tension in the normal section	Sec. 6.2.30	Sec. 8.1.29
Strength in an oblique concrete strip under Q_z	Sec. 6.2.33	Sec. 8.1.32, 8.1.34
Strength in an oblique concrete strip under Q_y	Sec. 6.2.33	Sec. 8.1.32, 8.1.34
Strength of the oblique section under Q_z	Sec. 6.2.34	Sec. 8.1.33, 8.1.34
Strength of the oblique section under Q_y	Sec. 6.2.34	Sec. 8.1.33, 8.1.34
Strength of the oblique section under the moment M_z	Sec. 6.2.35	Sec. 8.1.35
Strength of the oblique section under the moment M_y	Sec. 6.2.35	Sec. 8.1.35
Torsion strength of concrete	Sec. 6.2.37	Sec. 8.1.37
Torsion strength of the reinforcement	Sec. 6.2.38	Sec. 8.1.38
Element strength between spatial sections	Sec. 6.2.41	Sec. 8.1.41
Short-term crack opening width	Sec. 7.2.3	Sec. 8.2.6
Long-term crack opening width	Sec. 7.2.3	Sec. 8.2.6

It should be noted that if SP 63.13330 is selected, the torsional analysis performed by the program is based on the classical formulas by Mersch and Rausch [10]. This is due to the fact that when the deformation model recommended by SP is used in the analysis of compressed elements subjected to torsion, Sec. 8.1.40 has a reference to Sec. 8.1.9, which, firstly, deals with a force model (although the analysis for a deformation model enables to define such a concept as the ultimate bending moment) and, secondly, deals with the analysis of flexural elements with only a rectangular cross-section. There are similar inconsistencies in SP 52-101-2003 as well.

When the calculations are performed according to SP 52-101-2003 or SP 63.13330 the software calculates the following factors:

- Strength for the ultimate longitudinal force;
- Ultimate moment strength.

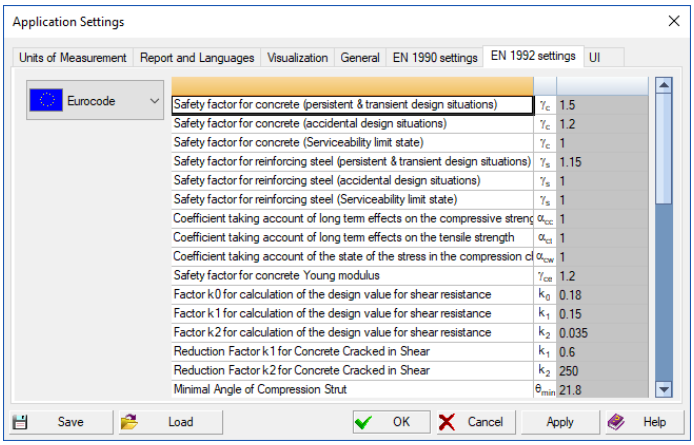
The first one is the ratio $\frac{|N|}{R_s A_{s,tot}}$ for the elements in tension and $\frac{|N|}{R_s A_{s,tot} + R_b A}$ for the elements in compression. Although there is no requirement for checking this factor in the codes (except for Sec. 6.2.19 of SP 52-101-2003 and Sec. 8.1.18 of SP 63.13330), we can not perform other calculations without first checking that this value in less than one, for example, calculate the coefficient φ_n in Sec. 8.1.34 of SP 63.13330. The *Ultimate moment strength* factor was introduced for similar reasons. N, M_y, M_z have to be divided by this number to obtain the maximum allowable values of the deformations of concrete or reinforcement.

19.18. Peculiarities of the Analysis according to EN 1992-1-1

If the analysis is performed according to EN 1992-1-1, you have to specify the following data on concrete:

- Concrete class;
- Concrete aggregate;
- Age of concrete;
- Strength class of cement;
- Time of creep development;
- Temperature T during the creep development;
- Number of days where a temperature T prevails;
- Relative humidity;
- Additional service factor.

Various factors and partial safety factors are taken from the respective National Annexes to Eurocode, and can be selected in the **EN 1992 Settings** tab of the multi-tab **Settings** dialog box, where you can also modify some of these parameters.



Additional service factors for concrete and reinforcement enable to modify the design strength of materials and are used together with the factors provided in EN 1992-1-1, such as γ_s, α_{CC}, … . Design resistances are *divided* by these additional factors.

When the analysis is performed according to EN 1992-1-1, you do not have to specify the data on the random eccentricities and static definability. Geometric imperfections are taken into account according to Sec. 5.2(9). Second order effects are taken into account by the method based on nominal stiffness (Sec. 5.8.7). Creep is taken into account based on the recommendations of Annex B. Formula (5.29) conservatively uses the value c_0=8, which is applicable both in the case of a uniaxial stress state and biaxial bending (see Note to Sec. 5.8.7.3(3)).

Strength analysis of normal sections is based on the concrete and reinforcement strain diagrams (Sec. 6.1). Bi-linear Prandtl diagrams are used (see Fig. 3.4, 3.8 of EN 1992-1-1).

Unlike SNiP, the user has to specify only one value in order to perform the crack resistance analysis — maximum crack width (Sec. 7.3.1(5)), which can be specified using, for example, Table 7.1N of EN 1992-1-1. Calculation of crack widths is provided in the case of single bending only, because EN 1992 considers only the case of a rectangular tension zones. For the same reason the calculation of the crack widths for circular sections is not implemented.

The analysis of plate members is based on the Karpenko theory using the properties of materials (concrete and reinforcement) provided in EN 1992-1-1. Shear analysis of slabs is performed according to the recommendations of EN 1992-1-1.

The cross-sections of bar structural members are checked for compliance with requirements of EN 1992-1-1 for the following factors:

Check	Reference to EN 1992-1-1
Ultimate moment strength	
Compressive strain in concrete	Sec. 6.1(3)P
Tensile strains in the reinforcing steel	Sec. 6.1(3)P
Axial force including second order effects	Sec. 5.8.7.3
Cracking stress	Sec. 7.1(2)
Crack width	Sec. 7.3.4
Shear resistance to Vz without shear reinforcement	Sec. 6.2.2
Shear resistance to Vy without shear reinforcement	Sec. 6.2.2
Shear resistance to Vz with shear reinforcement	Sec. 6.2.3
Shear resistance to Vy with shear reinforcement	Sec. 6.2.3
Design torsional resistance of a cross-section	Sec. 6.3.2 (4)
Torsional resistance of longitudinal reinforcement	Sec. 6.3.2 (4)
Torsional resistance of shear reinforcement	Sec. 6.3.2 (4)
Resistance of a member subjected to torsion and shear	Sec. 6.3.2(4)

The plate structural members are checked for the following factors:

Check	Reference to EN 1992-1-1
Strength of the slab section	
Shear resistance to Vx without shear reinforcement	Sec. 6.2.2
Shear resistance to Vy without shear reinforcement	Sec. 6.2.2
Shear resistance to Vx with shear reinforcement	Sec. 6.2.3
Shear resistance to Vy with shear reinforcement	Sec. 6.2.3
Possibility of cracking	Sec. 7.1(2)
Crack width	

19.19. Peculiarities of the Analysis according to DBN V.2.6-98:2009

If the analysis is performed according to DBN V.2.6-98:2009 (DSTU B V.2.6-156:2010), it should be noted that these design codes contain a huge number of errors and they are not consistent with other design codes of Ukraine, in particular, with DBN V.1.2-2:2006.

Since DBN V.2.6-98:2009 and DSTU B V.2.6-156:2010 are based on EN 1992-1-1, in case of conflict SCAD uses the corresponding provisions of EN 1992-1-1.

19.20. Factors in Stress-Strain Models

When comparing the results of the calculation for the limit forces and the nonlinear deformation model, the factors (utilization factors of restriction) can differ by many times. In order to explain the reason for this discrepancy, let's consider the strength analysis of the normal cross-section of a bar under central tension.

When performing the calculation for limit forces, the following restriction must be satisfied (see Sec. 8.1.18 of SP 63.13330)

$$N \le R_s A_{s,tot},$$

where N is a longitudinal tensile force from external loads;
R_s is the design tensile strength of the reinforcement;
$A_{s,tot}$ is a cross-sectional area of all longitudinal reinforcement.

In this case, the respective factor is equal to

$$k_{force} = N/R_s A_{s,tot},$$

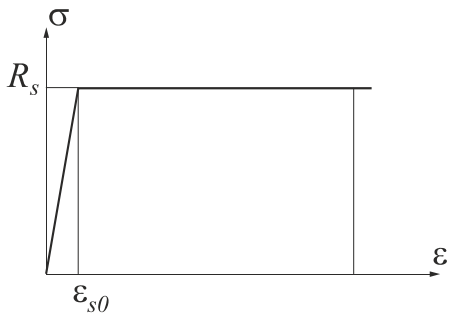
i.e. k_{force} depends linearly on N.

When using a nonlinear deformation model, the following restriction must be satisfied (see Sec. 8.1.24 of SP 63.13330)

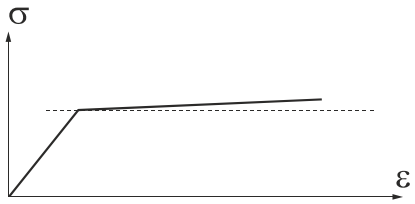
$$k_{def} = \varepsilon_{s,max}/\varepsilon_{s,ult} \le 1,$$

where $\varepsilon_{s,max}$ is a relative deformation of a rebar under the greatest tension in the normal section of the element from the action of the external load;
$\varepsilon_{s,ult}$ is the limit value of the relative reinforcement elongation.

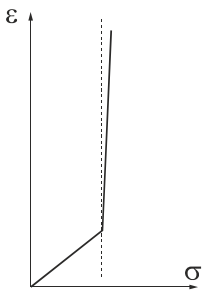
So, knowing the value of N (or the stress in the reinforcement $\sigma_s = N/A_{s,tot}$), it is necessary to determine the relative deformation of the reinforcement ε on the basis of the state diagram and compare it with the maximum allowable one ($\varepsilon_{s,ult}$). SP 63.13330 recommends to use the following type of diagram



If $N/A_{s,tot} < R_s$, then no problems arise, but even when $N/A_{s,tot} = R_s$, the diagram recommended by SP does not allow to determine ε unambiguously (ε can take any value greater than ε_{s0}) and obtain a factor allowing to make a conclusion about how much it is necessary to increase the load-bearing capacity of the section (or to reduce the value of the longitudinal force) in order to meet the requirements of the codes. Therefore, a modified diagram is used in the calculations in **SCAD** and **Arbat**, in which the upper section is not horizontal but slightly inclined:

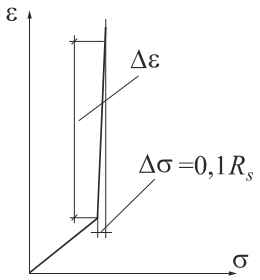


Let's redraw this diagram as follows



Suppose that the longitudinal force exceeds the maximum allowable value given in Sec. 8.1.18 by 10%, i.e. $k_{force} = 1{,}1$.

If we use a nonlinear deformation model, we will get the following picture,



i.e., the relative deformation has increased significantly (much more than 10%) and we get the factor $k_{def} >> 1{,}1$.

Let us consider another example that shows significant difference between k_{def} and k_{force} when the normal section strength requirements of SP are satisfied. Considering an element under axial tension with A500 reinforcement again, we assume that the longitudinal force is such that the stress of $0.99R_s$ occurs in the reinforcement. It is obvious that in this case k_{force}=0.99. And the reinforcement strain is equal to $0.99e_{s0}$, where (according to Sec. 6.2.11 of SP 63.13330) $e_{s0}=R_s/E_s$=0.002175. That is $k_{def}=0.99e_{s0}/e_{s,ult}$=0.8613.

A similar situation can arise when comparing other factors.

19.21. Analysis of Round Sections

Technically, the rules for the analysis of bar elements in shear provided in the design codes do not contain any restrictions on the shape of the cross section. However, using parameters b and h_0 for the analysis of round sections makes no sense.

The following technique is used to get around this problem. A round section with radius R is replaced by a square with an area equal to that of a circle with radius R-a, where a is the distance to the center of gravity of the reinforcement. The analysis of ring sections is performed in a similar way.

It should be noted that similar recommendations can be found in Sec. 3.3 of *Guidelines for the Design and Analysis of Bored Pile Retaining Walls*.

19.22. Examples of Using the Results of the Reinforcement Selection in Designing

Let's consider how the obtained results can be used when making design decisions using the examples of the transverse reinforcement selected by the program.

Example №1:

The calculated intensity of transverse reinforcement per meter in a shell finite element:

$$W_x = 4{,}5 \text{ cm}^2/\text{m};$$
$$W_y = 6{,}5 \text{ cm}^2/\text{m}.$$

Problem:

Determine the spacing of transverse reinforcement s_x and s_y to provide this intensity.

Solution:

Let's take rebars with the diameter of 10 mm for transverse reinforcement (bar cross-sectional area a_w = 78,54 mm^2 = 0,7854 cm^2). The spacing of reinforcement along the Y_1 axis will be:

$$s_y = \frac{a_w}{W_y} = \frac{0{,}7854}{6{,}5} = 0{,}121\text{m} = 120\text{mm};$$

The spacing of reinforcement along the X_1 axis will be:

$$s_x = \frac{a_w}{W_x} = \frac{0{,}7854}{4{,}5} = 0{,}175\text{m} = 175\text{mm}.$$

Let's reduce the spacing of transverse reinforcement to the "standard" one, — multiple of 50 mm (for example, 50, 100, 150, …). In order to provide the necessary intensity, let's take the spacing of transverse reinforcement as (rounded down) s_y = 100 mm, s_x = 150 mm.

Example №2:

The calculated intensity of transverse reinforcement per meter in a shell finite element:

$$W_x = 4{,}5 \text{ cm}^2/\text{m};$$
$$W_y = 6{,}5 \text{ cm}^2/\text{m}.$$

Problem:

Determine the diameter of transverse reinforcement to provide this intensity when:

$$s_x = s_y = 150 \text{ mm} = 0{,}15 \text{ m}.$$

Solution:

Cross-sectional area of transverse rebars (located along the Y_1 axis with the spacing s_y) providing the intensity IW_y:

$$a_w = W_y \cdot s_y = 6{,}5 \cdot 0{,}15 = 0{,}975 \text{ cm}^2.$$

The closest assortment reinforcement — ∅ 12 mm, (a_w = 113,1 mm^2 = 1,131 cm^2).

Cross-sectional area of transverse rebars (located along the X_1 axis with the spacing s_x) providing the intensity IW_x:

$$a_w = W_x \cdot s_x = 4{,}5 \cdot 0{,}15 = 0{,}675 \text{ cm}^2.$$

The closest assortment reinforcement — ∅ 10 mm (a_w = 78,54 mm^2 = 0,7854 cm^2).

Since a bar of transverse reinforcement works simultaneously in both directions, in order to provide the necessary intensity of reinforcement at the given spacing s_x = s_y = 0,15 m, let's take the diameter of ∅12 mm.

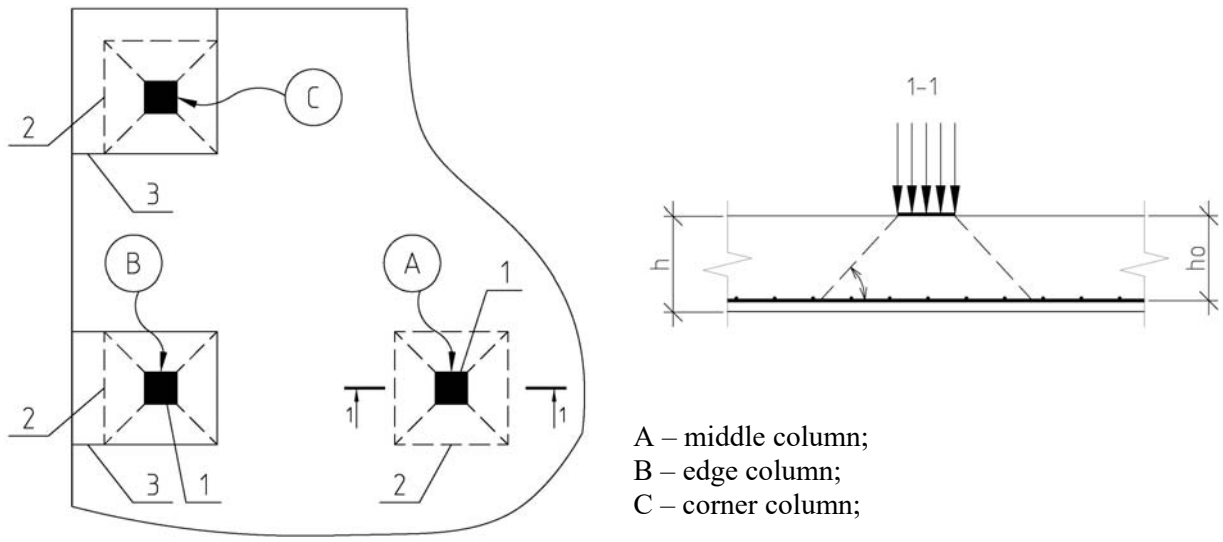
19.23. Punching Analysis

The punching analysis of plate elements is performed for the local concentrated forces (normal to the plane of the element) and bending moments in two planes according to the method given in SNiP 2.03.01-84*, SP 52-101-2003 and SP 63.13330.

It is assumed that punching occurs on the lateral surface of the pyramid.

The software uses the same general approaches for the classification of the design case of punching, classification of the column type (middle, edge and corner – see the figure) for SNiP, and for SP (although, for example, SNiP considers only the case of a middle column with a rectangular cross-section).

The punching analysis takes into account the position of the column with respect to the edge of the slab. There are the following types of columns: middle, edge and corner (see the figure). When performing the analysis the length of the design punching contour is determined in different ways depending on the position of the column.



A – middle column;
B – edge column;
C – corner column;

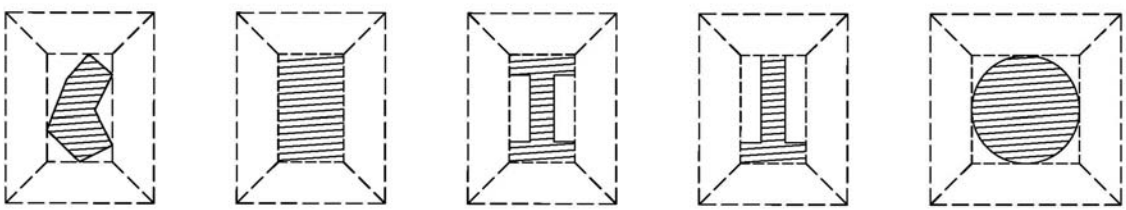
1 – load application area (load area); 2 – closed design contour;
3 – open design contour;

The analysis is performed both for the concrete slab cross-section, and for the cross-section of a slab with transverse reinforcement. The software considers only the uniform arrangement of the transverse reinforcement around the load application area.

The strength check of the concrete section is always performed, and when necessary, the minimum required intensity of transverse reinforcement is determined (diameter and spacing in two directions).

In addition to the actual check of the strength of elements for the action of the local load at punching the check of the number of structural requirements of the codes is also performed during the calculation (in some cases they are obligatory).

The load application area at punching is considered only as a rectangle in the codes. In order to calculate other shapes of the column cross-section a minimum circumscribed rectangle with the sides parallel to the principal axes of inertia of the column section is constructed around them.



Reduction of different column sections to the design rectangle

A contour of the design cross-section is constructed around each column. In the case of the **middle** column this contour is always closed and it is located around the load transfer area (contour of the load area) at the distance specified in the codes. In the case of the **corner** and **edge** columns two punching contours are considered – closed and open ones (L-shaped for a corner column and U-shaped for an edge column). Moreover, according to the recommendations of SP 52-101-2003 and SP 63.13330 the strength check is performed for the contour beyond which the transverse reinforcement is not considered.

In order to perform the punching analysis it is necessary to first calculate the design combinations of punching forces. It should be noted that the calculation of the given design combinations in the process of the general calculation is not performed automatically, and the user has to invoke this calculation in the project tree.

Checks at the punching analysis

Check	SNiP 2.03.01-84*	SP 52-101-03	SP 63.13330
Strength without taking the reinforcement into account	Sec.3.42	Sec. 6.2.49	Sec.8.1.49
Strength taking the reinforcement into account	Sec.3.42	Sec. 6.2.50	Sec. 8.1.50

Peculiarities of the implementation

SNiP 2.03.01-84* considers only the case of the middle column. In the case of a corner or edge column the method given in SP 63.13330 is used to determine the length of the lower base of the pyramid.

If the plate elements located near the punching node have different thicknesses, the minimum thickness is assumed.

The codes do not provide any clear rules for the classification of the column type (middle, edge and corner). When the calculation is performed in the program, it is assumed that the distances to the unsupported edges of the slab (for the corner and edge columns) are taken into account in the case when the distance from the boundary of the load application area to the unsupported edge does not exceed 3 effective heights (h_0) of the longitudinal reinforcement of the slab along one of the axes X, Y (the unsupported edge of the slab), or simultaneously along both axes X, Y (unsupported angle of the slab).

Since a particular arrangement of the transverse reinforcement around a column is unknown, the following restriction is introduced: the boundary of the contour for the check calculation is assumed without the consideration of the reinforcement at a distance of $0{,}5h_0$ from the boundary of the area within which the transverse reinforcement is taken into account in the calculation.

Data Preparation, Calculation and Analysis of Results

The operations of the preparation of the initial data, calculation and the analysis of the results at the punching analysis can be performed with the help of the toolbar which is invoked from the **Reinforced Concrete** toolbar (Fig. 19.23-1).

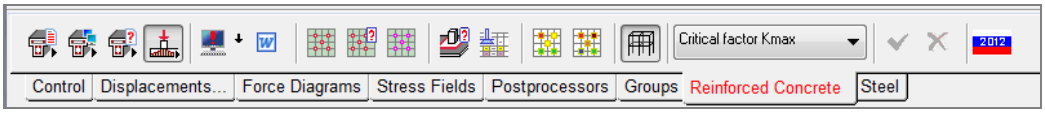


Figure 19.23-1. *Toolbar of the punching analysis*

This toolbar includes the buttons for invoking the following operations:

— punching analysis;

— report on punching;

— specify punching nodes;

— select punching nodes;

— show punching nodes;

— show punching contours;

— display the results on the model (3-color scale);

— display the results on the model (full color scale);

— show results of the punching analysis;

— display design model;

— select the design codes.

Specify Punching Nodes

This operation is used to create a list of punching nodes and to edit it by replacing the list, adding and removing the nodes from the list. After invoking this operation the **Create Punching Nodes** dialog box appears (Fig. 19.23-2), where you can select a method for specifying nodes, and the operations with the list (in the case of an existing list).

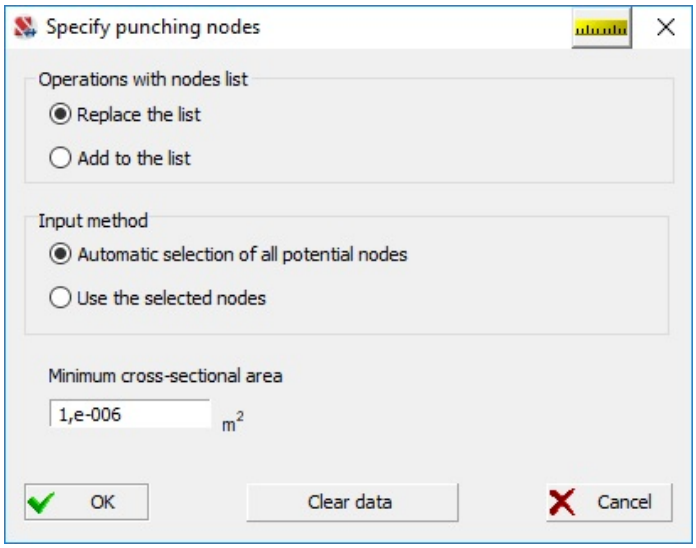


Figure 19.23-2. *The* **Create Punching Nodes** *dialog box*

There are two modes for specifying the punching nodes — automatic and “manual”. The automatic generation of the list implies that any node which belongs simultaneously to the horizontal plate elements and to the vertical bar located below or above the plate is specified as a punching node. Moreover, this node must not belong to other horizontal or inclined bars.

The following procedure is used for the “manual” specification of the list of nodes or its adjustment:

- invoke the **Specify punching nodes** operation, the **Create Punching Nodes** dialog box will appear where you have to select the **Use the selected nodes** radio button in the **Input method** group;
- select the **Replace the list** radio button in the **Operations with nodes list** group. In this case if the list is not yet generated, a new list will be created from the selected nodes; otherwise the selected

nodes will replace those specified in the list. If the **Add to the list** radio button is selected, the selected nodes will be added to the existing list;

- close the dialog box by clicking the **OK** button and select the nodes in the design model for which you want to perform the punching analysis;
- click the **Confirmation** button in the toolbar and the list will be generated.

The **Clear data** button enables to delete the generated list.

Moreover, the minimum cross-sectional area of the columns which should be taken into account when performing the punching analysis can be specified in the considered dialog box. The nodes of the columns with a smaller cross-sectional area will not be included in the list of punching nodes.

Select Punching Nodes

This operation is used to select the nodes included in the list of punching nodes (if you want to perform certain actions with these nodes).

Show Punching Nodes

This operation enables to highlight the punching nodes in the design model. This method of selection does not imply that operations will be performed with thus selected objects.

Calculation

The punching analysis can be performed only if all the horizontal plate elements adjacent to a certain node or lying in the vicinity of this node inside or on the boundary of the punching contour belong to one main group of the reinforcement of plates.

If there is no information on the specified transverse reinforcement for the plate finite elements included in the area of the analysis of the punching node, the calculation is performed without taking the reinforcement into account.

Show Punching Contours

This operation enables to display the punching contours on the design model obtained in the result of the calculation, and the load areas. The contours are displayed in magenta, and the areas — in blue (Fig. 19.23-3). When the calculations are performed according to SP and a corner or an edge column is considered, a contour (closed or open) corresponding to a greater utilization factor of restrictions will be displayed.

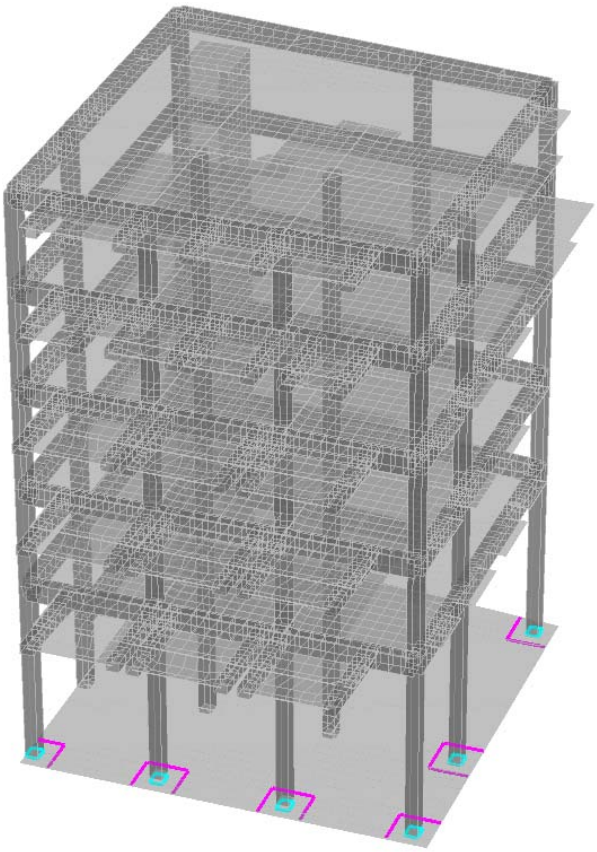
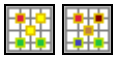


Figure 19.23-3. *Design model and punching contours*

Display Results



The results of the check for punching are given as the utilization factors of restrictions.

There are two types of the display of the results of the check — as a 3-color and a full color scale. In the first case the **Display the results on the model (3-color scale)** — is used for the display. When it is performed, the punching nodes are displayed in three colors:

- green — the load-bearing capacity is provided;
- red — the load-bearing capacity is insufficient;
- yellow — the value of the factor of restrictions is close to the critical value (the factor is close to 1,0).

The critical range is specified by setting the color scale in the **Settings** dialog box which is invoked by clicking the respective button — .

For example, if you specify the value 0,03 in the dialog box (Fig. 19.23-4), all elements with the value of the factor less than 0.97 will be displayed in green, if the factor is greater than 1,03 — in red, and if the factor is in the range [0,97;1,03] — yellow will be used.

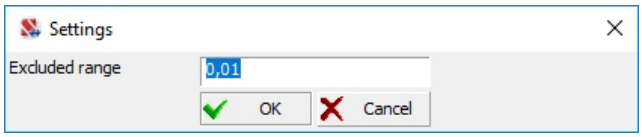


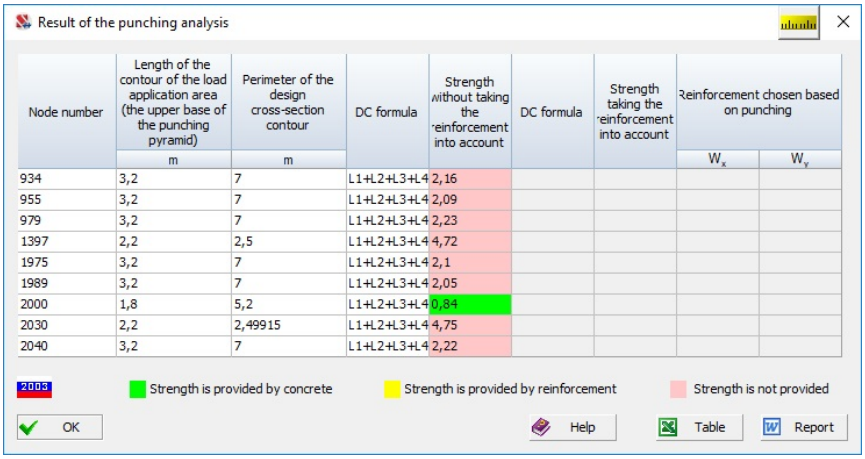
Figure 19.23-4. *The* **Settings** *dialog box*

If a display mode in the form of a full color scale is used (the **Display the results on the model (full color scale)** button —), an ordinary color scale with a custom number of colors will appear.

The list of factors contains the list of performed checks, and you can display on the model the results of the check for each factor defining the load-bearing capacity of the considered structures (for example, strength without taking the reinforcement into account). The values of the selected factor can be output on the model using the filters button **Digitization** — .

Table of Results

This operation enables to obtain the results of the punching analysis in the tabular form (Fig. 19.23-5). The lengths of contours of the lower and upper base of the punching pyramid, factor for strength without taking the reinforcement into account K_1 and/or factor for strength taking the reinforcement into account K_2 are output for each punching node. Factor K_2 is calculated only when all horizontal plates adjacent to the punching node have the same specified transverse reinforcement; moreover, the diameter and spacing (in the X, Y direction) of the optimal (in terms of the consumption of transverse reinforcement) transverse reinforcement of plates are output in the **Selected reinforcement** columns.



Node number	Length of the contour of the load application area (the upper base of the punching pyramid)	Perimeter of the design cross-section contour	DC formula	Strength without taking the reinforcement into account	DC formula	Strength taking the reinforcement into account	Reinforcement chosen based on punching	
	m	m					W_x	W_y
934	3,2	7	L1+L2+L3+L4	2,16				
955	3,2	7	L1+L2+L3+L4	2,09				
979	3,2	7	L1+L2+L3+L4	2,23				
1397	2,2	2,5	L1+L2+L3+L4	4,72				
1975	3,2	7	L1+L2+L3+L4	2,1				
1989	3,2	7	L1+L2+L3+L4	2,05				
2000	1,8	5,2	L1+L2+L3+L4	0,84				
2030	2,2	2,49915	L1+L2+L3+L4	4,75				
2040	3,2	7	L1+L2+L3+L4	2,22				

Figure 19.23-5. *The* **Result of the Punching Analysis** *dialog box*

It should be noted that it is impossible to select reinforcement in some cases. For example, if the calculations are performed according to SNiP 2.03.01-84* and the factor for strength without taking the reinforcement into account is greater than 2,0, the installation of transverse reinforcement can not provide the punching strength (see limitation Fb+0,8Fsw ≤ 2Fb in Sec. 3.42).

Generating a Report

A report can be generated on the basis of the results of the check. It is generated in the **RTF** format and is automatically loaded into an application associated with this format (for example, MS Word or WordPad). The dialog box (Fig. 19.23-6) appears before generating the report, where you have to select punching nodes, information on which has to be included in the report.

The dialog box provides the following options of the output control:

- **For all nodes** — the report will include all punching nodes;
- **For selected nodes** — in this case you have to specify the numbers of nodes for which the report will be generated in the text field.

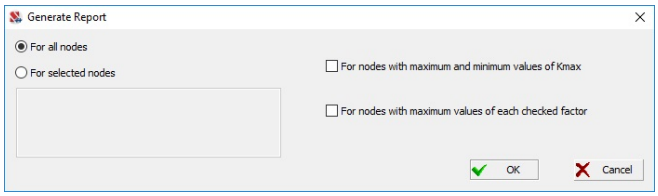


Figure 19.23-6. *The* **Generate Report** *dialog box*

Moreover, the **For nodes with maximum and minimum values of K_{max}** and **For nodes with maximum values of each checked factor** checkboxes enable to obtain a report only for the nodes with the maximum values of the factors.

References to Chapter 19

1. A.A. Gvozdev, S.A. Dmitriev, To the calculation of prestressed, ordinary reinforced concrete and concrete sections on the formation of cracks // Concrete and reinforced concrete, 1957, No. 5.
2. A.S. Zalesov, T.A. Mukhamediev, E.A. Chistyakov, Analysis on crack resistance of reinforced concrete structures according to the new standards and regulations // Concrete and reinforced concrete, 2002, No. 5.
3. N.I. Karpenko, The theory of deformation of reinforced concrete with cracks. — M.: Stroyizdat, 1976.
4. V.I. Murashev, Crack resistance, rigidity and strength of reinforced concrete. — M.: Mashstroyizdat, 1950.
5. Guide on designing of concrete and reinforced concrete structures made of heavy-weight and lightweight concrete (no prestressing) (to SNiP 2.03.01–84*) / Central Res. Inst. of Industrial Buildings and Res. Inst. for Reinforced Concrete of USSR State Comm. for Constr. and Archit.— M.: Stroyizdat, 1986.
6. SNiP 2.03.01-84*. Concrete and reinforced concrete structures. — M.: Stroyizdat, 1985.
7. SNiP II-7-81*. Construction in seismic regions. — M.: Stroyizdat, 1982.
8. A.P. Filin, Applied solid mechanics Volume II. — M.: Nauka, 1978.
9. V.S. Karpilovsky, E.Z. Kriksunov, A.A. Malyarenko, M.A. Mikitarenko, A.V. Perelmuter, M.A. Perelmuter, V.G. Fedorovsky, V.V. Yurchenko, “SCAD Office. Implementation of SNiP in Computer-Aided Design Applications” — M.: SCAD SOFT, 2014 — 480p.
10. V.I. Murashev, E.B. Sigalov, V.N. Baikov, Ferroconcrete constructions. Moscow, “Gosstroyizdat”, 1962. 660 p.

20. Check of the Load-bearing Capacity of Steel Sections

This mode enables to **check** the load-bearing capacity of bar elements of steel structures in accordance with the requirements of EN 1993, SNiP II-23-81*, SP 53-102-2004, SP 16.13330.2011, СП 16.13330.2017, SNiP RK 5.04-23-2002, DBN B.2.6-163:2010, DBN B.2.6-198:2014, STR 2.05.08:2005.

In the current version of the software the cross-sections of bars can be made from rolled profiles, from I-sections and hollow sections specified as parametric sections, and also from custom configuration sections created with the help of **Section Builder**, **Consul** or **TONUS**. This mode can be also used for the **selection** of sections of bars in those cases when their cross-sections are taken from rolled profiles or from welded sections given as an assortment (for example, according to the enterprise specifications).

The control of the operations of the input of initial data, analysis of the load-bearing capacity and selection of sections in the elements of steel structures is performed from the **Steel** toolbar. The following toolbars contain the controls of these operations:

— input data;

— results for steel.

The design combinations of forces (DCF) must be specified, otherwise this mode will not be available. You should also make sure that the codes specified at the generation of DCF correspond to the codes used in the analysis of elements of steel structures.

Input the Initial Data

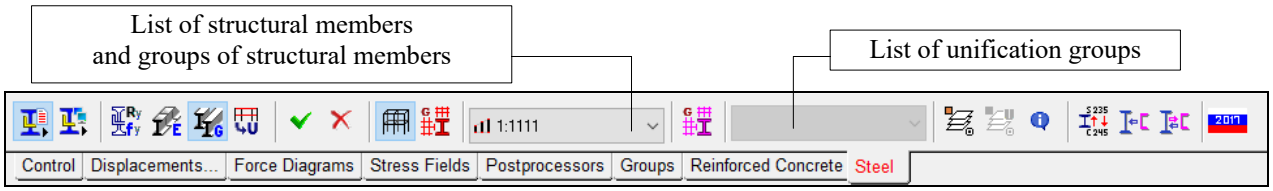


Figure 20-1. *The* **Input data** *toolbar for the calculation of the load-bearing capacity of steel sections*

The **Input data** toolbar (Fig. 20-1) includes the buttons for invoking the following operations:

— set parameters;

— specify structural members;

— specify groups of structural members;

— specify unification groups;

— confirm the selection of structural members;

— cancel the current mode;

— display design model;

— control the groups of steel structural members;

— control the unification groups;

— show active group (here purple color is used both in the image of the icon and in the model unlike the group selection button where the red color is used);

— specify steel;

— information on the groups of steel structural members;

— show modified sections;

— select active group; — restore initial sections;

— select the design codes.

The check of the load-bearing capacity of steel sections is performed for **structural members** or **groups of structural members**.

Structural member models a physically homogeneous element of a structure, for example, a column of a frame, an undercrane (overcrane) part of the column, a plate girder, a truss chord etc.

The geometric length of a structural member is equal to the sum of the lengths of its constituent finite elements.

A continuous chain of bar finite elements with the following properties is considered as a structural member:

- elements included in the chain lie on one straight line without gaps (the accuracy is defined by the parameter of matching nodes);
- all elements of the chain have the same type of rigidity;
- all finite elements of the chain have the same type;
- all elements of the chain have the same orientation of the principal axes of the cross-section;
- elements of the chain do not have rigid inserts and hinges (hinges are allowed only in the first and the last node of the chain);
- **elements of the chain can be included only in one structural member**.

In some cases **a structural member can consist of only one bar finite element**.

When it is necessary to consider a number of individual bar finite elements (for example, verticals or diagonals of a truss), they can be combined into **groups of structural members**, and the check of the load-bearing capacity is performed for the entire structural group. All elements included in a group must have the same section type. Moreover, the same effective length factors (or effective lengths) are used for all elements of a structural group.

Since there are special requirements listed above to the structural members and groups, and their control is performed in the process of specification and calculation, in this case the usual groups of elements are not used.

Design combinations of forces (DCF) must be calculated for all elements included in the structural members and groups of structural members.

Structural members and groups of structural members (hereinafter — structural groups) can be created beforehand in the preprocessor or in the postprocessor in the **Groups** toolbar (the **Elements of steel structures** operation —). If when switching to the check of sections the data of the structural groups created earlier do not have to be corrected, you can skip the data input mode, and proceed to the analysis of sections at once. Otherwise, the structural groups are created and/or corrected in the **Input data** section of the **Steel** toolbar.

20.1. Setting the Parameters

The initial setting of the parameters is performed in the **Settings** dialog box (Fig. 20.1-1). The data specified in the dialog box are automatically assigned to all newly created structural members and groups of structural members. In those cases when a structural member or a group has values of parameters different from those specified in the dialog box, these parameters are specified in the **Structural Member** or **Group of Structural Members** dialog boxes.

The following values are taken by default: service factor — 1.0, limit slenderness of members in compression— 180, limit slenderness of members in tension — 300. The specified service factor must be in the range 0.7…1.2.

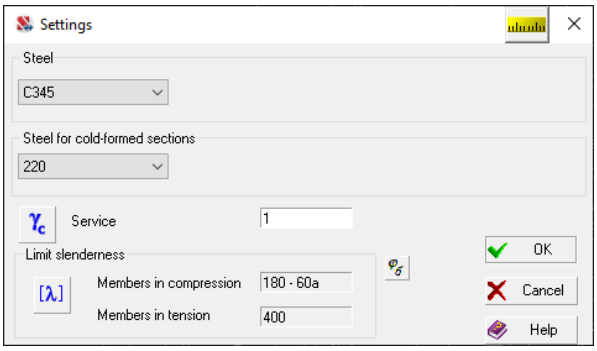


Figure. 20.1-1. *The* **Settings** *dialog box*

Steel

Steel is selected from the respective list. If for some reason it is necessary to specify a nonstandard value of the design strength, select the **Other** (steel) line in the list of steels and specify the value of $\mathbf{R_y}$ in the text field. In this case the design strength will not depend on the type and thickness of the rolled profile.

Steel for cold-formed cross-sections is selected in the same way.

Service Factors

Clicking the button invokes the dialog box (see Fig. 20.1-2), where you can select a standard (specified by the codes) value of the service factor and transfer this value into the text field by clicking the **Apply** button.

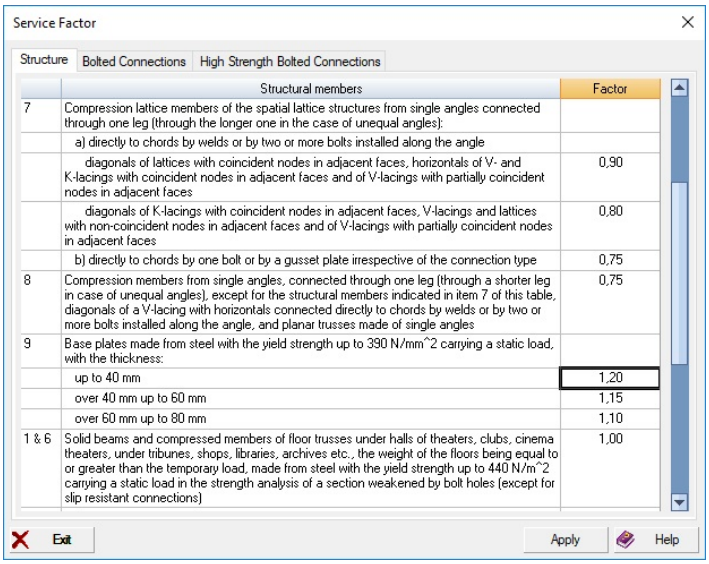
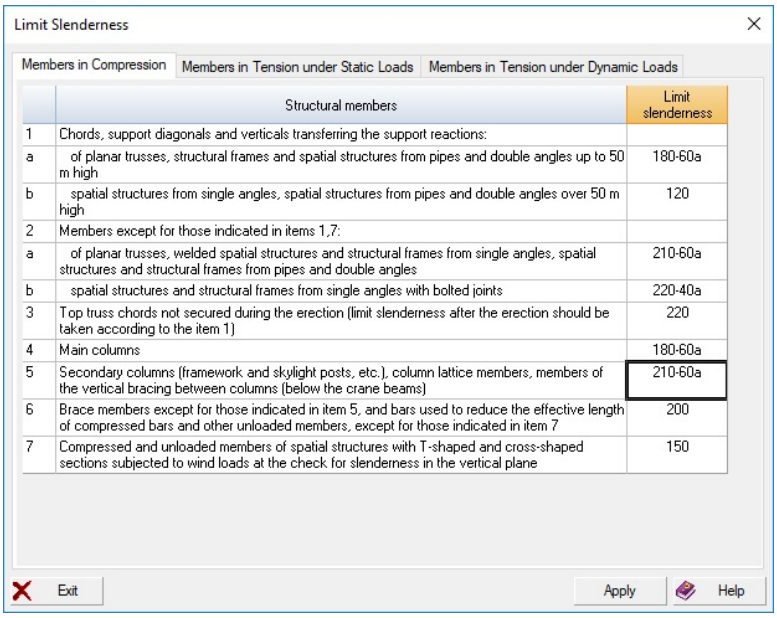


Figure 20.1-2. *The* **Service Factor** *dialog box*

If EN 1993 is selected for designing steel structures, this control is not available.

Limit Slenderness

Clicking the button invokes the dialog box (see Fig. 20.1-3), where you can select a standard (specified by the codes) value of the limit slenderness and transfer this value into the text field by clicking the **Apply** button.



	Structural members	Limit slenderness
1	Chords, support diagonals and verticals transferring the support reactions:	
a	of planar trusses, structural frames and spatial structures from pipes and double angles up to 50 m high	180-60a
b	spatial structures from single angles, spatial structures from pipes and double angles over 50 m high	120
2	Members except for those indicated in items 1,7:	
a	of planar trusses, welded spatial structures and structural frames from single angles, spatial structures and structural frames from pipes and double angles	210-60a
b	spatial structures and structural frames from single angles with bolted joints	220-40a
3	Top truss chords not secured during the erection (limit slenderness after the erection should be taken according to the item 1)	220
4	Main columns	180-60a
5	Secondary columns (framework and skylight posts, etc.), column lattice members, members of the vertical bracing between columns (below the crane beams)	210-60a
6	Brace members except for those indicated in item 5, and bars used to reduce the effective length of compressed bars and other unloaded members, except for those indicated in item 7	200
7	Compressed and unloaded members of spatial structures with T-shaped and cross-shaped sections subjected to wind loads at the check for slenderness in the vertical plane	150

Figure 20.1-3. *The* **Limit Slenderness** *dialog box*

If EN 1993 is selected for designing steel structures, this control is not available.

Creating New Structural Groups

The following procedure is recommended for preparing the new structural groups:

- select elements included in the structural group in the model;
- select the **Input data** operation — in the **Steel** section of the toolbar and the toolbar for the preparation of the initial data for the analysis of elements of steel structures will appear;
- click one of the buttons corresponding to the type of the created group:
- — **Specify structural members**;
- — **Specify groups of structural members**.
- click the button — **Confirmation** in the toolbar;
- the **Structural Member** (Fig. 20.1-1) or **Groups of Structural Members for Checking Sections** dialog box will appear, where you have to specify the parameters of the group.

20.2. Specifying New Structural Members

The following characteristics of the member are specified in the **Structural Member** dialog box: name of the member (must be specified), type of the structural member, effective length factors, length between restraints out of the bending plane, importance factor, and steel, service factors and limit slenderness (the last three parameters correspond to those specified in the **Settings** dialog box by default). If these parameters have been changed, the structural member will have the characteristics different from those specified for the whole model.

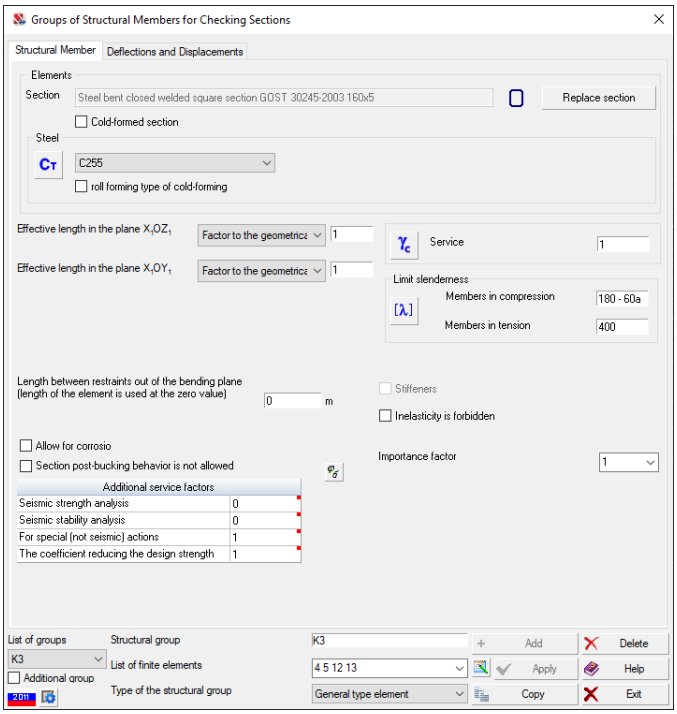


Figure 20.2-1. *The* **Structural Member** *dialog box*

Effective Length Factors

Geometric length of a structural member is multiplied by the effective length factors, which are used to determine the effective lengths of a structural member (physically homogeneous bar) in the planes of the principal axes of inertia of the cross-section. They should be specified based on the length between restraints out of the bending plane of the element in a real structure and the type of this restraint (rigid or elastic support, constraining angular or linear displacements).

You can specify it either in the form of the effective length factor or the effective length, which can be selected from the drop-down list next to the text field.

Critical Moment

In case of analysis per EN 1993 and SP 2601325800 the **Critical Moment** tab (Fig. 20.2-2) is used to specify data for calculation of critical moment.

The calculation of the critical moment is performed using the analytical solution given in Annex F ENV 1993-1-1 and SP 260.1325800, depending on the character of the bending moment diagram, different boundary conditions of the bar element, restraints against warping, load application with respect to the shear center of the section, and also depending on the asymmetry of the section with respect to the axis of greater stiffness. Moreover, the calculation of the critical moment takes into account the pre-buckling curvature in accordance with *Designers Guide to Eurocode 3: Design of Steel Buildings, EN 1993-1-1, EN 1993-1-3, EN 1993-1-8, ICE Publishing, 2011*.

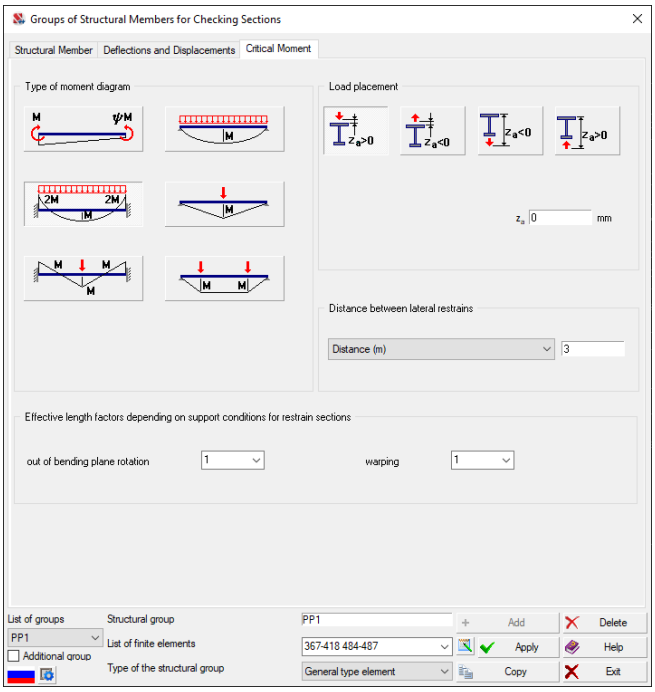


Fig. 20.2-2. *The* **Critical Moment** *tab*

The calculations are performed only for the cross-sections symmetric with respect to the axis of the lower stiffness at their bending with respect to the axis of the greater stiffness. Moreover, the codes limit the use of this check method for the sections which are characterized by a significant rotation of the principal axes of inertia of the "effective" section with respect to the principal axes of inertia of the gross cross-section.
The respective limitations are introduced in the software as well. Thus, if the user has selected a section which is not symmetric with respect to the axis of the lower stiffness at its bending with respect to the axis of the greater stiffness, it is assumed that the stability of in-plane bending of the considered bar member is ensured by the appropriate restraints.

In addition, to calculate the critical moment, two effective length factors should be specified. The factor k refers to end rotation on plan and the factor k_w refers to end warping. Recommendations for assigning the coefficients k, k_w for some special cases can be found in the books

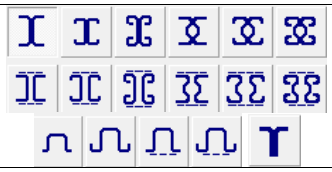
- L. Gardner, *Stability of Steel Beams and Columns: In Accordance with Eurocodes and the UK National Annexes*, Steel Construction Institute, 2011.
- N.Boissonnade, R.Greiner, J.P.Jaspart, J.Lindner, *Rules for member stability in EN 1993-1-1 : Background documentation and design guidelines*, ECCS European Convention for Constructional Steelwork, 2006.

The buttons of the **Type of moment diagram** group enable to specify a bending moment diagram type for a bar member. Since SP 260.1325800 provides guidelines for determining the critical moment for only four types of bending moment diagrams, when the analysis is performed according to SP 260.1325800, this group contains only the respective four buttons.

If the first option is selected, you should also specify the parameter ψ, which characterizes the ratio of the end moments. The effective length factor for the lateral-torsional buckling has to be specified in the **l/L for a compressed chord** field. Moreover, the distance from the load application point to the top chord level, z_a, should be specified. The sign of z_a is assigned according to a schematic given in this tab.

The lateral-torsional buckling analysis depends on the cross-section type of the bar member (see Table 20.2.1).

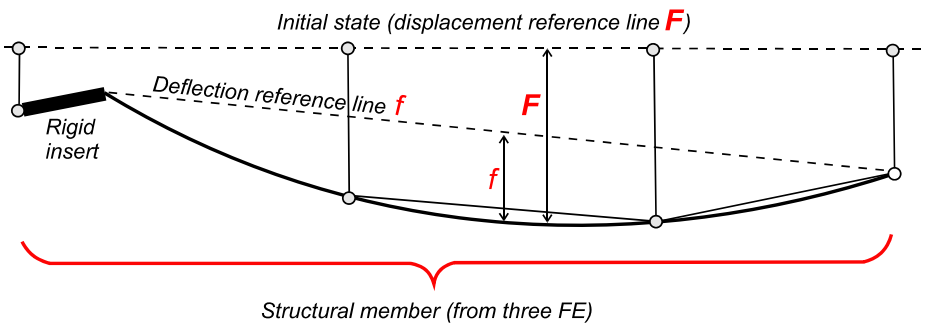
Table 20.2.1. The lateral-torsional buckling analysis for bar members with various cross-section types

Cross-section type of the bar member	A response of the **Critical Moment** mode to the section type
	It is assumed that the stability of in-plane bending is provided by the respective restraints of the compressed chord of the flexural member
	The lateral-torsional buckling analysis is performed
	The codes do not provide any recommendations on the determination of the critical moment and on the lateral-torsional buckling analysis
	It is assumed that lateral-torsional buckling does not occur

Accounting for the Maximum Allowable Deflections and Displacements

The maximum allowable horizontal and vertical deflections and displacements are taken into account in accordance with the code selected for the analysis. For example, if SP 20.13330.2011 is used, the checks are performed in accordance with Section F.2.1 of Annex F, and when you are working with DSTU B.B.1.2-3: 2006 (Deflections and displacements) — according to item 5.1 of Section 5.

The deflection reference line is a straight line that connects the beginning and the end of the flexible part of a structural member. The figure below shows an example where the structural member consists of three finite elements.



The check is performed on the basis of the parameters selected in the respective table in the **Deflections and Displacements** tab.

Maximum allowable displacements and deflections:	Relavite k*L	Absolute mm
vertical deflection for all loads	Automatically	0.7
vertical deflection for permanent and long-term loads	0.00833	0.7
vertical deflection only for live loads	0.00833	0.7
vertical displacements for all loads	0.00667	0.7
vertical displacements for permanent and long-term lo	0.00667	0.7
vertical displacements for live-loads	0.00667	0.7
horizontal deflection for all loads	0.00833	0.7
horizontal deflection for permanent and long-term loa	0.00833	0.7
horizontal deflection only for live loads	0.00833	0.7
horizontal displacements	Unlimited	Unlimited
horizontal displacements for permanent and long-term	Unlimited	Unlimited
horizontal displacements only for live loads	Unlimited	0.7

You can specify both relative and absolute values of the allowable deflections and displacements. In the first case, the following variants of the accounting for displacement are considered (the variants can be selected from the drop-down list of the respective column of the table):

- **Unlimited** — deflections and displacements are not taken into account in the check;
- **Automatically** — relative (with respect to the length of the structural member) deflections and displacements are calculated by the software depending on the length of the member; the range of lengths and corresponding limit displacements are determined by the code;
- the limit value is selected in the list.

The absolute values are specified by the user or are not taken into account if the "**Unlimited**" variant is selected.

If both relative and absolute limitations are simultaneously specified, the value resulting in a greater coefficient is considered when calculating the utilization factor of restrictions.

It should be noted that the automatic check by the relative value makes sense only in those cases where a structural member has no constraints in the span between the end nodes.

Taking into account the fact that there are situations when the designer is interested in deflections and displacements caused only by the variable actions (for example, the check of the instability of the structure, or the case when the displacements from the self-weight are counteracted by the camber), the software enables to perform the checks for all loads, or, separately for only the temporary ones. The type of calculation is selected by checking the checkboxes in the respective lines and activating the design combinations. In any case the checks are performed taking into account the design combinations of deflections and displacements obtained on the basis of the characteristic values of loads.

Maximum allowable displacements and deflections are taken into account only in the check mode and are not used when selecting the sections.

Importance Factors

All design modes require data on the importance factor according to the selected safety codes (GOST 27751-88, GOST 27751-2014, DBN B.1.2-14-2009, …).

The following importance factors are used in the analysis:

- ultimate limit state importance factor γ_n;
- serviceability limit state importance factor;
- importance factor for accidental/special design situations (it is also used in the progressive collapse and fire resistance analyses).

The factor γ_n can be selected from a drop-down list in accordance with the characteristic of a structure (it can be of *special*, *essential* or *limited* importance) or entered by the user (in non-standard situations).

DBN B.1.2-14-2009 provides three importance factors (for the ultimate limit state, for the serviceability limit state and for accidental situations). The user can specify each of these three factors. GOST 27751-88 provides only one factor, which is used in all design situations. If GOST 27751-2014 is selected as a safety code, the user-defined factor γ_n is used in the ultimate limit state analyses. The following value is assumed for the serviceability limit state analyses γ_n=1.0 (but not greater than γ_n for the ultimate limit state in order to avoid a situation in which the characteristic values turn out to be greater than the design ones). And the following value is used for the analysis of special design situations γ_n =1.0 (see Sec. 5.5 of SP 296.1325800.2017).

Replacing a Section of a Structural Member

The **Replace section** button is used to replace a section of elements included in a structural member.

In the latter case the check of the load-bearing capacity will be performed for a section different from that specified in the stiffness properties **(an automatic recalculation of the modified model is not performed!**). Moreover, you should also remember about the unification, since this member could be included in one of the unified groups.

Type of the Structural Member

The following types can be assigned to structural members:

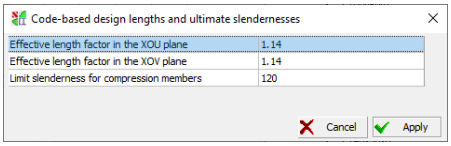
- general type element (or column);
- beam (element subjected to uniaxial bending with respect to the Y axis);
- truss chord member;
- truss lattice member;
- support diagonal;
- support vertical.

If cross-section is a a single angle section, you can also specify the following types of the structural member:

- the leg member of a space lattice structure with symmetrical bracing;
- the leg member of a space lattice structure with unsymmetrical bracing;
- the bracing member of a space lattice structure;
- the member of a space structure.

Based on the selected type, software will automatically perform the stability analysis in the principal axes or structural axes according to the requirements of the selected design codes.

Clicking the button invokes the dialog box which provides the standard values of the effective lengths and limit slenderness recommended by the codes. Clicking the **Apply** button will transfer this information to the calculation parameters.



A set of sections the check is performed for depends on the type of a structural member and is given in Table 20.2.2.

Table 20.2.2.

Member type	Set of sections	Internal forces and moments taken into account in the check
General type element		N, M_y, Q_z, M_z, Q_y
Column		
Beam		M_y, Q_z
Truss lattice member		N
Truss chord member		
Support diagonal		
Support vertical		

The selected type affects the set of data which have to be specified for the member. For example, the information on limit slenderness is not required for truss members (these data are specified in SNiP); the information on the effective lengths is not required for beams etc. The type defines not only a set of checks according to SNiP, but also the forces which will be taken into account when performing the check of sections. For example, a longitudinal force will be ignored for beams, even if it appeared during the calculation. Bending moments in the following types of structural members “Truss lattice member”, “Truss

chord member", "Support diagonal" and "Support vertical" are neglected, assuming that a bar element is a part of a hinged-bar statically determinate system – truss, but a special checkbox enables to take into account the bending moments and shear forces as well. In this case the calculation is similar to that performed for a general type element, but the values of effective length and limit slenderness are selected automatically in accordance with the recommendations of the codes for truss members.

The *length between restraints out of the bending plane* has to be specified for general type elements and beams (this value will be used in the analysis of stability of in-plane bending). If this value is set to zero, the geometric length of the element will be used. If EN 1993 is selected for designing steel structures, this control is not available.

If the transverse stiffeners can be installed for the given cross-sections, you can use the **Stiffeners** checkbox, thus indicating that the stiffeners are installed, and specify their spacing. If this spacing is greater than the length of the element, the local stability analysis of the web is performed as for a web without stiffeners. If the web has such a slenderness that the element can be classified as an element with a flexible web, the application outputs the **Critical ratio between height and width of the web** factor with a value greater than 1,0. The calculation of elements with a flexible web is not implemented in the program due to the extremely limited scope of the method for calculating such structures (only for continuous beams bearing the static load).

Two possible strength checks of the general type elements taking a longitudinal force and a bending moment are recommended by the design codes for steel structures:

- allowing for the possible elastic-plastic behavior;
- when the inelasticity is forbidden.

The possibility of the elastic-plastic behavior is limited by a number of conditions, for example, such as the absence of direct action of dynamic loads. The **Inelasticity is forbidden** checkbox is provided for the case when there is direct action of the dynamic loads, and also for those cases when the user for some reason does not want to go beyond the elastic behavior. Automatic rejection of the elastic-plastic calculation in the case when DCF contain dynamic loadings is not provided, since SNiP does not clearly define the concept of "direct action of the dynamic loads". If EN 1993 is selected for designing steel structures, this control is not available, and the respective option can be selected in the settings in the **EN 1993 Settings** tab (see Sec. 34.7).

The **Web instability is forbidden** checkbox is used to perform the check of the section taking into account its postbuckling behavior (after the local buckling of the web). The checked checkbox enables to reject the postbuckling behavior of the section if the check indicates local buckling of the web.

According to the design codes the structural members of the **Beam** type, which take only a bending moment and a shear force, have to be checked for strength and lateral torsion buckling depending on the beam class:

– for sections of beams of the 1st class – within the limits of elastic deformation of steel;

– for sections of beams of the 2nd class – within the limits of elastoplastic deformation of steel.

The codes allow the elastoplastic behavior only for simply supported beams. If the **Simply supported** checkbox is checked and the section has been classified to the 2-nd class after the analysis of its normal stresses, the checks of strength and lateral torsion buckling will be performed taking into account the elastoplastic deformations. The **Inelasticity is forbidden** checkbox is provided for the case when the user for some reason does not want to go beyond the elastic behavior. If the analysis of steel structures is performed according to EN 1993, there is no **Simply supported** checkbox.

Since different groups of elements can have different degree of responsibility, the importance factors are specified when creating a structural member and can be different for different structural members.

After specifying all the data on the structural member including its name, click the **Add** button.

Corrosion analysis

Check calculations can be performed taking the corrosion into account. In order to do it you have to check the respective checkbox and specify the thickness of the corrosion layer. When performing the calculations it is assumed that the thickness of the corrosion layer is the same over the whole perimeter of the section.

The corrosion is obviously not taken into account when selecting the sections. If EN 1993 is selected for designing steel structures, this control is not available.

Advanced Settings

There is a button next to the icon indicating the selected design codes which invokes a special dialog box for specifying additional restrictions for selecting steel sections (Fig. 20.2-3).

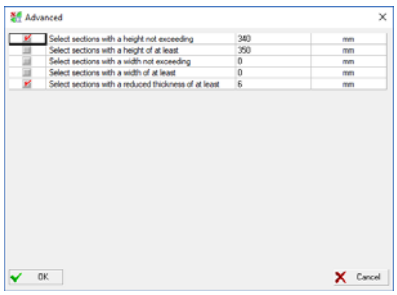


Figure 20.2-3. *The* **Advanced Settings** *dialog box*

In particular, you can specify the restrictions on the height, width and reduced thickness of a section (if the fire resistance analysis is enabled).

Copying the Parameters of Structural Members

If all or most of the parameters of a new structural member are the same as those of a member created earlier, you can copy the properties of the existing member and assign them to the new one. It is necessary to perform the following actions to copy the parameters:

- invoke the operation of creating a structural member;
- select finite elements in the model which make up a new structural member;
- click the **Confirmation** button in the toolbar;
- click the **Copy** button in the **Structural Member** dialog box and select a member the parameters of which you want to copy in the list that will appear;
- correct, if necessary, the parameters of the new member including its name;
- click the **Add** (new member) button;
- close the dialog box by clicking the **Apply** button.

The **Copy** operation will be available if the model has structural members created earlier.

Control of Structural Members

Before including a new structural member into the list of structural members that have to be checked, the control of the correctness of the specification is performed. If in the result of the control it turns out that the selected elements do not meet the above requirements to finite elements which make up a structural member the **Result of the Performed Operation** dialog box appears (Fig. 20.2-4).

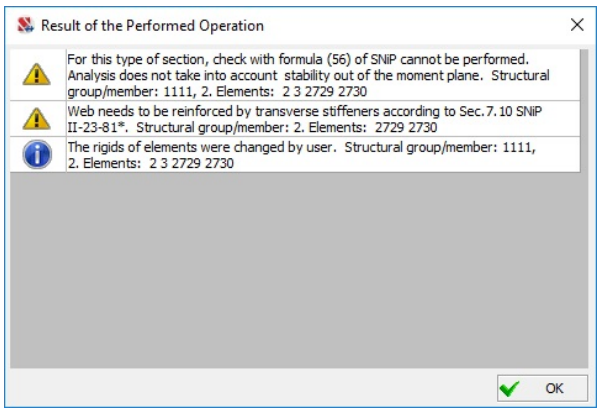


Figure 20.2-4. *The* **Result of the Performed Operation** *dialog box*

The options indicating the mistakes made when specifying a structural member will be available in the list of possible errors given in this dialog box. After closing this dialog box all messages will appear in the log with the numbers of elements of the model where these mistakes were made. Using the selection buttons in the log you can perform the visualization of the elements in the model and their selection.

20.3. Specifying Groups of Structural Members

In those cases when a structural member has only one bar, you can combine these bars into groups and perform a check, and when necessary a unification of groups. Operations of specifying groups of structural members and their characteristics are similar to those described above for structural members. The characteristics of groups are specified in the **Groups of Structural Members for Checking Sections** dialog box (Fig. 20.3-1) similar to the dialog box for creating a structural member.

You should keep in mind that the section replacement is performed for all structural members of the group. Thus, after performing this operation all structural members of the group will have the same section.

When the errors made when creating a group of structural members are detected, a dialog with a list of errors (Fig. 20.3-2) appears after clicking the **Add** button. The rules for working with this dialog box are the same as the rules for working with a dialog box with errors for structural members.

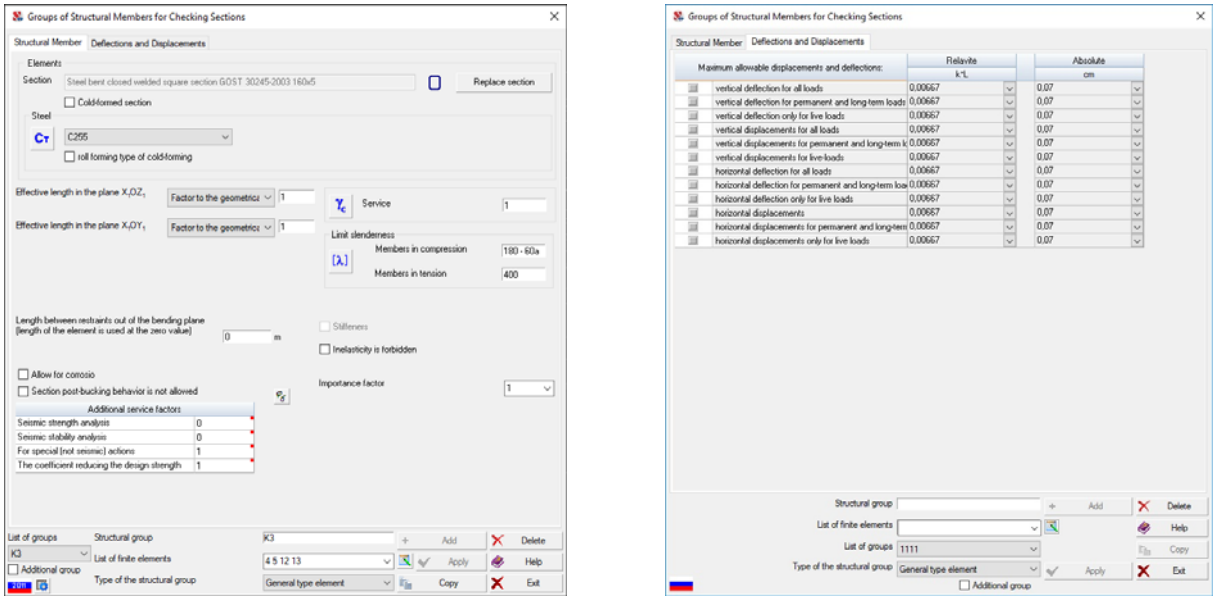


Figure 20.3-1. *The* **Groups of Structural Members for Checking Sections** *dialog box (two pages)*

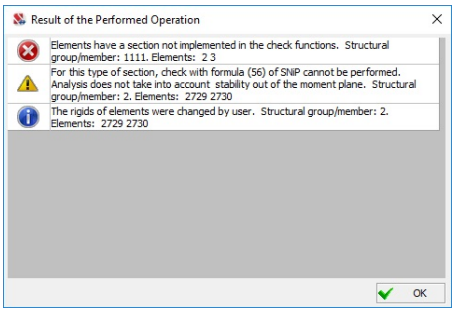


Figure 20.3-2. *The* **Result of the Performed Operation** *dialog box*

Accounting for the Maximum Allowable Displacements

Accounting for the maximum allowable vertical displacements for groups of structural members is performed according to the same rules as for structural members (Section 20.2). The difference is that the

maximum value of displacements is selected for each finite element included in the group by analyzing the values of displacements of nodes and deflections.

20.4. Correcting the Parameters of Structural Members and Groups of Structural Members

Any of the initially specified parameters of structural members or groups of structural members (including the set of their constituent finite elements) can be changed during the check or selection.

If the changes deal with the composition and parameters of a structural member or a group, after making these changes click the **Apply** button on the right from the list of structural members and their groups. The **Replace section** button mentioned above is used to replace a section.

The changes in the composition of structural members and their groups are made in a similar way as the specification. The following procedure is used:

- select the name of the member or group you want to correct in the list of the **Input data** toolbar;
- click the **Select active group** button— , and the finite elements included in the structural member or group will be selected or highlighted in red;
- select finite elements with the cursor you want to exclude from the structural group (these elements will be deselected);
- click the **Confirmation** button — in the toolbar, and the respective dialog box will appear (**Structural Member** or **Groups of Structural Members…**);
- click the **Apply** button in this dialog box.

Finite elements can be also added to the created structural member in a similar way.

The respective button is used to edit the existing list of the group, clicking which invokes a dialog box with a list of elements.

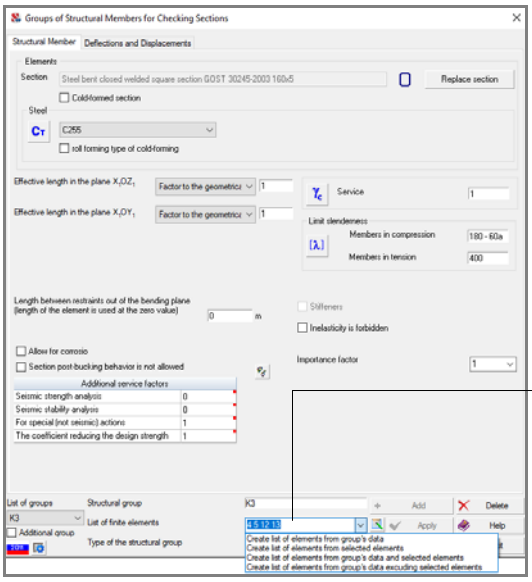


Figure 20.4-1. *Editing the list of elements included in a structural group*

The lower part of the dialog box (Fig. 20.4-1) contains a list of *finite elements* with numbers of elements included in a structural member or group. This list can be edited directly in the dialog box. If there are selected elements in the model, you can perform the following actions with the help of a set of operations which can be opened by clicking the button on the right from the list:

- **Create list of elements from selected elements**, i. e. actually replace the list of elements created earlier by the list of elements selected in the model;
- **Create list of elements from group's data and selected elements**, i. e. add the elements selected in the model to the list of elements created earlier;

- **Create list of elements from group's data excluding selected elements**, i. e. exclude the elements selected in the model from the list of the group created earlier.

If after performing a series of operations with the list of elements (before clicking the **Apply** button), it is necessary to return to its initial state, you should use the first operation in the considered list — **Create list of elements from group's data**, and the initial list of the group will be restored.

In order to delete a structural member or a group of structural members it is necessary to select a name from the list in the **Input data** toolbar and click the button . Click the **Delete** button in the dialog box that will appear.

If you have to delete several objects, it is necessary to perform the above operations for a single object, and then consecutively select the objects you want to delete from the list of structural members (or groups) and click the **Delete** button without closing the dialog box.

Copying the Parameters of Groups of Structural Members

If all or most of the parameters of a new group of structural members are the same as those of a group created earlier, you can copy the properties of the existing group and assign them to the new one. It is necessary to perform the following actions to copy the parameters:

- invoke the operation of creating a group of structural members;
- select finite elements in the model which make up a new group;
- click the **Confirmation** button in the toolbar;
- click the **Copy** button in the **Groups of Structural Members…** dialog box and select a group the parameters of which you want to copy in the list that will appear;
- correct, if necessary, the parameters of the new group including its name;
- click the **Add** (new group) button;
- close the dialog box by clicking the **Apply** button.

The **Copy** operation will be available if the model has groups of structural members created earlier.

20.5. Main and Additional Groups

If a finite element simultaneously belongs to several structural members (or groups of structural members), it causes many problems and questions. For example, if you perform a selection of sections, the results of which group should be considered as the result of selection? Which results should be used when assigning the rigidities according to the results of the selection? Which data should be used to calculate the value of K_{max} after performing the check when displaying the utilization factor of restrictions for the whole model and not for an individual structural member?

At the same time, in some cases it is convenient to be able to create several structural members (or groups) which have the same set of finite elements, but different parameters (for example, steel grade or service factors). This enables to perform the comparison of results using a multi-tab mode, and make a reasoned decision.

You can create additional structural groups (Fig. 20.5-1). This enables to work with a set of structural groups with matching finite elements. There are the following limitations when using the main and additional groups:

- a finite element can not belong simultaneously to two main groups;
- additional groups are not used when determining K_{max} for the whole model;
- additional groups are not used when assigning the rigidities according to the results of the selection;
- if a finite element simultaneously belongs to the main and additional group/groups, after clicking the buttons **Steel. Factors**, **Resistance of sections**, ... in the information on the element, a menu appears with the list of groups and you have to select a group the results of which you want to be displayed;
- additional groups are not used in the analysis for the progressive collapse.

Additional groups are distinguished by the green background of icons in the lists of structural groups in toolbars and dialog boxes. Unique names should be given when creating the main and additional groups.

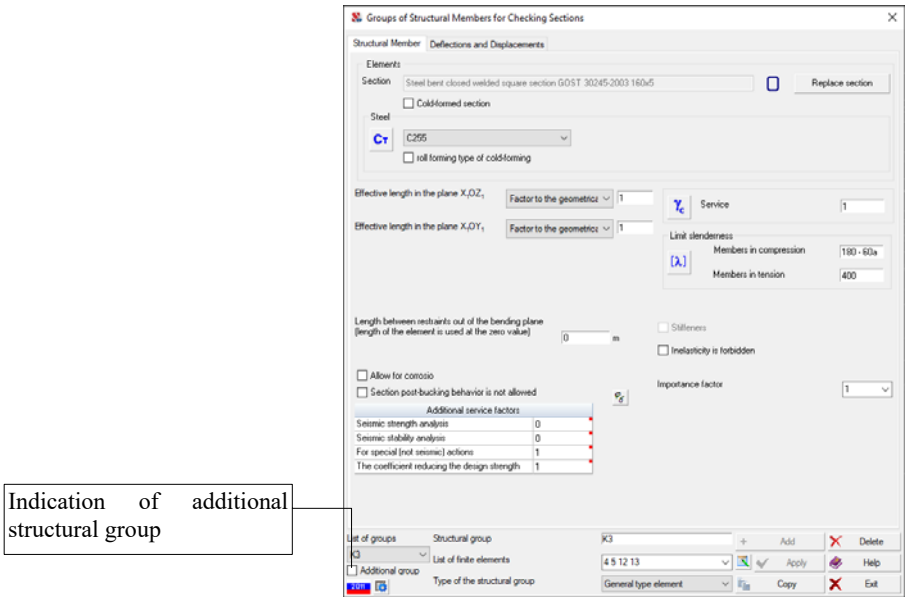


Figure. 20.5-1. *The* **Groups of Structural Members for Checking Sections** *dialog box (additional group)*

20.6. Analysis of cold-formed profiles

If a certain group consists of cold-formed steel members, you have to check the **Cold-formed section** checkbox to perform their analysis. In this case, if EN 1993 is selected as the design code for steel structures, then the analysis of the members of this group will be performed according to the requirements of EN 1993-1-3. If, for example, SP 16.13330 is selected, then the analysis of cold-formed sections will be performed according to SP 260.1325800.

For cold-formed steel members you can also use the **Not subject to torsion** and **Not subject to warping** checkboxes. The first option assumes that the element is reliably restrained by torsional constraints along its entire length and the torque will not be taken into account in the analysis (including the torsion caused by the shear force at the eccentricity of the bending center with respect to the center of mass). When the second option is selected, free warping of the support sections of the element is assumed, so the bimoment and the constrained torque will not be taken into account in the analysis.

The following table provide lists of checks for members with cold-formed cross-sections.

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
Tensile strength of the member subject to axial force N	Sec. 6.2.3 (6.5)	Sec. 6.1.2 (6.1)	Sec. 7.7.2.1 (7.68)
Compression strength of the member subject to axial force N	Sec. 6.2.4 (6.9)	Sec. 6.1.3 (6.3)	Sec. 7.7.2.2 (7.69)
Post-buckling compression strength of the member subject to axial force N	Sec. 6.2.4 (6.9)	Sec. 6.1.3 (6.2)	Sec. 7.7.2.2 (7.69)
Elasto-plastic bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.5)	not calculated
Elastic bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.6)	Sec. 7.7.3 (7.75)
Post-buckling bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.4)	Sec. 7.7.3 (7.74)
Elasto-plastic bending strength of the member subject to bending moment Mz	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.5)	not calculated
Elastic bending strength of the member subject to bending moment Mz	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.6)	Sec. 7.7.3 (7.75)
Post-buckling bending strength of the member subject to bending moment Mz	Sec. 6.2.5 (6.12)	Sec. 6.1.4 (6.4)	Sec. 7.7.3 (7.74)
Elasto-plastic bi-axial bending strength of the member		Sec. 6.1.4 (6.5), (6.7)	not calculated
Elastic bi-axial bending strength of the member		Sec. 6.1.4 (6.6), (6.7)	Sec. 7.7.3 (7.76)
Post-buckling bi-axial bending strength of the member		Sec. 6.1.4 (6.4),	Sec. 7.7.3 (7.76)

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
		(6.7)	
Shear strength of the member subject to shear force Qz	Sec. 6.2.6 (6.17)	Sec. 6.1.5 (6.8)	Sec. 7.7.6 (7.79)
Shear strength of the member subject to shear force Qy	Sec. 6.2.6 (6.17)	Sec. 6.1.5 (6.8)	Sec. 7.7.6 (7.79)
Bending strength of the member subject to bending moment My taken into account shear force Qz	Sec. 6.2.8 (6.29)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.75), 7.7.5 (7.78)
Post-buckling bending strength of the member subject to bending moment My taken into account shear force Qz	Sec. 6.2.8 (6.29)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.74), 7.7.5 (7.78)
Bending strength of the member subject to bending moment Mz taken into account shear force Qy	Sec. 6.2.8 (6.29)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.75), 7.7.5 (7.78)
Post-buckling bending strength of the member subject to bending moment Mz taken into account shear force Qy	Sec. 6.2.8 (6.29)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.74), 7.7.5 (7.78)
Strength of the member subject to combined axial force N and bending moments My, Mz		Sec. 6.1.8 (6.23), (6.24), 6.1.9 (6.25), (6.26)	Sec. 7.7.4 (7.77)
Post-buckling strength of the member subject to combined axial force N and bending moments My, Mz		Sec. 6.1.8 (6.23), (6.24), 6.1.9 (6.25), (6.26)	Sec. 7.7.4 (7.77)
Strength of the member subject to combined axial force N, bending moments My, Mz and shear forces Qz, Qy		Sec. 6.1.10 (6.27)	Sec. 7.7.4 (7.77), 7.7.5 (7.78)
Post-buckling strength of the member subject to combined axial force N, bending moments My, Mz and shear forces Qz, Qy		Sec. 6.1.10 (6.27)	Sec. 7.7.4 (7.77), 7.7.5 (7.78)
Strength by the direct stresses in the member due to combined action of normal force N, bending moments My, Mz and warping B	–	Sec. 6.1.6 (6.11a)	Sec. 7.7.7 (7.82)
Post-buckling strength by the direct stresses in the member due to combined action of normal force N, bending moments My, Mz and warping B	–	Sec. 6.1.6 (6.11a)	Sec. 7.7.7 (7.82)
Strength by the shear stresses in the member due to combined action of shear forces Qz, Qy, uniform (St. Venant) torsion T and warping torsion Mw	–	Sec. 6.1.6 (6.11b)	Sec. 7.7.7 (7.83)
Strength by the equivalent stresses in the member due to combined action of normal force N, bending moments My, Mz, shear forces Qz, Qy, warping B, uniform (St. Venant) torsion T and warping torsion Mw	–	Sec. 6.1.6 (6.11c)	Sec. 7.7.7 (7.84)
Post-buckling strength by the equivalent stresses in the member due to combined action of normal force N, bending moments My, Mz, shear forces Qz, Qy, warping B, uniform (St. Venant) torsion T and warping torsion Mw		Sec. 6.1.6 (6.11c)	Sec. 7.7.7 (7.84)
Buckling strength of the member subject to flexural buckling relating to axes y-y due to axial force N	Sec. 6.3.1 (6.46) – (6.51)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the member subject to flexural buckling relating to axes y-y due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.51)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the member subject to flexural buckling relating to axes z-z due to axial force N	Sec. 6.3.1 (6.46) – (6.51)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the member subject to flexural buckling relating to axes z-z due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.51)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the member subject to torsional and torsional-flexural buckling due to axial force N	Sec. 6.3.1 (6.46) – (6.49), (6.52), (6.53)	Sec. 6.2.3	Sec. 7.7.8 (7.87), (7.89) – (7.92)
Buckling strength of the member subject to torsional and torsional-flexural buckling due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.49), (6.52), (6.53)	Sec. 6.2.3	Sec. 7.7.8 (7.87), (7.89) – (7.92)
Elasto-plastic buckling strength of the member subject to lateral-torsional buckling due to bending moment	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	—

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
My			
Buckling strength of the member subject to lateral-torsional buckling due to bending moment My	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the member subject to lateral-torsional buckling due to bending moment My (post-buckling behavior)	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Elasto-plastic buckling strength of the member subject to lateral-torsional buckling due to bending moment Mz	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	—
Buckling strength of the member subject to lateral-torsional buckling due to bending moment Mz	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the member subject to lateral-torsional buckling due to bending moment My (post-buckling behavior)	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the member subjected to axial force N and bending moments My, Mz	Sec. 6.3.3 (6.61) – (6.62)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the member subjected to axial force N and bending moments My, Mz (post-buckling behavior)	Sec. 6.3.3 (6.61) – (6.62)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the member subjected to axial force N and bending moments My, Mz taken into account lateral-torsional buckling	Sec. 6.3.3 (6.61) – (6.62)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the member subjected to axial force N and bending moments My, Mz taken into account lateral-torsional buckling (post-buckling behavior)	Sec. 6.3.3 (6.61) – (6.62)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Chord's tensile strength of the member subject to axial force N	Sec. 6.2.3 (6.5), 6.4.1 (6.69)	Sec. 6.1.2 (6.1)	Sec. 7.7.2.1 (7.68)
Chord's compression strength of the member subject to axial force N	Sec. 6.2.4 (6.9), 6.4.1 (6.69)	Sec. 6.1.3 (6.3)	Sec. 7.7.2.2 (7.68)
Chord's post-buckling compression strength of the member subject to axial force N	Sec. 6.2.4 (6.9), 6.4.1 (6.69)	Sec. 6.1.3 (6.2)	Sec. 7.7.2.1 (7.69)
Chord's elasto-plastic bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12), 6.4.1 (6.69)	Sec. 6.1.4 (6.5)	not calculated
Chord's elastic bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12), 6.4.1 (6.69)	Sec. 6.1.4 (6.6)	Sec. 7.7.3 (7.75)
Chord's post-buckling bending strength of the member subject to bending moment My	Sec. 6.2.5 (6.12), 6.4.1 (6.69)	Sec. 6.1.4 (6.4)	Sec. 7.7.3 (7.74)
Chord's shear strength of the member subject to shear force Qz	Sec. 6.2.6 (6.17), 6.4.1 (6.69)	Sec. 6.1.5 (6.8)	Sec. 7.7.6 (7.79)
Chord's shear strength of the member subject to shear force Qy	Sec. 6.2.6 (6.17), 6.4.1 (6.69)	Sec. 6.1.5 (6.8)	Sec. 7.7.6 (7.79)
Chord's bending strength of the member subject to bending moment My taken into account shear force Qz	Sec. 6.2.8 (6.29), 6.4.1 (6.69)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.75), 7.7.5 (7.78)
Chord's post-buckling bending strength of the member subject to bending moment My taken into account shear force Qz	Sec. 6.2.8 (6.29), 6.4.1 (6.69)	Sec. 6.1.10 (6.27)	Sec. 7.7.3 (7.74), 7.7.5 (7.78)
Chord's strength of the member subject to combined axial force N and bending moment My	Sec. 6.4.1 (6.69)	Sec. 6.1.8 (6.23), (6.24), 6.1.9 (6.25), (6.26)	Sec. 7.7.4 (7.77)
Chord's post-buckling strength of the member subject to combined axial force N and bending moment My	Sec. 6.4.1 (6.69)	Sec. 6.1.8 (6.23), (6.24), 6.1.9 (6.25), (6.26)	Sec. 7.7.4 (7.77)
Chord's strength of the member subject to combined axial force N, bending moment My and shear forces Qz, Qy	Sec. 6.4.1 (6.69)	Sec. 6.1.10 (6.27)	Sec. 7.7.4 (7.77), 7.7.5 (7.78)
Chord's post-buckling strength of the member subject to combined axial force N, bending moment My and shear forces Qz, Qy	Sec. 6.4.1 (6.69)	Sec. 6.1.10 Sec. (6.27)	Sec. 7.7.4 (7.77), Sec. 7.7.5 (7.78)
Chord's strength by the direct stresses in the member due to combined action of normal force N, bending	–	Sec. 6.1.6 (6.11a)	Sec. 7.7.7 (7.82)

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
moments My, Mz and warping B			
Chord's post-buckling strength by the direct stresses in the member due to combined action of normal force N, bending moments My, Mz and warping B	–	Sec. 6.1.6 (6.11a)	Sec. 7.7.7 (7.82)
Chord's strength by the shear stresses in the member due to combined action of shear forces Qz, Qy, uniform (St. Venant) torsion T and warping torsion Mw	–	Sec. 6.1.6 (6.11b)	Sec. 7.7.7 (7.83)
Chord's strength by the equivalent stresses in the member due to combined action of normal force N, bending moments My, Mz, shear forces Qz, Qy, warping B, uniform (St. Venant) torsion T and warping torsion Mw	–	Sec. 6.1.6 (6.11c)	Sec. 7.7.7 (7.84)
Chord's post-buckling strength by the equivalent stresses in the member due to combined action of normal force N, bending moments My, Mz, shear forces Qz, Qy, warping B, uniform (St. Venant) torsion T and warping torsion Mw		Sec. 6.1.6 (6.11c)	Sec. 7.7.7 (7.84)
Buckling strength of the chord subject to flexural buckling relating to axes y-y due to axial force N	Sec. 6.3.1 (6.46) – (6.51), 6.4.1 (6.69)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the chord subject to flexural buckling relating to axes y-y due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.51), 6.4.1 (6.69)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the chord subject to flexural buckling relating to axes z-z due to axial force N	6.3.1 (6.46) – (6.51), 6.4.1 (6.69)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the chord subject to flexural buckling relating to axes z-z due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.51), 6.4.1 (6.69)	Sec. 6.2.2	Sec. 7.7.8 (7.87), (7.88)
Buckling strength of the chord subject to torsional and torsional-flexural buckling due to axial force N	Sec. 6.3.1 (6.46) – (6.49), (6.52), (6.53), 6.4.1 (6.69)	Sec. 6.2.3	Sec. 7.7.8 (7.87), (7.89) – (7.92)
Buckling strength of the chord subject to torsional and torsional-flexural buckling due to axial force N (post-buckling behavior)	Sec. 6.3.1 (6.46) – (6.49), (6.52), (6.53), 6.4.1 (6.69)	Sec. 6.2.3	Sec. 7.7.8 (7.87), (7.89) – (7.92)
Elasto-plastic buckling strength of the chord member subject to lateral-torsional buckling due to bending moment My	Sec. 6.3.2 (6.54) – (6.57)	Sec. 6.2.4	not calculated
Buckling strength of the chord subject to lateral-torsional buckling due to bending moment My	Sec. 6.3.2 (6.54) – (6.57), 6.4.1 (6.69)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the chord subject to lateral-torsional buckling due to bending moment My (post-buckling behavior)	Sec. 6.3.2 (6.54) – (6.57), 6.4.1 (6.69)	Sec. 6.2.4	Sec. 7.7.9 (7.93) – (7.98)
Buckling strength of the chord subjected to axial force N and bending moment My	Sec. 6.3.3 (6.61) – (6.62), 6.4.1 (6.69)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the chord subjected to axial force N and bending moment My (post-buckling behavior)	Sec. 6.3.3 (6.61) – (6.62), 6.4.1 (6.69)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the chord subjected to axial force N and bending moment My taken into account lateral-torsional buckling	Sec. 6.3.3 (6.61) – (6.62), 6.4.1 (6.69)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Buckling strength of the chord subjected to axial force N and bending moment My taken into account lateral-torsional buckling (post-buckling behavior)	Sec. 6.3.3 (6.61) – (6.62), 6.4.1 (6.69)	Sec. 6.2.5(1)	Sec. 7.7.10 (7.99) – (7.100)
Strength of lattice	Sec. 6.2.3 (6.5), 6.2.4 (6.9), 6.4.1 (6.70), 6.4.2.1 (6.72)	–	Sec. 7.7.11.7 (7.106), 7.7.11.8
Stability of lattic	Sec. 6.3.1 (6.46) – (6.53), 6.4.1 (6.70), 6.4.2.1 (6.71), (6.72)	–	Sec. 7.7.11.7 (7.106), 7.7.11.8
Load-carrying capacity of a batten	Sec. 6.2.5 (6.12), 6.2.6 (6.17), 6.2.8 (6.29), 6.4.1 (6.70),	–	Sec. 7.7.11.7 (7.106), 7.7.11.8

Check	EN 1993-1-1	EN 1993-1-3	SP 260.1325800.2016
	6.4.3.1 (6.73), (6.74)		
Buckling strength of the uniform built-up member subject to flexural buckling relating to axes y-y due to axial force N	–	–	Sec. 7.7.11.7 (7.87), (7.88), (7.102) – (7.105)
Buckling strength of the uniform built-up member subject to flexural buckling relating to axes y-y due to axial force N (post-buckling behavior)	–	–	Sec. 7.7.8 (7.87), (7.88), (7.102) – (7.105)
Buckling strength of the uniform built-up member subject to flexural buckling relating to axes z-z due to axial force N	–	–	Sec. 7.7.8 (7.87), Sec. (7.88), (7.102) – (7.105)
Buckling strength of the uniform built-up member subject to flexural buckling relating to axes z-z due to axial force N (post-buckling behavior)	–	–	Sec. 7.7.8 (7.87), (7.88), (7.102) – (7.105)
Slenderness of the chord	–	–	Sec. 7.7.11.5, 7.7.11.6

The following checks are not performed:

- of the weakened sections with holes for bolts;
- of the bolted connection between the battens and the cold-formed lattice chords;
- of the local stress in the cross-section web/webs from the concentrated force or the support reaction.

Peculiarities of the implementation

1. When determining the relative eccentricity of members under compression and bending SP 260.1325800 recommends taking the design moment as the moment in the section, which is located in a particular area of the bar. This area is determined depending on the boundary conditions of the bar, on which the program has no information. Therefore, the value of the bending moment maximal along the length of the element is conservatively used.

2. The codes do not consider the problem of stability for a bar under tension and bending, but it would be unwise not to check it for stability at all, because even a relatively small tension can cause the buckling of the bar (in an elastic bar this would occur when some fibers were in compression, while in an elastoplastic bar this boundary would be harder to locate). Since design codes do not define a boundary for such a “relatively small tension”, the check of stability of a bar under tension and bending is conservatively reduced to the check of the stability of in-plane bending.

20.7. Specifying Unification Groups

Unification groups for selecting sections can be created in the process of specifying structural members and their groups, or after creating them. Generation of the unification groups is performed in the **Unification Groups for the Selection of Sections** dialog box (Fig. 20.7-1). You can create a new unification group in this dialog box, and also include structural members and groups of structural members into a group created earlier.

The same cross-section will be selected for all elements included in a unification group in the process of the selection of sections.

It is necessary to perform the following actions to create a new unification group:

- specify the name of the group in the respective field;
- click the **New group** button and the name of the new group will appear in **Unification groups** list;
- select the names of structural members or groups included in a created unification group in the **Structural members and groups of structural members** list, and click the button ▶;
- repeat the above actions to create a new group;

- after finishing the work, close the dialog box by clicking the **OK** button.

If all objects of the left list are included in one unification group, click the button ▶▶. In order to exclude elements from the unification group select their names in the right list and click the button ◀ or exclude all objects from the group using the button ◀◀.

In order to correct a unification group created earlier, it is necessary to select its name in the list of groups of the **Unification groups** dialog box, and then add new structural groups to it or/and delete the existing ones using the buttons ▶ and ◀.

The following rules should be observed when creating unification groups:

- only the objects (structural members and/or their groups), all bars of which have the same type of rigidity can be included in a unification group;
- an object can be included only into one unification group.

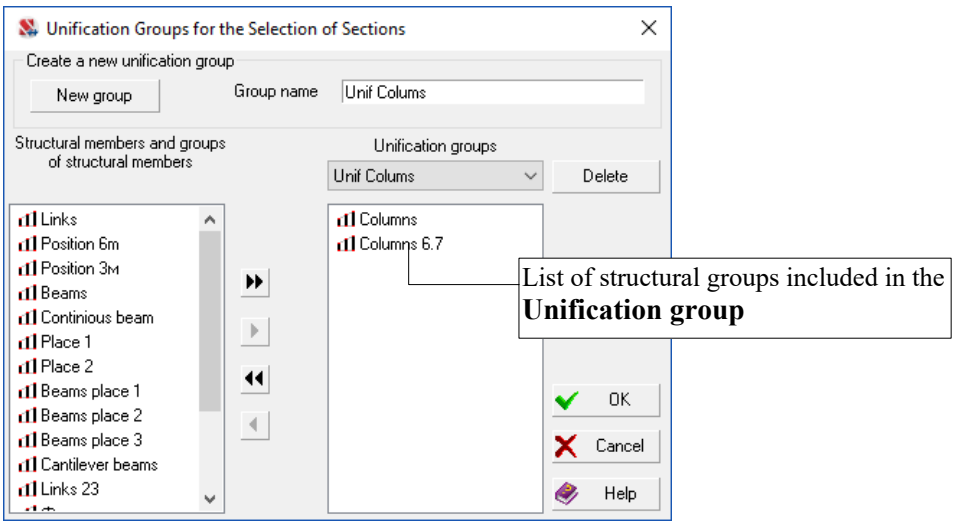


Figure 20.7-1. *The* **Unification Groups for the Selection of Sections** *dialog box*

20.8. Selection of Elements of Groups

If a group of structural members or a structural member is selected in the list of structural groups in the **Input data** toolbar, clicking this button will select all finite elements of the group. Red color is assumed for the selected nodes and elements by default. You can change it in the **Color Palette Settings** section of the ***Settings*** menu (the **Model** tab, the **Select nodes and elements** item).

20.9. Active Group Visualization Mode

In those cases when you do not need to perform any operations with the selected structural group, and you only want to highlight it in the model, an operation of visualization of a structural group is used. A group selected in the list of structural groups in the **Input data** toolbar will be "highlighted" in purple in the model.

20.10. Control of the Groups of Steel Structural Members and Groups of Unifications

The operation is used to control the attributes of structural groups and enables to perform the following actions:

- rename a group;
- sort groups by a specified criterion;

- delete a group;
- change the place of the selected group in the list of groups.

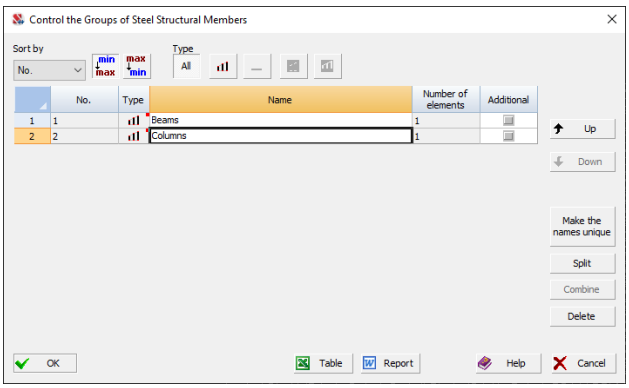


Figure 20.10-1. *The* **Control the Structural Groups of Steel Elements** *dialog box*

This operation invokes the **Control the Structural Groups of Steel Elements** dialog box (Fig. 20.10-1), with a list of structural groups. Columns of the table of structural groups include:

- current number of the structural group;
- **No.** — number of the group in the order of creation;
- **Type** — icon of the type of the structural group (— structural member, — group of structural members). If a group is additional, the respective icon has a green background;
- **Name** — name of the structural group;
- **Number of elements** in the group.

In order to rename a group place the cursor over the name of the respective group and double left click. The line will become available for editing.

The characteristic of the group with respect to which the sorting is performed is selected from the **Sort by** list. The following types of sorting are provided:

- by the number (in the order of creation of groups — when the **Min/Max** button is pressed; and vice verse — when the **Max/Min** button is pressed);
- by the type of group (when the **Min/Max** button is pressed structural members will be first in the list and they will be followed by the structural members, and vice verse — when the **Max/Min** button is pressed);
- by the name of the group (in alphabetical order — when the **Min/Max** button is pressed, and vice verse — when the **Max/Min** button is pressed);
- by the number of elements in the group (in the ascending order of the number of elements in the list of structural groups — when the **Min/Max** button is pressed, and vice verse — when the **Max/Min** button is pressed).

To change the position of one or several structural groups in the list it is necessary to select these groups and move them in the list by clicking the **Up** and **Down** buttons.

In order to delete structural groups select them in the table, place the cursor over the current number of the group and left click. Then click the **Delete** button. If the groups you want to delete go in the successive order in the list, place the cursor over the first group in the list to select them, click and hold the left mouse button, and drag the pointer across the numbers of the groups you want to delete.

If when specifying the data the user has created several groups with the same names, clicking the **Make the names unique** button enables to rename them automatically (by adding the suffix № XXX to the name) and obtain unique names.

In order to combine groups of structural members it is necessary to select these groups in the list while pressing the **Ctrl** key in the keyboard and then click the **Combine** button. The operation will be available only if all parameters of the combined groups of structural members are matching. The dialog box will appear after invoking the operation, where you can specify the name of the combined group.

The **Split** button enables to automate the process of correcting errors in the data on structural members and automatically create structural members and groups of structural members. Clicking this button will perform the following operations for each selected group.

- All the elements other than bars are removed from the list of elements.
- All elements that cannot be interpreted as steel are removed from the list of elements.
- Structural members are created for the remaining elements by combining bars with the same rigidity that lie on one line and are successively adjacent to each other. In addition, the absence of hinges in intermediate nodes is controlled. Allowed the presence of rigid inserts, provided that these inserts are the same for all elements.
- All other parameters (material, effective lengths, ...) are inherited from the properties of the original group.
- Groups of structural members are created for all the remaining finite elements, each group containing elements with the same type of rigidity.

Buttons in the upper part of the dialog box enable to filter groups which should be displayed (for example, to display only the additional steel structural members).

All specifications made in the dialog box will be saved and performed after closing the dialog box by clicking the **OK** button. The position of the structural groups in the list of the toolbar will correspond to their position in the list of the dialog box.

Similarly, the operation allows you to change the names, delete and change the order of unification groups.

20.11. Changing Steels in the Groups of Structural Members

This operation enables to change steels (for example, when changing the design codes) in several structural members or groups of structural members. It is performed in the **Specify Steel** dialog box (Fig. 20.11-1), which appears after invoking it.

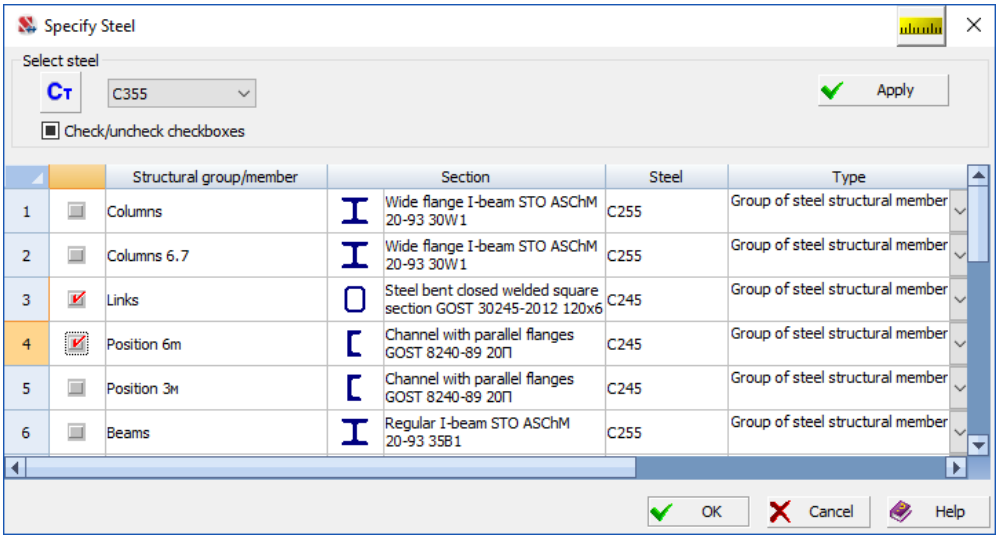


Figure 20.11-1. *The* **Specify Steel** *dialog box*

It is necessary to perform the following actions to change the steel:

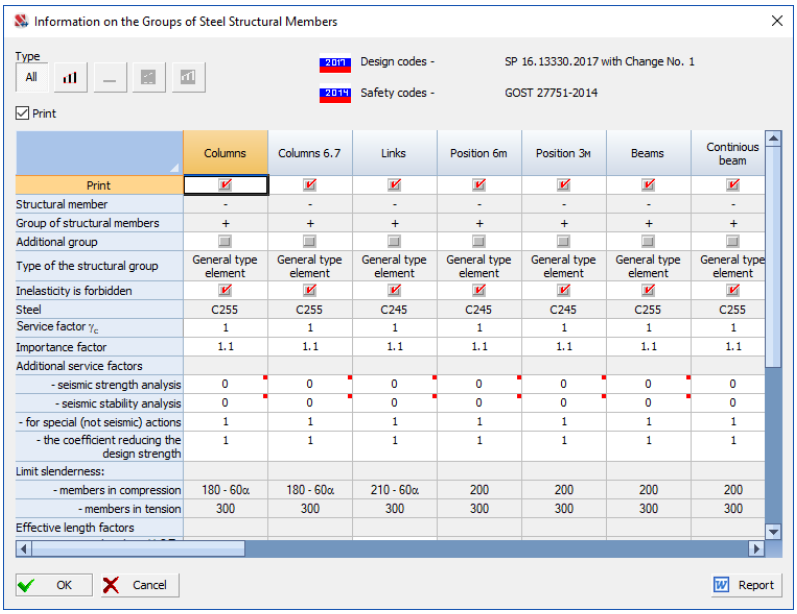
- select the necessary steel from the list or use the operation of selection — to specify steel in accordance with the used design codes;
- indicate the groups in which the selected steel will be used with the help of the checkboxes in the list of structural groups;

- click the **Apply** button, and the name of the selected steel will appear in the **Steel** column of the selected groups;
- repeat the above operations to assign steels for other structural groups.

Moreover, the type of the group is specified in the **Type** column and you can replace *Steel structural member* by *Group of steel structural members* and vice verse.

If the specification is performed for all or most structural groups, you can use the **Check/uncheck checkboxes** checkbox to check all checkboxes. All the specifications made in the dialog box will be saved and performed after closing the dialog box by clicking the **OK** button.

20.12. Information on the Groups of Steel Structural Members



	Columns	Columns 6.7	Links	Position 6m	Position 3м	Beams	Continious beam
Print	☑	☑	☑	☑	☑	☑	☑
Structural member	-	-	-	-	-	-	-
Group of structural members	+	+	+	+	+	+	+
Additional group	☐	☐	☐	☐	☐	☐	☐
Type of the structural group	General type element	General type element	General type element	General type element	General type element	General type element	General type element
Inelasticity is forbidden	☑	☑	☑	☑	☑	☑	☑
Steel	C255	C255	C245	C245	C245	C255	C255
Service factor γ_c	1	1	1	1	1	1	1
Importance factor	1.1	1.1	1.1	1.1	1.1	1.1	1.1
Additional service factors							
- seismic strength analysis	0	0	0	0	0	0	0
- seismic stability analysis	0	0	0	0	0	0	0
- for special (not seismic) actions	1	1	1	1	1	1	1
- the coefficient reducing the design strength	1	1	1	1	1	1	1
Limit slenderness:							
- members in compression	180 - 60α	180 - 60α	210 - 60α	200	200	200	200
- members in tension	300	300	300	300	300	300	300
Effective length factors							

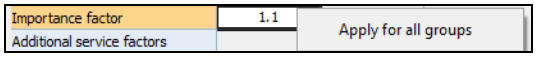
Figure 20.12-1. *The* **Information on the Groups of Steel Structural Members** *dialog box*

This operation enables to obtain the dialog box (Fig. 20.12-1) with the information in all groups of structural members. The **Report** button invokes the generation of the report.

Buttons in the upper part of the dialog box enable to filter groups which should be displayed (for example, to display only the additional steel structural members).

Checkboxes in the **Print** line enable to select only the groups information on which must be given in the report document.

Moreover, this dialog box also enables to change some data (effective length factors, service factors, …). In the case of editable data cells, right-clicking in this dialog box will invoke a context menu, where you can select the **Apply for all groups** option, and the data in the active cell will be copied to all groups:



20.13. Visualization of the Modified Sections

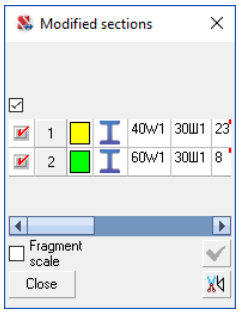


Figure 20.13-1. *The* **Modified Sections** *dialog box*

This operation enables to highlight structural groups in the design model the sections of which were replaced when these groups were created or corrected (i.e. sections of finite elements included in such structural members or groups are different from those specified in the stiffness properties). In this case a color scale is output similar to the scale of the color display of rigidities (Fig. 20.13-1).

20.14. Restore Initial Sections

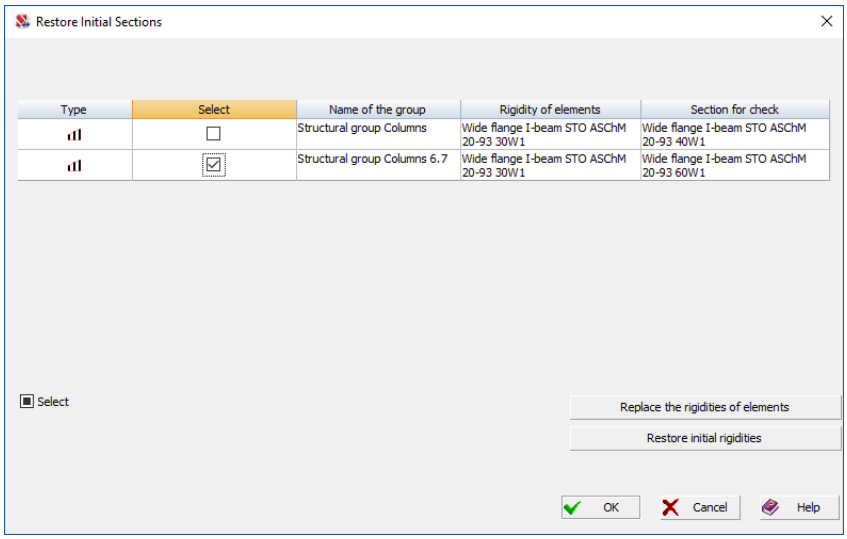


Figure 20.14-1. *The* **Restore Initial Sections** *dialog box*

This operation enables to restore the initial sections (specified in the stiffness properties) in the elements of structural groups the sections of which were replaced when these groups were created or corrected.

This operation invokes the **Restore Initial Sections** dialog box (Fig. 20.14 -1), where you have to indicate structural groups you want to change, select the action (restore initial rigidities or replace the rigidities of elements) and click the **OK** button. The **Select** checkbox enables to select or deselect all groups. If the **Replace the rigidities of elements** operation is selected, they will be actually replaced after closing the postprocessor.

20.15. Changing the Design Codes

This operation enables to change the design codes for steel structures and safety codes. You should select the necessary codes from the drop-down lists in the dialog box (Fig. 20.15-1) and click the **OK**

button. When changing the codes you should remember that the codes according to which the design combinations of forces were selected and the design codes should correspond to each other.

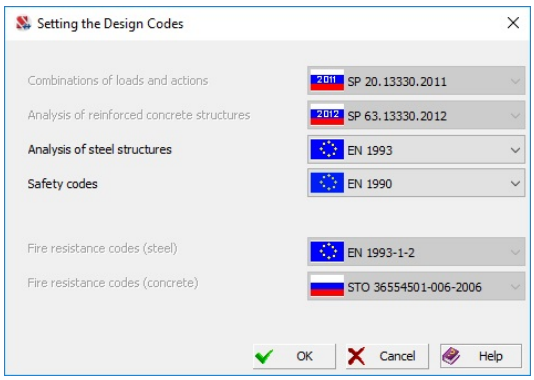


Figure 20.15-1. *The* **Setting the Design Codes** *dialog box*

20.16. Check and Selection of Sections

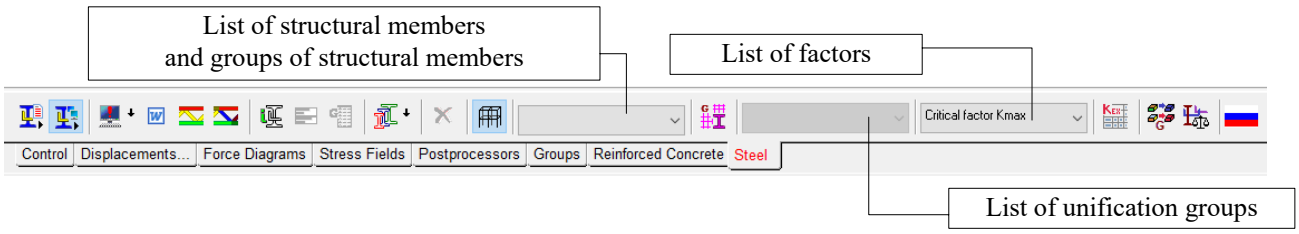


Figure 20.16-1. *The toolbar for calculating the load-bearing capacity of steel sections and the analysis of results*

The toolbar for the check and analysis of results (Fig. 20.16-1) includes buttons for invoking the following operations:

— calculation (check of the sections);

— generate report;

— display results of the check on the model (3-color scale);

— display results of the check on the model (full color scale);

— display information on sections;

— selective visualization of results;

— composition of the group of structural members;

— selection of sections;

— cancel the selected group;

— display the design model;

— show active group;

— create elements of the technical specification for steel;

— information on steel consumption;

— select the design codes.

The check of the load-bearing capacity of structural members and/or groups of structural members is performed by clicking the **Calculation** button — . This button has an arrow which besides the full selection or check for all structural groups enables to perform the calculation only for those groups that have been modified, to perform the calculation for user' selected groups only (in this case dialog box will appear and user must select groups for calculation; when a group included in the unification group is selected during the section selection, all groups included in the corresponding unification group will be automatically selected), to invoke the operations of the data control (without performing the calculation), view messages that were output during the previous calculation or control, and delete results. Regardless of whether changes have been made in all structural groups or only in some of them, the control of the initial data is performed for all groups and diagnostic messages will be generated for all groups, and not just for those which were recalculated.

The check is performed for all sections of finite elements included in a structural member or a group of structural members. The maximum value of each factor defining the load-bearing capacity of the element is obtained in the result of the check.

The software implements the whole set of checks for strength and stability in compliance with the recommendations of different codes. Since the limitations have the same meaning but refer to different sections of codes of different years, as an example, they will be given as references to sections of SNiP II-23-81*:

- bars in tension are not checked for strength by the formula (6) as members which can be used even after the yield point is reached;
- the permission to perform the check of members allowing for the plasticity by the formula (49) provided that the local stability is ensured given in the last paragraph of Sec. 5.25 is not used ;

when calculating the φ_b coefficient, it is assumed that the load is uniformly distributed and applied to the compression chord which is not restrained in its span against buckling.

The approach to the determination of the coefficient $\boldsymbol{\varphi_b}$ is based on the following considerations:

- SNiP does not provide all possible combinations of restraints of the compression chord and the distribution of load over the span, therefore any solution will not be complete;
- there are almost no real structures which have only one loading (loadings can include distributed loads as well). It should be taken into account that the performed calculation is conditional since the design combinations of forces are used and the envelope of the moment diagram can not be clearly identified as that resulting from a certain load applied to a certain chord.

The approach implements the concept of SNiP, where it is, for example, allowed to find the unsupported lengths for one ("the most compressed") loading and to use these lengths in checks for other loadings determining in particular the buckling coefficients $\boldsymbol{\varphi}$ based on these lengths (see Sec.6.11* of SNiP II-23-81*). Coefficient $\boldsymbol{\varphi_b}$ is used for the same purposes as the buckling coefficient $\boldsymbol{\varphi}$.

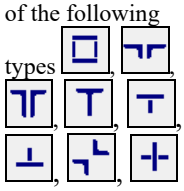
However, a special dialog box can be invoked by clicking the button , where you can select one of the rules for calculating this factor provided by the codes (the user is responsible for the consequences of this selection), or use the **Default** button to return to the standard behavior of the program (described above).

The set of checks according to SNiP II-23-81* depends on the type of the member cross-section and the set of loads it is subjected to. The following tables provide lists of checks for different types of structural members.

Checks of columns and general type elements

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Strength under axial tension/compression	Sec. 5.1	Sec. 5.1	Sec. 8.1.1	Sec. 7.1.1	Sec. 1.4.1.3	Sec. 8.1.3

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Excessive deformations of the tension fiber	Sec. 5.28	Sec. 5.28	Sec. 10.1.3	Sec. 9.1.3	Sec. 1.6.1.3	Sec. 10.1.3
Strength under action of bending moment M_y	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1
Strength under action of bending moment M_z	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1
Strength under action of lateral force Q_y	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 10.1.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1
Strength under action of lateral force Q_z	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 10.1.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1
Strength under combined action of longitudinal force and bending moments	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1.1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1
Strength under combined action of longitudinal force and bending moments, allowing for plasticity	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1.1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1
Strength under combined action of longitudinal force and bending moments, no plasticity	Sec. 5.24, 5.25	Sec. 5.24, 5.25	Sec. 10.1.1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1
Strength for reduced stresses at the simultaneous action of the bending moment and the lateral force	Sec. 5.14*	Sec. 5.14	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1
Stability under compression in XoY (XoU) plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3
Stability under compression in XoY (XoU) plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5
Stability under compression in XoZ (XoV) plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3
Stability under compression in XoZ (XoV) plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5
Stability in compression of angle	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3
Stability in the moment M_y plane under eccentric compression	Sec. 5.27*	Sec. 5.27	Sec. 10.2.9, 10.2.10	Sec. 9.2.9, 9.2.10	Sec. 1.6.2.9, 1.6.2.10	Sec. 10.2.9, 10.2.10

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Stability in the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.27, 7.20*	Sec. 5.27, 7.30	Sec. 10.2.9, 10.2.10, 10.4.6	Sec. 9.2.2, 9.2.10, 9.4.6	Sec. 1.6.2.2, 1.6.2.10, 1.6.4.5	Sec. 10.2.2, 10.2.10, 10.4.5
Stability in the moment M_z plane under eccentric compression	Sec. 5.27*	Sec. 5.27	Sec. 10.2.9, 10.2.10, 10.3.1, 10.3.2	Sec. 9.2.9, 9.2.10, 9.3.1, 9.3.2	Sec. 1.6.2.9, 1.6.2.10, 1.6.3.1, 1.6.3.2	Sec. 10.2.9, 10.2.10, 10.3.1, 10.3.2
Stability in the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27, 7.20*	Sec. 5.27, 7.30	Sec. 10.2.9, 10.2.10, 10.3.1, 10.3.2, 10.4.6	Sec. 9.2.8, 9.2.10, 9.3.1, 9.3.2, 9.4.6	Sec. 1.6.2.8, 1.6.2.10, 1.6.3.1, 1.6.3.2, 1.6.4.5	Sec. 10.2.8, 10.2.10, 10.3.1, 10.3.2, 10.4.5
Stability under compression and bending in two planes	Sec. 5.34	Sec. 5.35	Sec. 10.2.9	Sec. 9.2.9	Sec. 1.6.2.9	Sec. 10.2.9
Stability under compression and bending in two planes (post-buckling behavior)	Sec. 5.34, 7.20*	Sec. 5.35, 7.30	Sec. 10.2.9, 10.4.6	Sec. 9.2.9, 9.2.10, 9.4.6	Sec. 1.6.2.9, 1.6.2.10, 1.6.4.5	Sec. 10.2.9, 10.2.10, 10.4.5
Stability out of the moment M_y plane under eccentric compression	Sec. 5.30-5.32	Sec. 5.30-5.32	Sec. 10.2.4, 10.2.5, 10.2.8	Sec. 9.2.4, 9.2.5, 9.2.8	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8	Sec. 10.2.4, 10.2.5, 10.2.8
Stability out of the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.30-5.32, 7.20*	Sec. 5.30-5.32, 7.30	Sec. 10.2.4, 10.2.5, 10.2.8, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.4.5
Stability out of the moment M_z plane under eccentric compression (sections of the following types are not checked)	Sec. 5.27*,5.30-5.32	Sec. 5.27, 5.30-5.32	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2	Sec. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.2	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.1, 1.6.3.2	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2
Stability out of the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.30-5.32, 7.20*	Sec. 5.27, 5.30-5.32, 7.30	Sec. 10.2.4,10.2.5,10.2.8, 10.3.1, 10.3.2, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.3.1, 9.3.2, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.1, 1.6.3.2, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.1, 10.3.2, 10.4.5
Stability of in-plane bending (sections of the following types  are not checked)	Sec. 5.15	Sec. 5.15	Sec. 9.4.1	Sec. 8.4.1	Sec. 1.5.4.1	Sec. 9.4.1

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Lateral-torsional buckling taking into account plastic deformation			Sec. 9.4.6	Sec. 8.4.6	Sec. 1.5.4.6	Sec. 9.4.6
Ultimate web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9, 7.10; Sec. 7.14, 7.16*, 7.17*, 7.18*, Table 27*	Sec. 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10; 7.23, 7.26, 7.27, 7.28,	Sec. 8.3.2, Table 8; Sec. 8.3.10 ; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 20 ; Sec. 10.4.3; Sec. 10.4.9	Sec. 7.3.2, Table 9; Sec. 7.3.11 ; Sec. 8.5.1–8.5.9; Sec. 9.4.2, Table 22 ; Sec. 9.4.3; Sec. 9.4.9	Sec. 1.4.3.2, Table 1.4.3; Sec. 1.5.5.1–1.5.5.9; Sec. 1.6.4.2, Table 1.6.3 ; Sec. 1.6.4.5	Sec. 8.3.2, Table 8.3; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 10.3 ; Sec. 10.4.5
Ultimate flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.24, Table 30; Sec. 7.27*	Sec. 7.32, 7.33, 7.34, 7.37	Sec. 8.3.7, Table 9; Sec. 8.3.10 ; Sec. 9.5.14; Sec. 10.4.7, Table 21 ; Sec. 10.4.9	Sec. 7.3.8, Table 10; Sec. 7.3.11 ; Sec. 8.5.18; Sec. 9.4.7, Table 23 ; Sec. 9.4.9	Sec. 1.4.3.7, Table 1.4.4; Sec. 1.5.5.14; Sec. 1.6.4.8, Table 1.6.4 ; Sec. 1.6.4.7	Sec. 8.3.7, Table 8.4; Sec. 9.5.14; Sec. 10.4.8, Table 10.4 ; Sec. 10.4.7
Ultimate pipe radius to thickness ratio based on local stability constraint	Sec. 8.6	Sec. 8.6	Sec. 12.2.2	Sec. 11.2.2	Sec. 1.10.2.2	Sec. 14.2.2
Local stability of the pipe wall based on closed circular cylindric shell calculation	Sec. 8.5-8.13	Sec. 8.5-8.13	Sec. 12.2.1-12.2.8	Sec. 11.2.1-11.2.9	Sec. 1.10.2.1-1.10.2.9	Sec. 14.2.1-14.2.9
Critical beam web height to thickness ratio	Sec. 7.4*	Sec. 7.4	Sec. 9.5.3	Sec. 8.5.3	Sec. 1.5.5.3	Sec. 9.5.3
General stability of a built-up member under axial compression in XoY plane	Sec. 5.3-5.6	Sec. 5.3-5.6	Sec. 8.1.3-8.1.5, 8.2.2	Sec. 7.1.3-7.1.5, 7.2.2	Sec. 1.4.1.3, 1.4.1.5, 1.4.2.2, 1.4.2.5	Sec. 8.1.3, 8.1.5, 8.2.2, 8.2.5
General stability of a built-up member under axial compression in XoZ plane	Sec. 5.3-5.6	Sec. 5.3-5.6	Sec. 8.1.3-8.1.5, 8.2.2	Sec. 7.1.3-7.1.5, 7.2.2	Sec. 1.4.1.3, 1.4.1.5, 1.4.2.2, 1.4.2.5	Sec. 8.1.3, 8.1.5, 8.2.2, 8.2.5
Stability out of the moment M_z plane	Sec. 5.30-5.32	Sec. 5.27, 5.30-5.32	Sec. 10.2.4,10.2.5,10.2.8	Sec. 9.2.4, 9.2.5, 9.2.8	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8	Sec. 10.2.4, 10.2.5, 10.2.8
Resistance of a batten to bending	Sec. 5.8, 5.9, 5.36	Sec. 5.8, 5.9, 5.38	Sec. 8.2.7, 8.2.8, 10.3.7	Sec. 7.2.7, 7.2.8, 9.3.7	Sec. 1.4.2.7, 1.4.2.8, 1.6.3.7	Sec. 8.2.7, 8.2.8, 10.3.7
Strength of chord under bending moment M_y	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1
Strength of chord under bending moment M_z	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1
Strength of chord under lateral force Q_y	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 9.2.3	Sec. 8.2.1, 8.2.3	Sec. 1.5.2.1, 1.5.2.3	Sec. 9.2.1, 9.2.3
Strength of chord under lateral force Q_z	Sec. 5.12, 5.18*	Sec. 5.12, 5.18	Sec. 9.2.1, 9.2.3	Sec. 8.2.1, 8.2.3	Sec. 1.5.2.1, 1.5.2.3	Sec. 9.2.1, 9.2.3

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Strength of chord under combined action of longitudinal force and bending moments	Sec. 5.24, 5.25, 5.33	Sec. 5.24, 5.25, 5.33	Sec. 10.1.1, 10.3.3	Sec. 9.1.1, 9.3.3	Sec. 1.6.1.1, 1.6.3.3	Sec. 10.1.1, 10.3.3
Strength of chord under combined action of longitudinal force and bending moments, allowing for plasticity	Sec. 5.24, 5.25, 5.33	Sec. 5.24, 5.25, 5.33	Sec. 10.1.1, 10.3.3	Sec. 9.1.1, 9.3.3	Sec. 1.6.1.1, 1.6.3.3	Sec. 10.1.1, 10.3.3
Strength of chord under combined action of longitudinal force and bending moments, no plasticity	Sec. 5.24, 5.25, 5.33	Sec. 5.24, 5.25, 5.33	Sec. 10.1.1, 10.3.3	Sec. 9.1.1, 9.3.3	Sec. 1.6.1.1, 1.6.3.3	Sec. 10.1.1, 10.3.3
Stability of chord under compression in XoY plane	Sec. 5.3, 5.6	Sec. 5.3, 5.6	Sec. 8.1.3, 8.2.3-8.2.5	Sec. 7.1.3, 7.2.3-7.2.5	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4	Sec. 8.1.3, 8.2.3, 8.2.4
Stability of chord under compression in XoY plane (post-buckling behavior)	Sec. 5.3, 5.6, 7.20*	Sec. 5.3, 5.6, 7.30	Sec. 8.1.3, 8.2.3-8.2.5, 8.3.5	Sec. 7.1.3, 7.2.3-7.2.5, 7.3.6	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4, 1.4.3.5	Sec. 8.1.3, 8.2.3, 8.2.4, 8.3.5
Stability of chord under compression in XoZ plane	Sec. 5.3, 5.6	Sec. 5.3, 5.6	Sec. 8.1.3, 8.2.3-8.2.5	Sec. 7.1.3, 7.2.3-7.2.5	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4	Sec. 8.1.3, 8.2.3, 8.2.4
Stability of chord under compression in XoZ plane (post-buckling behavior)	Sec. 5.3, 5.6, 7.20*	Sec. 5.3, 5.6, 7.30	Sec. 8.1.3, 8.2.3-8.2.5, 8.3.5	Sec. 7.1.3, 7.2.3-7.2.5, 7.3.6	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4, 1.4.3.5	Sec. 8.1.3, 8.2.3, 8.2.4, 8.3.5
Stability of chord in the moment M_y plane under eccentric compression	Sec. 5.27*, 5.33, 5.35	Sec. 5.27, 5.33, 5.37	Sec. 10.2.9, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.9, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.9, 1.6.3.3-1.6.3.5	Sec. 10.2.9, 10.3.3-10.3.5
Stability of chord in the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.33, 5.35, 7.20*	Sec. 5.27, 5.33, 5.37, 7.30	Sec. 10.2.9, 10.2.10, 10.3.3, 10.3.4, 10.3.6, 10.4.6	Sec. 9.2.2, 9.2.10, 9.3.3, 9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.2, 1.6.2.10, 1.6.3.3-1.6.3.5, 1.6.4.5	Sec. 10.2.2, 10.2.10, 10.3.3-10.3.5, 10.4.5
Stability of chord in the moment M_z plane under eccentric compression	Sec. 5.27*, 5.33, 5.35	Sec. 5.27, 5.33, 5.37	Sec. 10.2.9, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.9, 9.3.1, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.9, 1.6.3.3-1.6.3.5	Sec. 10.2.9, 10.3.3-10.3.5
Stability of chord in the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.33, 5.35, 7.20*	Sec. 5.27, 5.33, 5.37, 7.30	Sec. 10.2.9, 10.2.10, 10.3.2-10.3.4, 10.3.6, 10.4.6	Sec. 9.2.8, 9.2.10, 9.3.1-9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.8, 1.6.2.10, 1.6.3.2-1.6.3.5, 1.6.4.5	Sec. 10.2.8, 10.2.10, 10.3.2-10.3.5, 10.4.5
Bending of chord in two principal planes	Sec. 5.24,5.25	Sec. 5.24, 5.25	Sec. 10.1.1	Sec. 9.1.1	Sec. 1.6.1.1	Sec. 10.1.1

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Stability of chord out of the moment M_y plane under eccentric compression	Sec. 5.27*, 5.30-5.33, 5.35	Sec. 5.27, 5.30-5.33, 5.37	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5
Stability of chord out of the moment M_y plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.30-5.33, 5.35, 7.20*	Sec. 5.27, 5.30-5.33, 5.37, 7.30	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.3.1, 9.3.3, 9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5, 10.4.5
Stability of chord out of the moment M_z plane under eccentric compression	Sec. 5.27*, 5.30-5.33, 5.35	Sec. 5.27, 5.30-5.33, 5.37	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.3.1, 9.3.3, 9.3.4, 9.3.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5
Stability of chord out of the moment M_z plane under eccentric compression (post-buckling behavior)	Sec. 5.27*, 5.30-5.33, 5.35, 7.20*	Sec. 5.27, 5.30-5.33, 5.37, 7.30	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3, 10.3.4, 10.3.6, 10.4.6	Sec. 9.2.4, 9.2.5, 9.2.8, 9.2.10, 9.3.1, 9.3.3, 9.3.4, 9.3.6, 9.4.6	Sec. 1.6.2.4, 1.6.2.5, 1.6.2.8, 1.6.3.3-1.6.3.5, 1.6.4.5	Sec. 10.2.4, 10.2.5, 10.2.8, 10.3.3-10.3.5, 10.4.5
Strength of chord under tension	Sec. 5.1, 5.6	Sec. 5.1, 5.6	Sec. 8.1.1, 8.2.1	Sec. 7.1.1, 7.2.1	Sec. 1.4.1.3, 1.4.2.1	Sec. 8.1.3, 8.2.1
Stability of chord under compression	Sec. 5.3, 5.6	Sec. 5.3, 5.6	Sec. 8.1.3, 8.2.3-8.2.5	Sec. 7.1.3, 7.2.3-7.2.5	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4	Sec. 8.1.3, 8.2.3, 8.2.4
Stability of chord under compression (post-buckling behavior)	Sec. 5.3, 5.6, 7.20*	Sec. 5.3, 5.6, 7.30	Sec. 8.1.3, 8.2.3-8.2.5, 8.3.5	Sec. 7.1.3, 7.2.3-7.2.5, 7.3.6	Sec. 1.4.1.3, 1.4.2.3, 1.4.2.4, 1.4.3.5	Sec. 8.1.3, 8.2.3, 8.2.4, 8.3.5
Excessive deformations of the tension chord fiber	Sec. 5.28	Sec. 5.28	Sec. 10.1.3	Sec. 9.1.3	Sec. 1.6.1.3	Sec. 10.1.3
Stability of in-plane bending of the chord	Sec. 5.15	Sec. 5.15	Sec. 9.4.1	Sec. 8.4.1	Sec. 1.5.4.1	Sec. 9.4.1
Ultimate chord web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9,7.10; Sec. 7.14, 7.16*, 7.17*, 7.18*, Table 27*	Sec. 7.1. 7.2, 7.3, 7.4–7.6, 7.9, 7.10; Sec. 7.23, 7.26, 7.27, 7.28	Sec. 8.3.2, Table 8; Sec. 8.3.10 ; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 20 ; Sec. 10.4.3; Sec. 10.4.9	Sec. 7.3.2, Table 9; Sec. 7.3.11 ; Sec. 8.5.1–8.5.9; Sec. 9.4.2, Table 22 ; Sec. 9.4.3; Sec. 9.4.9	Sec. 1.4.3.2, Table 1.4.3; Sec. 1.5.5.1–1.5.5.9; Sec. 1.6.4.2, Table 1.6.3 ; Sec. 1.6.4.5	Sec. 8.3.2, Table 8.3; Sec. 9.5.1–9.5.9; Sec. 10.4.2, Table 10.3 ; Sec. 10.4.5
Ultimate chord flange slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.24, Table 30; Sec. 7.27*	Sec. 7.32, 7.33, Sec. 7.34, п. 7.37	Sec. 8.3.7, Table 9; Sec. 8.3.10 ; Sec. 9.5.14; Sec. 10.4.7, Table 21 ; Sec. 10.4.9	Sec. 7.3.8, Table 10; Sec. 7.3.11 ; Sec. 8.5.18; Sec. 9.4.7, Table 23 ; Sec. 9.4.9	Sec. 1.4.3.7, Table 1.4.4; Sec. 1.5.5.14; Sec. 1.6.4.8, Table 1.6.4 ; Sec. 1.6.4.7	Sec. 8.3.7, Table 8.4; Sec. 9.5.14; Sec. 10.4.8, Table 10.4 ; Sec. 10.4.7
Strength of lattice posts	Sec. 5.8*, 5.10	Sec. 5.8, 5.10	Sec. 8.2.7, 8.2.9	Sec. 7.2.7, 7.2.9	Sec. 1.4.2.7, 1.4.2.9	Sec. 8.2.7, 8.2.9
Strength of lattice struts	Sec. 5.8*, 5.10	Sec. 5.8, 5.10	Sec. 8.2.7, 8.2.9	Sec. 7.2.7, 7.2.9	Sec. 1.4.2.7, 1.4.2.9	Sec. 8.2.7, 8.2.9
Stability of lattice posts under compression	Sec. 5.8*, 5.10, 5.3	Sec. 5.8, 5.10, 5.3	Sec. 8.2.7, 8.2.9, 8.1.3	Sec. 7.2.7, 7.2.9, 7.1.3	Sec. 1.4.2.7, 1.4.2.9, 1.4.1.3	Sec. 8.2.7, 8.2.9, 8.1.3
Stability of lattice struts under compression	Sec. 5.8*, 5.10, 5.3	Sec. 5.8, 5.10, 5.3	Sec. 8.2.7, 8.2.9, 8.1.3	Sec. 7.2.7, 7.2.9, 7.1.3	Sec. 1.4.2.7, 1.4.2.9, 1.4.1.3	Sec. 8.2.7, 8.2.9, 8.1.3

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Limit slenderness in XoY plane	Sec. 6.15, 6.16	Sec. 6.14, 6.15	Sec. 11.4.1	Sec. 10.4.1	Sec. 1.9.4.1	Sec. 13.4.1
Limit slenderness in XoZ plane	Sec. 6.15, 6.16	Sec. 6.14, 6.15	Sec. 11.4.1	Sec. 10.4.1	Sec. 1.9.4.1	Sec. 13.4.1

Checks of beams

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Strength under lateral force	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1
Strength under bending moment	Sec. 5.12	Sec. 5.12	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1
Stability of in-plane bending under moment	Sec. 5.15	Sec. 5.15	Sec. 9.4.1	Sec. 8.4.1	Sec. 1.5.4.1	Sec. 9.4.1
Lateral-torsional buckling taking into account plastic deformation			Sec. 9.4.6	Sec. 8.4.6	Sec. 1.5.4.6	Sec. 9.4.6
Ultimate web slenderness based on local stability constraint	Sec. 7.1. 7.2*, 7.3, 7.4*–7.6*, 7.9,7.10	Sec. 7.1, 7.2, 7.3, 7.4–7.6, 7.9, 7.10	Sec. 9.5.1–9.5.9	Sec. 8.5.1–8.5.9	Sec. 1.5.5.1–1.5.5.9	Sec. 9.5.1–9.5.9
Ultimate flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.24, Table 30	Sec. 7.34	Sec. 9.5.14	Sec. 8.5.18	Sec. 1.5.5.14	Sec. 9.5.14
Critical height to thickness ration of the beam web	Sec. 7.4*	Sec. 7.4	Sec. 9.5.3	Sec. 8.5.3	Sec. 1.5.5.3	Sec. 9.5.3
Strength for reduced stresses at the simultaneous action of the bending moment and the lateral force	Sec. 5.14*	Sec. 5.14	Sec. 9.2.1	Sec. 8.2.1	Sec. 1.5.2.1	Sec. 9.2.1

Checks of truss members

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Strength	Sec. 5.1	Sec. 5.1	Sec. 8.1.1	Sec. 7.1.1	Sec. 1.4.1.1	Sec. 8.1.1
Stability in the truss plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3
Stability in the truss plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5
Stability out of the truss plane	Sec. 5.3	Sec. 5.3	Sec. 8.1.3	Sec. 7.1.3	Sec. 1.4.1.3	Sec. 8.1.3
Stability out of the truss plane (post-buckling behavior)	Sec. 5.3, 7.20*	Sec. 5.3, 7.30	Sec. 8.1.3, 8.3.5	Sec. 7.1.3, 7.3.6	Sec. 1.4.1.3, 1.4.3.5	Sec. 8.1.3, 8.3.5
Ultimate web slenderness based on local stability constraint	Sec. 7.1. 7.2*; Sec. 7.14, Table 27*	Sec. 7.1, 7.2, 7.23	Sec. 8.3.1, 8.3.2, Table 8; Sec. 8.3.10	Sec. 7.3.1, 7.3.2, Table 9; Sec. 7.3.11	Sec. 1.4.3.1, 1.4.3.2, Table 1.4.3	Sec. 8.3.1, 8.3.2, Table 8.3

Check	SNiP II-23-81*	SNiP RK 5.04-23-2002	SP 53-102-2004	SP 16.13330	DBN B.2.6-163:2010	DBN B.2.6-198:2014
Ultimate flange overhang (flange plate) slenderness based on local stability constraint	Sec. 7.22*, 7.23*, Table 29*, Sec. 7.27*	Sec. 7.32, 7.33, 7.37	Sec. 8.3.7, Table 9; Sec. 8.3.10	Sec. 7.3.8, Table 10; Sec. 7.3.11	Sec. 1.4.3.7, Table 1.4.4	Sec. 8.3.7, Table 8.4
Ultimate pipe radius to thickness ratio based on local stability constraint	Sec. 8.6	Sec. 8.6	Sec. 12.2.2	Sec. 11.2.2	Sec. 1.10.2.2	Sec. 14.2.2
Local stability of the pipe wall based on closed circular cylindric shell calculation	Sec. 8.5-8.13	Sec.. 8.5-8.13	Sec. 12.2.1-12.2.8	Sec. 11.2.1-11.2.9	Sec. 1.10.2.1-1.10.2.9	Sec. 14.2.1-14.2.9
Stability of the curved member						
Slenderness	Sec. 6.1-6.4,6.16	Sec. 6.1-6.4, 6.15	Sec. 11.1.1-11.1.4, 11.4.1	Sec. 10.1.1-10.1.4, 10.4.1	Sec. 1.9.1.1-1.9.1.4, 1.9.4.1	Sec. 13.1.1-13.1.4, 13.4.1

By default the parameters of the lattice are not specified in the software for the lattice bars. Therefore the lattice members are not checked in the calculation, and it is assumed that the lattice spacing for a hollow section from angles is equal to *min*(B, H), and for sections from double channels and double I-sections it is equal to the value B. When checking steel lattice bars subjected to the action of an axial force with bending in **SCAD** complex (see the respective section of SNiP II-23-81*, Sec. 5.24* — 5.36), it is assumed that the chords of a lattice bar are ***always*** connected by battens (see the "Resistance of Sections" mode of **Kristall**). All other data on battens are not specified, therefore **SCAD** does not perform any strength checks of the lattice (if it is necessary to check the strength of the lattice, you should use **Kristall** and select the respective type of lattice). The solution is ***always*** conservative when checking the chords according to Sec. 5.24* and 5.25* of SNiP II-23-81*, since a bending moment appears in the chords from the action of a conditional lateral force in battens. This moment acts on the chord in the plane parallel to the planes of the chords.

Checking the **Specify lattice parameters** checkbox invokes the **Lattice Parameters** dialog box (Fig. 20.16-2), where you can specify detailed information on the lattice, including the cross-section of the battens or lattice members (diagonals, horizontals), the spacing of the battens or horizontals.

Since the standard requirement to the limitation of the slenderness of a tension member is related to the limitation of its self-weight sag, the slenderness check of tension members is performed only in the vertical plane (for example, according to Note 1 to Table 33 of SP 16.13330). The truss plane is assumed to be the vertical one for the **Truss Members**. Slenderness of the tension truss members out of the truss plane is not checked. Thus, the out-of-plane bracing of the bottom (tension) chord does not affect the result of its slenderness check. If the user has to check the slenderness of the tension members out of the truss plane (for horizontal and inclined trusses), the **General Type** can be used. When the calculation of a tension member is performed for the **General Type** elements, both slenderness values are checked, because the very concept of a vertical plane is absent in this mode.

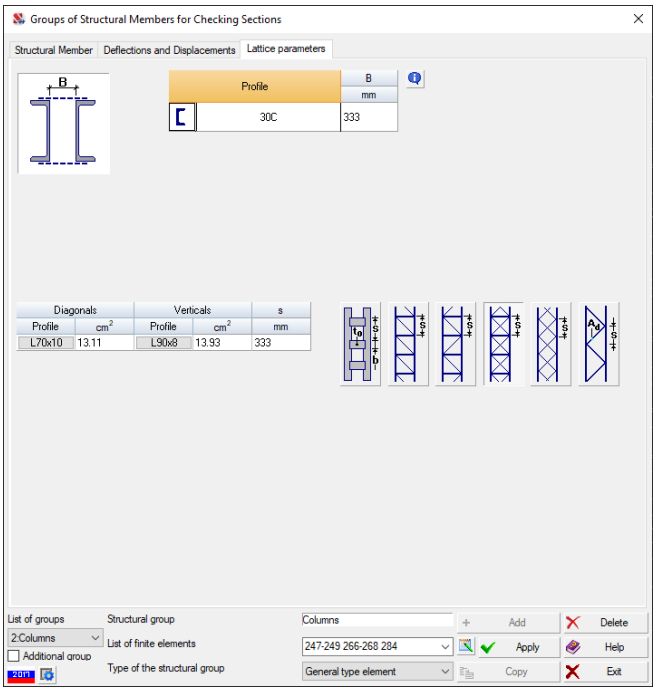


Figure 20.16-2. *Page* **Lattice parameters**

Peculiarities of the implementation

Technically the codes do not require to check the axial compression stability of structures under eccentric compression (for example, under the action of N,M_y,M_z). However, without checking that the axial compression stability is ensured, other checks may become meaningless. For example, the value of the limit slenderness of the 180-60α type can become negative (the value of α coincides with the axial compression stability factor). Therefore, the axial compression stability factors are always calculated.

If EN 1993 is selected for designing steel structures, the following checks are performed

Factor	EN 1993-1-1	EN 1993-1-5
Strength under action of longitudinal tensile force N	6.2.3 (6.5), (6.6)	
Strength under action of longitudinal compressive force N	6.2.4 (6.9)…(6.11)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under action of bending moment M_y	6.2.5 (6.12)…(6.15)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under action of bending moment M_z	6.2.5 (6.12)…(6.15)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under action of lateral force V_z	6.2.6 (6.17)…(6.20)	
Strength under action of lateral force V_y	6.2.6 (6.17)…(6.20)	
Strength under combined action of bending moment M_z and lateral force V_y	6.2.5 (6.12)…(6.15), 6.2.8 (6.29), (6.30)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under combined action of bending moment M_y and lateral force V_z	6.2.5 (6.12)…(6.15), 6.2.8 (6.29), (6.30)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under combined action of bending moment M_y and longitudinal force N	6.2.5 (6.12)…(6.15), 6.2.9 (6.36)…(6.44)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Strength under combined action of bending moments M_y, M_z, longitudinal force N and lateral forces V_z and V_y	6.2.5 (6.12)…(6.15), 6.2.9 (6.36)…(6.44), 6.2.10 (6.45)	4.4 (4.1)...(4.4), Table 4.1, 4.2
Stability at flexural buckling about the y-y axis under action of longitudinal compressive force N	6.3.1 (6.46)…(6.51)	

Factor	EN 1993-1-1	EN 1993-1-5
Stability at flexural buckling about the z-z axis under action of longitudinal compressive force *N*	6.3.1 (6.46)…(6.51)	
Stability at torsional-flexural buckling under action of longitudinal compressive force *N*	6.3.1 (6.46)…(6.49), 6.3.1 (6.52), (6.53)	
Stability of in-plane bending under action of bending moment M_y	6.3.2 (6.54)...(6.56)	
Stability of in-plane bending under action of bending moment M_z	6.3.2 (6.54)...(6.56)	
Stability under combined action of longitudinal force and bending moments	6.3.3 (6.61), (6.62)	
Stability under combined action of longitudinal force and bending moments taking into account in-plane bending	6.3.3 (6.61), (6.62)	
Stability of the web under action of lateral force V_z		5.2 (5.1), (5.2), 5.3 (5.3)...(5.6), 5.4 (5.8), (5.9), 5.5 (5.10)
Stability of the web under action of lateral force V_y		5.2 (5.1), (5.2), 5.3 (5.3)...(5.6), 5.4 (5.8), (5.9), 5.5 (5.10)

On Formulas (49) and (50) from SNiP II-23-81*

Formula (49) is totally acceptable in application to rectangular cross-sections with a characteristic angular point A (Fig. 20.16-3), where conditional stresses $M_x/(c_x W_{xn,\min} R_y \gamma_c)$ and $M_y/(c_y W_{yn,\min} R_y \gamma_c)$ are summed.

It is impossible to sum the stresses in sections without such an angular point (Fig. 20.16-4). Formula (50) of SNiP II-23-81* is better in this sense. It allows to choose between coordinates *x* and *y* of various cross-section points. This problem exists in SP and DBN as well.

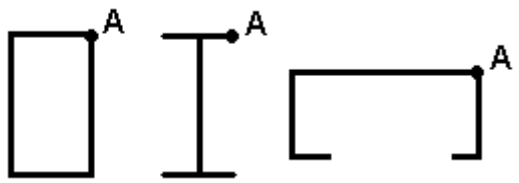


Figure 20.16-3. *Cross-sections with an angular point*

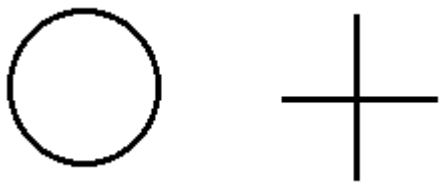


Figure 20.16-4. *Cross-sections without angular points*

The said difference between the considered cases was noticed by L.B. Katznelson (Central Res. Inst. for Build. Structures). Authors of the application are very grateful to him for initiating a discussion of this and many other details of the design code implementation.

20.17. On Seismic and Special Actions

If a seismic action is included in a design combination, **SCAD** automatically takes into account the requirement of the selected code and introduces an additional service factor. Thus, coefficient m_{tr} is introduced in the strength and stability calculation taking into account the seismic load for SP 14.13330.2014 “Construction in Seismic Regions” according to Sec. 5.15. A similar coefficient is taken into account for other codes, for example, DBN B.1.1-12:2014 (Ukraine), KMK 2.01.03-96 (Republic of Uzbekistan) etc.

In the special table you can specify the coefficients allowing for seismic action at the strength and stability analysis. If zero values are specified, the values are taken in accordance with the respective seismic codes by default. Having specified, for example, these two coefficients equal to 0.9, you can take into account the standard requirements for the calculation of steel structures operating in unheated rooms or in the open air at the design temperature below minus 40°C.

If the loads and actions are combined according to SP 20.13330, the special table enables to specify a service factor which will be used in the calculation for combinations of loadings including special (non-seismic) loadings in accordance with Annex B of SP 296.1325800.2017 (moreover, the importance factor γ_n=1,0 in accordance with Sec. 5.5 of SP 296.1325800.2017 will be used for such combinations). An

additional service factor which reduces the design strength and is considered in Sec. 5.11 and Annex C of SP 296.1325800.2017 can be taken into account as well.

20.18. Analysis of Custom Sections

SNiP II-23-81* and other codes regulate the checks only for a limited set of cross-section shapes. However, the practice of structural design often requires using the section shapes not defined by the codes. **SCAD** enables to specify stiffness properties of bars using the data obtained with the help of a set of programs for generating sections (**Section Builder**, **Consul**, **Tonus**). In this case the problem of selecting the design strength of steel $\mathbf{R_y}$, the value of which depends on the thickness of the section, appears when checking the sections.

The value of the design strength $\mathbf{R_y}$ will be taken according to the following rules for arbitrary sections with an explicitly specified steel grade:

- for sections created by **Consul** — the minimum design strength value of the steel of the given grade;
- for sections created by **Tonus** — design strength value of the thickest steel strip (if this thickness is not found in the respective table of the design code, the minimum design strength value of the steel of the given grade will be taken);
- for sections created by **Section Builder** — design strength value of the thickest steel plate or rolled profile (if this thickness is not found in the respective table of the design code, the minimum design strength value of the steel of the given grade will be taken); if the section is made from structural steel, the data from the codes for structural steel will be used, otherwise — the data from the codes for sheet and plate steel.

Based on the calculation considerations the most unfavorable values of the coefficient accounting for the effect of the section shape η, coefficients α and β etc are taken in the calculation. The check of stability of in-plane bending is not performed at all assuming that the possibility of such buckling mode is excluded by using the respective restraints.

SNiP, DBN and other similar design codes regulate the analysis of sections under shear forces according to the formulas that are difficult to adapt to the case of an arbitrary section. Therefore, when analyzing the strength of an arbitrary section under shear forces, the software uses the Eurocode approach, which compares the average tangential stresses, calculated as the ratio of the shear force to the shear area, and the design shear strength.

20.19. Display Results

The results of the check of the load-bearing capacity can be displayed on the model for all structural members, groups of structural members or unification groups, and also for each structural member or group of members separately.

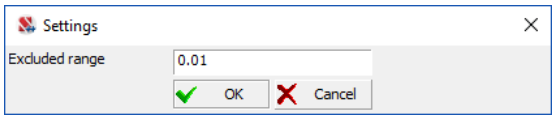


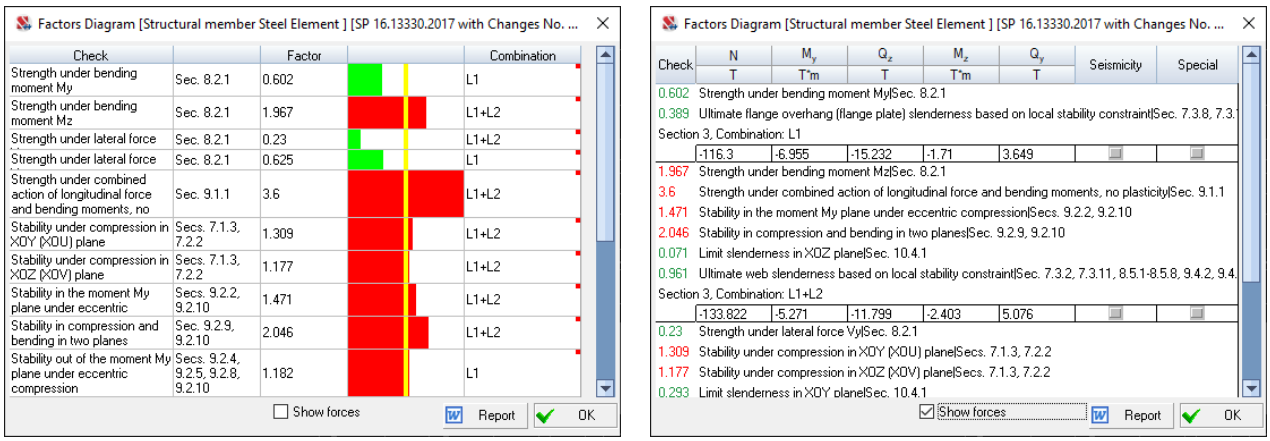
Figure 20.19-1. *The* **Settings** *dialog box*

When the display operation which is invoked by clicking the **Display results of the check on the model (3-color scale)** button — is used, structural members are displayed in the model in three colors — green, if the load-bearing capacity is sufficient, red — otherwise, or yellow, if the load-bearing capacity is close to the critical value (the factor is close to 1,0). The critical range is specified by setting the color scale (button in the window of the scale). For example, if you specify the value 0,03 in the dialog box (Fig. 20.19-1), all elements with the value of the factor less than 0.97 will be displayed in green, if the factor is greater than 1,03 — in red, and if the factor is in the range [0,97;1,03] — yellow will be used.

If a display mode which is invoked by clicking the **Display results of the check on the model (full color scale)** button — is used, an ordinary color scale with a custom number of colors will appear.

If not a single group of structural members is selected in the drop-down list, the visualization of the critical factor K_{max} is performed. If a particular group is selected, the list of factors will contain the full list of performed checks and you will be able to display on the model the results of the check for each factor defining the load-bearing capacity of the considered structures (for example, strength or stability). In the case of a preliminary design, not only can you output the utilization factor of restrictions K_{max}, but also the color scale of the utilization factor of restrictions without taking into account the limit slenderness and local stability factors. The values of the selected factor can be output on the model using the filters button **Digitization** — .

In order to visualize the results for each structural member or unified group you have to select the name of the object in the respective list and click the **Selective visualization of results** button . The **Factors Diagram** dialog box will appear (Fig. 20.19-2*a*), where a utilization factor is output for each factor checked. Factors the load-bearing capacity utilization factors of which exceed one will be marked in red in the diagram. The **Show forces** checkbox enables to get an alternative presentation of information showing forces/stresses in an element (Fig. 20.19-2*b*).



a) *b)*

Figure 20.19-2. *The* **Factors Diagram** *dialog box*

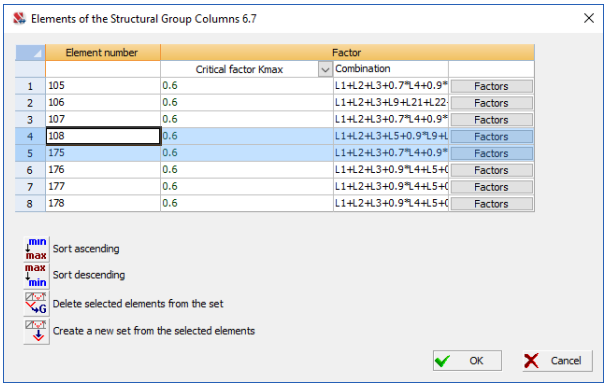


Figure 20.19-3. *The* **Elements of the Structural Group** *dialog box*

The **Elements of the Structural Group** dialog box, shown in Fig. 20.19-3, which is invoked by clicking the **Composition of the group of structural members** button — , is used for the analysis of the load-bearing capacity of each element of the structural group.

In order to invoke the dialog box it is necessary to select the name of the considered group in the list of the toolbar and click the respective button. The dialog box contains a table with the numbers of elements included in the selected group, values of the utilization factors for a factor given in the first line of the table, and also the **Factors** buttons which enable to obtain the load-bearing capacity utilization factors for an individual element.

Buttons located under the table enable to perform the following actions:

- sort elements (ascending or descending) of the selected factor;
- — delete elements selected in the table from the group;
- — create a new set from the selected elements — a new group of elements is created which includes selected elements (these elements are deleted from the current group).

The **Steel. Factors** and **Resistance of sections** operations, which can be invoked from the **Information on the Element** dialog box (button in the filters toolbar) provide additional possibilities for the analysis of the load-bearing capacity of elements of steel structures included in the structural groups. These operations are implemented according to the same rules as the respective operations of **Kristall**[1].

Steel. Factors — enables to obtain a factors diagram for the selected element (Fig. 20.19-2*a*), based on the results of the check of sections. The **Show forces** checkbox enables to get an alternative presentation of information showing forces/stresses in an element (Fig. 20.19-2*b*).

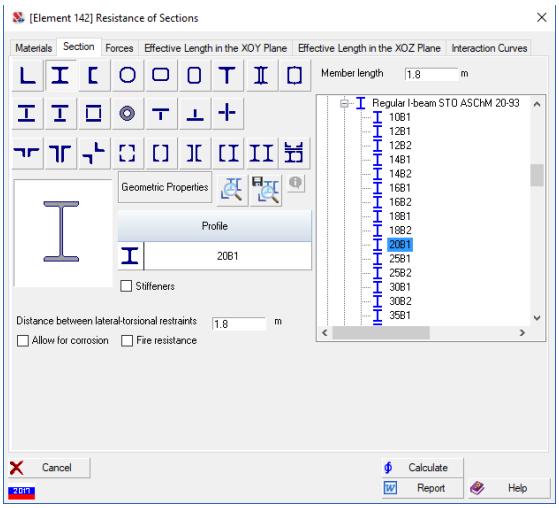


Figure 20.19-4. *The* **Resistance of Sections** *dialog box*

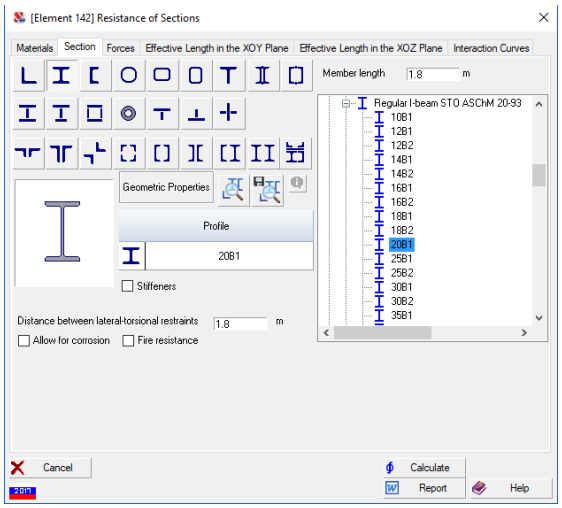


Figure 20.19-5. *The* **Truss Member** *dialog box*

Resistance of sections — enables to perform an expanded analysis of the load-bearing capacity of sections. The initial data for performing the check include the information on the finite element (length, section), and also the parameters of a structural group which includes the considered element. If a bar element is a part of the structural member, the length of the structural member will be used. Forces in the respective tab correspond to the DCF of this element. You can change different parameters, for example, the service factors, steel etc. during the check, which enables to analyze their effect on the load-bearing capacity of the element. If a structural group which includes the element is defined as a truss member, clicking the **Resistance of sections** button (Fig. 20.19-4) will invoke the dialog box with the parameters of the calculation of the truss members (Fig. 20.19-5).

20.20. Generating a Report

A report can be generated on the basis of the results of the check. It is generated in the **RTF** format and is automatically loaded into an application associated with this format (for example, MS Word or WordPad). The dialog box (Fig. 20.20-1) appears before generating the report, where you have to select

[1] V.S. Karpilovsky, E.Z. Kriksunov, A.A. Malyarenko, M.A. Mikitarenko, A.V. Perelmuter, M.A. Perelmuter, V.G. Fedorovsky, V.V.Yurchenko "SCAD Office. Implementation of SNiP in Computer-Aided Design Applications" — M.: SCAD SOFT, 2014 — 480p.

structural members and/or groups of structural members, information on which has to be included in the report.

The dialog box provides the following options of the output control:

- **For all groups and structural members** — the report will include all structural groups from the list;
- **Only for specified groups and structural members** — the report will contain only the information on the groups selected in the list;
- **For specified elements** — in this case you have to specify the numbers of finite elements for which the report will be generated in the text field.

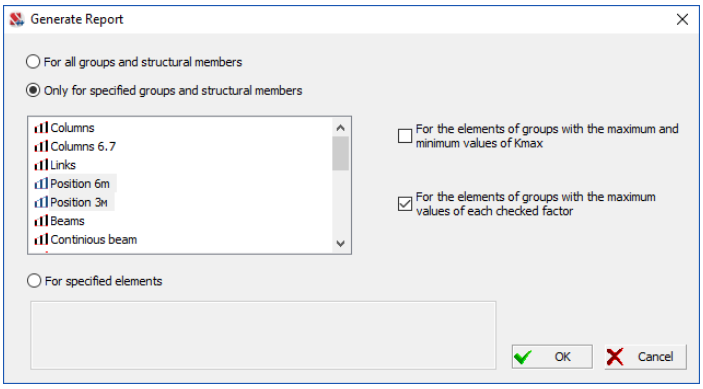


Figure 20.20-1. *The* **Generate Report** *dialog box*

Moreover, the **For the elements of groups with the maximum and minimum values of K_{max}** and **For the elements of groups with the maximum values of each checked factor** checkboxes enable to obtain a report only for the elements with the maximum values of the factors.

20.21. Extreme values of factors

This operation enables to generate a table (Fig. 20.21-1) for the selected group of structural members or structural members which includes the following information:

- list of factors;
- minimum and maximum value of each factor;
- numbers of finite elements where these minimum and maximum values took place.

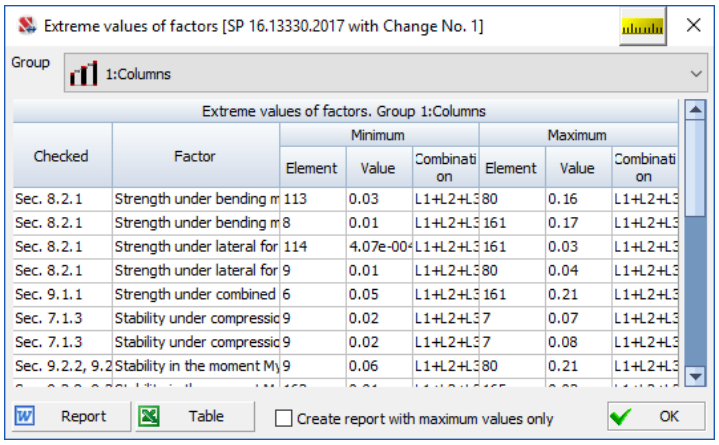


Figure 20.21-1. *The* **Extreme Factor Values** *dialog box*

The groups or structural members can be selected from the list in the upper part of the dialog box. The **Report** and **Table** buttons enable to obtain report documents for the selected group in the respective formats. Special checkbox allow to create report with maximum values only.

20.22. Create Elements of the Technical Specification for Steel

This operation enables to combine structural members and/or groups of structural members into special groups of elements of the structures which will be used later for the generation of the technical specification for steel (see Sec. 20.23).

This operation invokes the dialog box (Fig. 20.22-1) similar to that used when creating the unification groups.

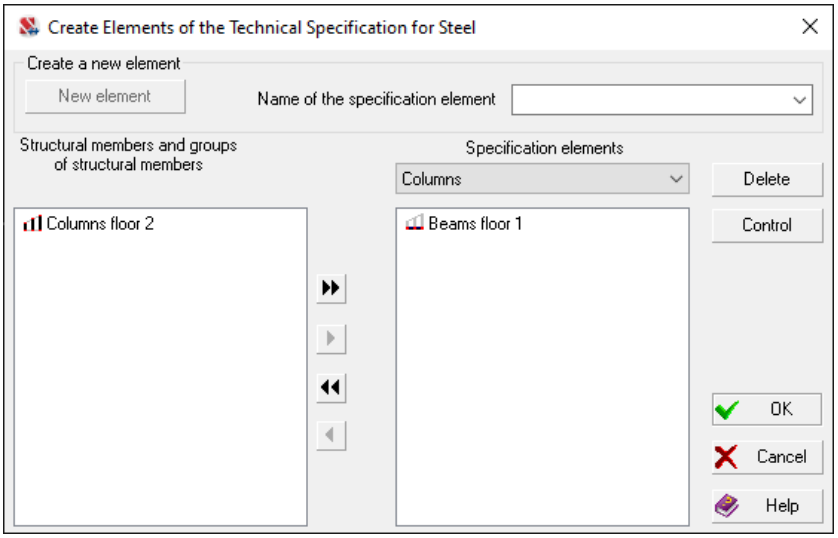


Figure 20.22-1. *The* **Create Elements of the Technical Specification for Steel** *dialog box*

The **Specification element name** drop-down list enables to specify the name of the element or select one of the standard names (which can be later edited). The following files contain the lists of the standard names:

%AllUsersProfile%\SCAD Soft\SteelSpecification1049.txt (when Russian is used)
%AllUsersProfile%\SCAD Soft\SteelSpecification1033.txt (when English is used)

The user can make changes to these files and thereby prepare their own list of standard specification element names.

The **Control** button allows you to open a dialog box (Fig. 20.22-2), in which you can change the order of displaying groups in the report document (**Up**/**Down** buttons) and / or change the names of the groups.

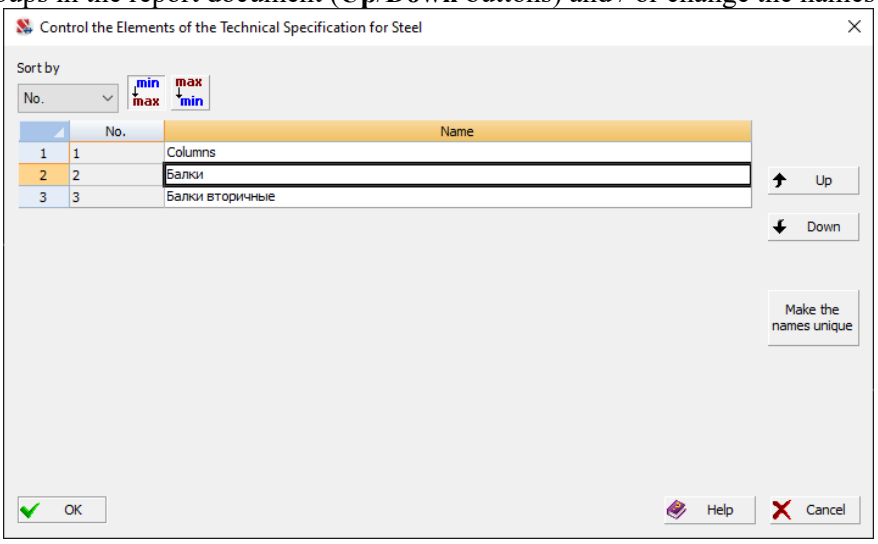
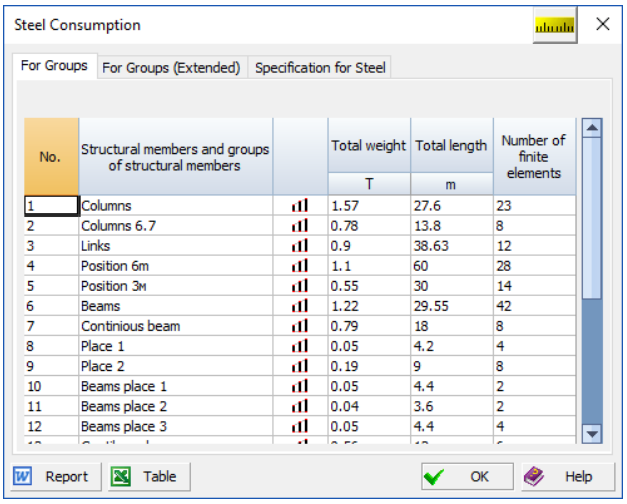


Figure 20.22-2. *The* **Control the Elements of the Technical Specification for Steel** *dialog box*

20.23. Steel Consumption

This operation enables to estimate the total consumption of steel for the design model. The dialog box invoked by this operation (see Fig. 20.23-1) is multi-tab.

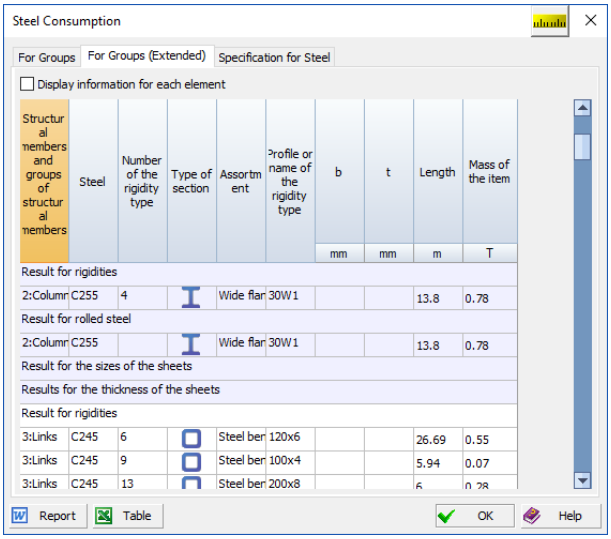
The first tab **For Groups** of the **Steel Consumption** dialog box enables to obtain information in the tabular form on the consumption of steel for each structural member and each group of structural members (the **Total weight** column), total length of the flexible parts of finite elements (the **Total length** column), and on the number of finite elements in the given group of structural members or in the given structural member (Fig. 20.23-1). Cross-sectional areas specified in the stiffness properties are used in the calculation of the weight of steel.



No.	Structural members and groups of structural members	Total weight	Total length	Number of finite elements
		T	m	
1	Columns	1.57	27.6	23
2	Columns 6.7	0.78	13.8	8
3	Links	0.9	38.63	12
4	Position 6m	1.1	60	28
5	Position 3м	0.55	30	14
6	Beams	1.22	29.55	42
7	Continious beam	0.79	18	8
8	Place 1	0.05	4.2	4
9	Place 2	0.19	9	8
10	Beams place 1	0.05	4.4	2
11	Beams place 2	0.04	3.6	2
12	Beams place 3	0.05	4.4	4

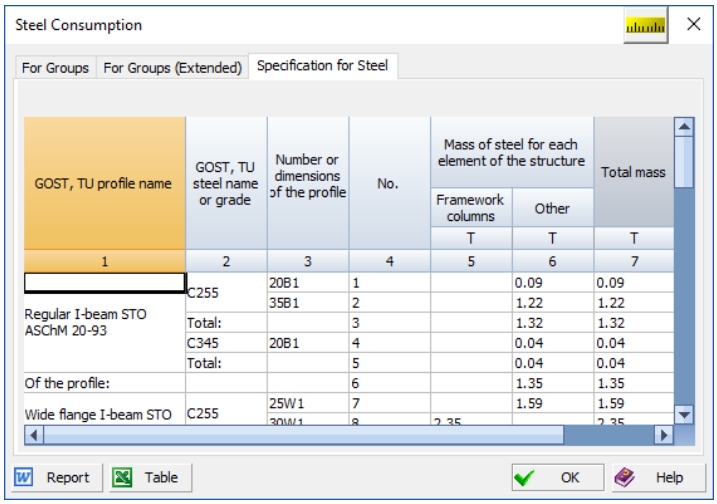
Figure 20.23-1. *The* **Steel Consumption** *dialog box*

The second tab **For Groups (Extended)** enables to obtain extended information in the tabular form on the material (the **Steel** column) and on the type of rigidity (the **Number of the rigidity type** and **Name of the rigidity type** columns) for each structural member and each group of structural members with the description of the type of section (the **Type of section** column). If a rolled section is specified, the **Assortment** column contains the name of the respective assortment, and the **Profile** column contains the profile number in this assortment. In the case of welded sections a table line is generated for each section element (flange or web) indicating the profile and dimensions of the structural steel. The length and mass are indicated for each item of the table (Fig. 20.23-2).



Structural members and groups of structural members	Steel	Number of the rigidity type	Type of section	Assortment	Profile or name of the rigidity type	b	t	Length	Mass of the item
						mm	mm	m	T
Result for rigidities									
2:Column	C255	4	I	Wide flan	30W1			13.8	0.78
Result for rolled steel									
2:Column	C255		I	Wide flan	30W1			13.8	0.78
Result for the sizes of the sheets									
Results for the thickness of the sheets									
Result for rigidities									
3:Links	C245	6		Steel ben	120x6			26.69	0.55
3:Links	C245	9		Steel ben	100x4			5.94	0.07
3:Links	C245	13		Steel ben	200x8			6	0.28

Figure 20.23-2. *The* **For Groups (Extended)** *tab of the* **Steel Consumption** *dialog box*



GOST, TU profile name	GOST, TU steel name or grade	Number or dimensions of the profile	No.	Mass of steel for each element of the structure: Framework columns	Mass of steel for each element of the structure: Other	Total mass
				T	T	T
1	2	3	4	5	6	7
Regular I-beam STO ASChM 20-93	C255	20B1	1		0.09	0.09
		35B1	2		1.22	1.22
	Total:		3		1.32	1.32
	C345	20B1	4		0.04	0.04
	Total:		5		0.04	0.04
Of the profile:			6		1.35	1.35
Wide flange I-beam STO	C255	25W1	7		1.59	1.59
		30W1	8	2.35		2.35

Figure 20.23-3. *The* **Specification for Steel** *tab of the* **Steel Consumption** *dialog box*

The third tab **Specification for Steel** is a table similar in form and content to the technical specification for steel according to GOST (Fig. 20.23-3), which is generated in accordance with the elements of the technical specification for steel created earlier (see Fig. 20.22-1) if the latter have been specified.

The **Report** and **Table** buttons invoke the generation of report documents in the respective formats.

20.24. Selection of Sections

The selection of sections of elements is performed in those cases when their cross-sections are from steel profiles. Structural members, groups of structural members, and unified groups can be involved in the selection. If the elements are combined into a unified group, all of them will have the same sections in the result of the selection.

The selection is performed using the assortment selected at the initial specification of the sections of elements, and only for the specified type of profile. This means that if an I-beam of type **K** has been selected as a profile, a similar profile will be selected in the result (or a message will appear that it is impossible to select a section from such profiles).

The results of the selection are displayed in the table of the **Results of the Selection of Sections** dialog box (Fig. 20.24-1). The name of the group or the structural member is output in the column of the table. The sections of elements the rigidities of which were used in the analysis of the structure, the initial section which was used in the selection, and the section obtained in the result of the selection (if it was performed) are output for the structural members. The following three columns provide information on the percent change of the main stiffness properties (EF, EI_y, EI_z) after the selection and the number of element in the model where this maximum took place. If the assortment was exhausted in the result of the selection, and the required load-bearing capacity was not reached, the following message will be displayed in red **Selection has failed**, and there will be no information on the selection in the table.

The name of the group is given in the table for the groups of structural members, and all the information regarding the results of the selection is output in the **Results of the Selection of Sections** dialog box [*group name*] (Fig. 20.24-2), which is invoked after double left-clicking on the line with the name of the group. The table in this dialog box contains the data on the results of the selection for each member of the group.

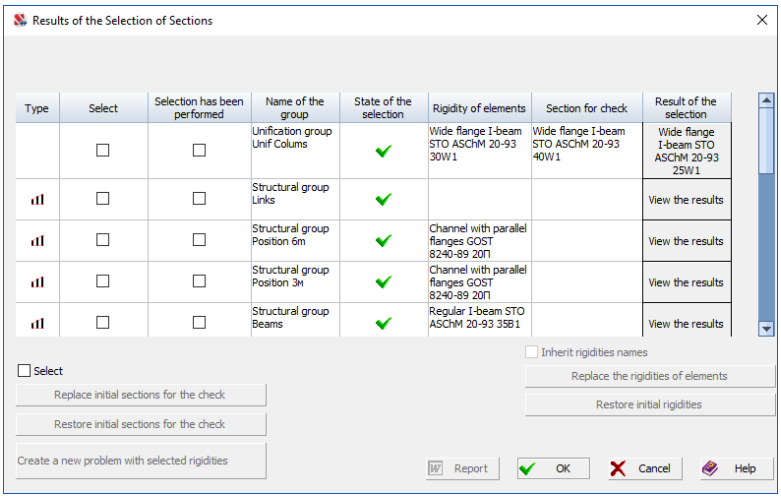


Figure 20.24-1. *The* **Results of the Selection of Sections** *dialog box*

If the selection could not be performed at least for one element of this group, the respective message will be displayed in red.

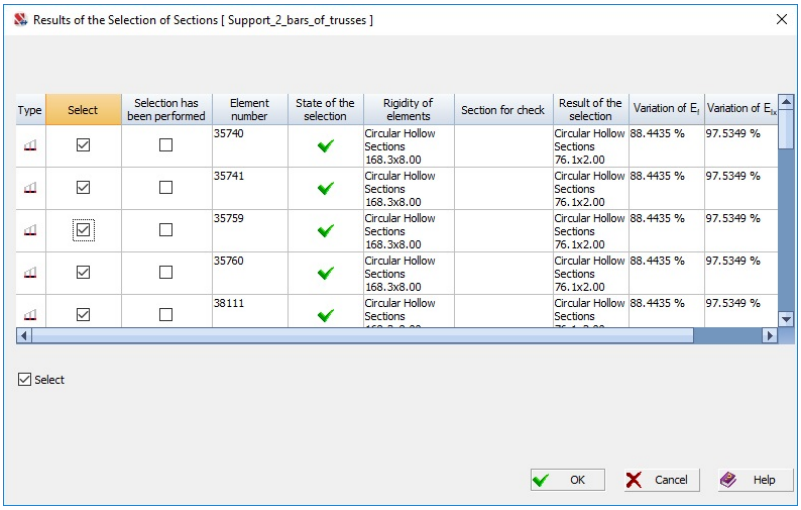


Figure 20.24-2. *The* **Results of the Selection of Sections** *dialog box*

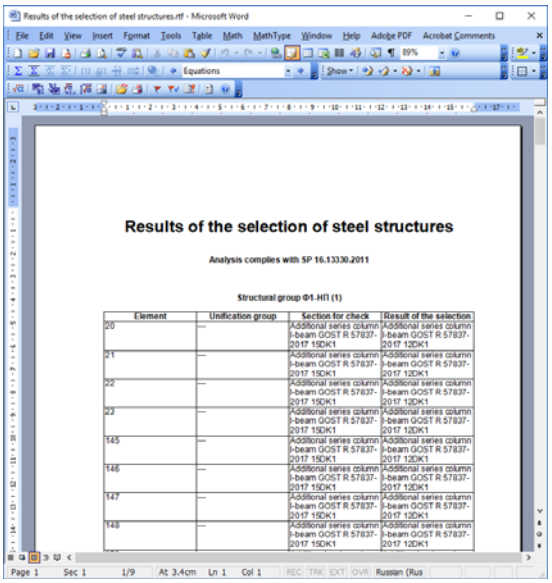
Only the name of the unification group is given in the table for unification groups, and all the information regarding the results of the selection is output in the dialog box which is invoked after left-clicking on the line with the name of the unification group.

A report can be generated in the result of the selection (the **Report** button). A report is generated in the **.rtf** format and is automatically loaded into an application associated with this format (Fig. 20.24-3).

If in the result of performing the operation of selection the sections of elements have changed, one of the following options can be selected in the **Results of the Selection of Sections** dialog box:

- save the results of the selection by creating a new task where the selected sections are used as the rigidities of elements;
- apply the selected sections as the rigidities of elements in the given problem. You should keep in mind that after performing this operation and finishing the work in the postprocessor a question will be asked concerning its confirmation, since this operation involves the recalculation of the problem;
- apply the selected sections as the initial sections for the check and further analysis;
- reject using the results of the selection (the **Cancel** button). The results of the selection will be saved and can be used later in the information on the results of the selection.

The above operations will be performed only for the elements with checked checkboxes in the first column of the table.



Results of the selection of steel structures

Analysis complies with SP 16.13330.2011

Structural group Ф1-Н11 (1)

Element	Unification group	Section for check	Result of the selection
20	—	Additional series column I-beam GOST R 57837-2017 15DK1	Additional series column I-beam GOST R 57837-2017 12DK1
21	—	Additional series column I-beam GOST R 57837-2017 15DK1	Additional series column I-beam GOST R 57837-2017 12DK1
22	—	Additional series column I-beam GOST R 57837-2017 15DK1	Additional series column I-beam GOST R 57837-2017 12DK1
23	—	Additional series column I-beam GOST R 57837-2017 15DK1	Additional series column I-beam GOST R 57837-2017 12DK1
145	—	Additional series column I-beam GOST R 57837-2017 15DK1	Additional series column I-beam GOST R 57837-2017 12DK1
146	—	Additional series column I-beam GOST R 57837-2017 15DK1	Additional series column I-beam GOST R 57837-2017 12DK1
147	—	Additional series column I-beam GOST R 57837-2017 15DK1	Additional series column I-beam GOST R 57837-2017 12DK1
148	—	Additional series column I-beam GOST R 57837-2017 15DK1	Additional series column I-beam GOST R 57837-2017 12DK1

Figure 20.24-3. *Report on the results of the selection of sections*

If the **Inherit the rigidity names** checkbox is checked in the **Results of the Selection of Sections** dialog box (Fig. 20.24-1), then after performing the selection and applying the selected sections the latter will have the same names of the types of rigidity as the initial sections.

The number of the types of rigidity can change considerably in the result of the selection. It is recommended to use the operation of deleting duplicate rigidities in the **Assign** section of the preprocessor before performing the calculation.

20.25. Information on the Results of the Selection

Information on the results of the selection given in the dialog boxes in Fig. 20.25-2, 20.25-3, is available only in the **Selection** mode. If for some reason after exiting from this mode you need to consider or use the results of the selection again, you should use the **Display information on sections** button— .

Information on the Work of the Postprocessor

If the operations of changing the rigidities of elements on the basis of the results of the selection (or on the basis of changing the sections in the groups of structural members) were performed in the **Steel** postprocessor, the **Results of the Selection of Sections** dialog box described above (Fig. 20.25-2) will appear when you exit to the project tree to confirm these operations.

21. Fire Resistance Analysis of Structural Elements

Fire resistance is the ability of structural elements to maintain strength under fire conditions. The fire resistance of a building depends on the fire resistance of its main structural parts. It is also necessary to take into account the variation of the thermal and mechanical properties of steel, concrete and reinforcement under the fire heating of the structure along with the mechanical actions.

21.1. Fire Resistance of Steel Structural Elements

The fire resistance analysis of individual unprotected elements of steel structures is implemented in **SCAD**. The check is performed by the critical temperature method according to DSTU-N B B.2.6-211:2016 (the recommendations of this standard are similar to EN 1993-1-2:2005). The method assumes that when the structure is heated by a fire, the temperature of the steel is uniformly distributed throughout the structure and its critical value is determined by the reserve of strength of the considered element. This reserve of strength is characterized by the load-bearing capacity utilization factor μ_0, the physical meaning of which corresponds to the coefficient K_{max}. Taking into account μ_0 the critical temperature is calculated by the following formula:

$$\theta_{a,cr} = 30,19 \ln\left(\frac{1}{0,967\mu_0^{3,833}} - 1\right) + 482.$$

The program determines the value of the load-bearing capacity utilization factor K_{max} from the combinations of loads which include only the values of constant and long-term loads at t = 0. The characteristic values of the strength of the material are used in the calculation (see, for example, SP 296.1325800.2017).

The fire resistance analysis is implemented for the following types of sections:

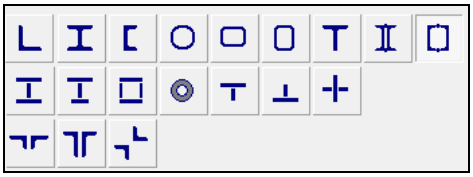


Figure. 21.1-1. Steel sections for which the fire resistance analysis is implemented

The critical temperature, time to reach this temperature, and the air and steel temperature-time curves are obtained in the result of the analysis.

The analysis is performed for one of the following types of the structural heating (Fig. 21.1-2):

- standard fire;
- hydrocarbon fire;
- external fire;
- smoldering fire.

The analysis can be performed for the following types of fire exposure:

- heating from four sides for all cross-sections;
- heating from three sides for all sections except for angles and hollow sections.

The temperature increase of steel at heating is calculated taking into account the change in the thermal conductivity and the specific heat of steel (Fig. 21.1-3).

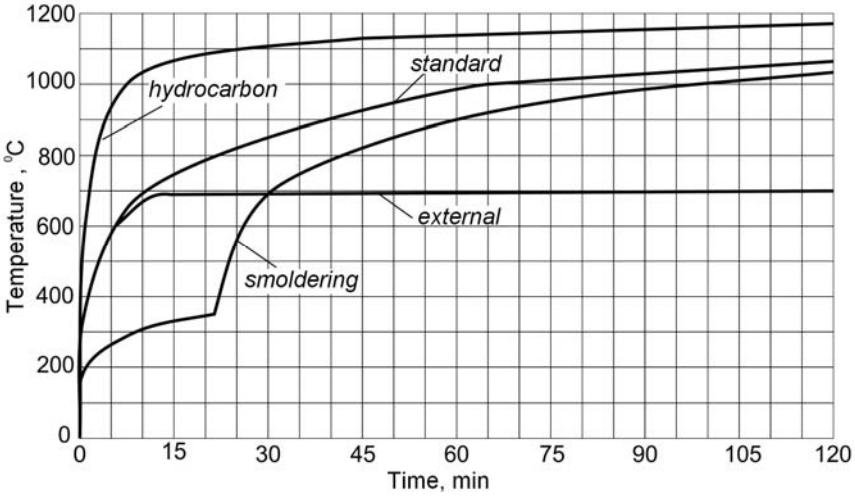


Figure 21.1-2. *Temperature-time curves*

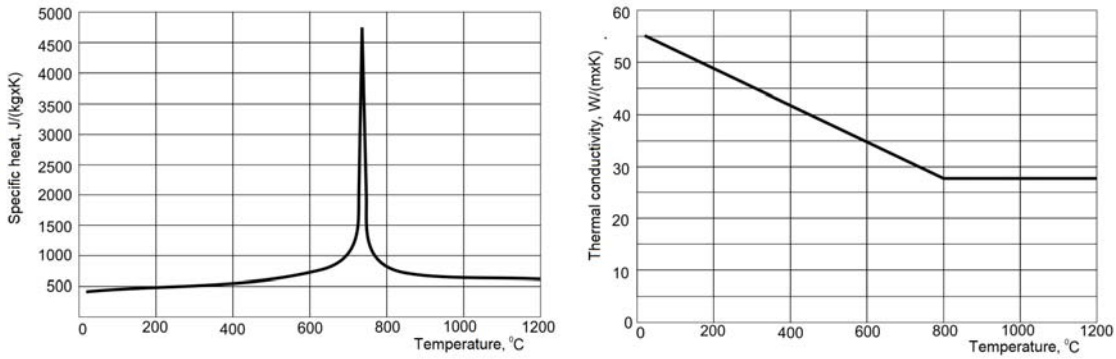


Figure 21.1-3. *Variation of the thermal properties of steel*

21.2. Fire Resistance of Reinforced Concrete Structural Elements

The fire resistance analysis of individual unprotected elements of reinforced concrete structures is implemented in **SCAD**.

Evaluation of fire resistance includes:

- thermal analysis — analysis of structural heating;
- strength analysis — analysis of the mechanical behavior of the structure.

A structure is fire resistant when the necessary safety margin can be provided for a certain period of time.

Thermal Analysis

The calculation is performed in order to determine the temperature distribution in a design cross-section, when the temperature field $T(y,z,t)$ is defined by the nonlinear transient heat equation

$$\frac{\partial}{\partial y}\left(\lambda\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(\lambda\frac{\partial T}{\partial z}\right)=c\rho\frac{\partial T}{\partial t},$$

where $\lambda(y,z,T)$ is the thermal conductivity factor, $c(y,z,T)$ is the specific heat, ρ is the density of the material. The relationship between these parameters and the temperature is given in the design codes.

The solution of the above heat equation is found for the boundary conditions describing the heat transfer from the hot air

$$\frac{\partial T(t)}{\partial n} = \alpha\left[\theta(t) - T(t)\right].$$

And the air heating is defined by the fire temperature θ(t), which is described by a certain time function.

The initial conditions considered when solving the heat equation describe a temperature field $T(y,z,0) = 20°C$ constant over the entire section.

Strength Analysis

The section strength check performed when evaluating the fire resistance differs from the usual case in that:

- a special (emergency) limit state is considered, and therefore only permanent and long-term loads are taken into account in addition to the fire action;
- the variation of physical and mechanical properties of materials (concrete and reinforcement) due to heating is taken into account, and since the temperature field is uneven, these characteristics are determined for each design point of the cross-section independently.

The program determines the value of the load-bearing capacity utilization factor $K_{\max}$ from the combination of loads which includes only the values of constant and long-term loads at t = 0. The characteristic values of the strength of the material are used in the calculation (see, for example, SP 296.1325800.2017).

The design strength of concrete R_{bt} and reinforcement R_{st}, as well as their elastic moduli E_{bt} and E_{st} are used taking into account the values R_b, R_s, E_b and E_s for the given classes of concrete and reinforcement, but they are adjusted for the heating temperature T,°C according to the formulas

$$R_{bt} = R_b\gamma_{bt}; \quad R_{st} = R_s\gamma_{st}; \quad E_{bt} = E_b\beta_b; \quad E_{st} = E_s\beta_s.$$

Moreover, the values of the critical strain of reinforcement ε_{s2} and concrete ε_{b2} depend on the temperature as well, because the "stress-strain" relationship changes with increasing temperature.

The following factors are obtained in the result of the analysis:

- Strain of concrete in a normal section;
- Strain of reinforcement in a normal section;
- Strength in a strip between oblique sections under the lateral force;
- Strength of an oblique section under the lateral force.

Peculiarities of the implementation

Unfortunately, there are many errors and typos in STO 36554501-006-2006 (and the Manual to this STO). Here are some corrections that were made when implementing the fire resistance analysis.

- The formula for determining specific heat provided in Sec. 6.3 of STO 36554501-006-2006 has a minus sign, which is obviously a typo, because the respective formulas in MDS 21-2.2000 and other sources have a plus sign.
- The proportionality factor given in Sec. 9.3 of STO 36554501-006-2006 has incorrect units of measurement (W m$^{3/2}$/kg). The correct units are W m$^{5/2}$/kg (see *Recommendations for the Protection of Concrete and Reinforced Concrete Structures from Brittle Fracture*, M., Stroyizdat, 1979).
- According to Sec. 5.11 of STO 36554501-006-2006 the value ε_{s2}=0,0025 at t_s = 20 – 200 °C. This is obviously a typo, and the value ε_{s2}=0,025 should be used (see Sec. 6.2.14 of SP 63.13330), because there must be a correspondence between the normal calculation and that taking into account the fire action. For similar reasons, the value ε_{b2}=0,0035 (see Sec. 6.1.20 of SP 63.13330) is used instead of the value ε_{b2}=0,00035 given in Sec. 5.7 of STO 36554501-006-2006 (or ε_{b2}=0,000035 given in Sec. 8.2 of the Manual to STO 36554501-006-2006).

21.3. Results of the Fire Resistance Analysis

The following approach is used for the unified presentation of the results of the fire resistance analysis for both steel and reinforced concrete structures. For any element the result is a factor that is equal to

- the maximum of the factors listed above at the time specified in the calculation parameters – for the reinforced concrete elements;
- the ratio of the required fire resistance specified by the user (in the calculation parameters) to the time it takes to reach the critical temperature – for steel elements.

21.4. Preparation of the Initial Data, Calculation and Analysis of Results

The **Fire Resistance** postprocessor is invoked from the **Postprocessors** toolbar. The analysis is performed for bar and plate elements. Reinforcement must be specified for reinforced concrete structural members (either "manually" or obtained in the result of the analysis), and steel elements must belong to one of the groups of structural members.

The following procedure is recommended for performing the analysis:

- create groups of elements combining bars or plates subjected to the same types of fire into one group;
- select the type of fire in the **Temperature-time curve** drop-down list and specify the control fire resistance time in the **Parameters of Fire Resistance Analysis** dialog box;
- perform the calculation.
- The results of the analysis are given as the utilization factors of restrictions and can be displayed with the help of a full color or 3-color scale.

The Fire Resistance Toolbar

Figure 21.4-1. *The* **Fire Resistance** *toolbar*

The **Fire Resistance** toolbar (Fig. 21.4-1) includes the following buttons:

— invoke the calculation;

— generate report;

— specify parameters of the fire resistance analysis;

— 3-color scale for displaying calculation results;

— full color scale for displaying calculation results;

— color scale for reduced thickness;

— color scale for proper fire resistance limit;

— generate report for steel members;

— create groups of bar elements;

— create groups of plate elements;

— invoke the operation of creating a group;

— cancel the selection of elements;

— select active group of fire resistance;

— show elements of the active group;

— control the groups of fire resistance;

— select the design codes (Steel);

— select the design codes (Concrete).

Moreover, the toolbar contains the list of groups of fire resistance.

Specifying the Parameters of Fire Resistance Analysis

The design parameters are specified in the respective dialog box (Fig. 21.4-2). They include the selection of a temperature-time curve (type of fire) and the control fire resistance time (required fire resistance). The program enables to perform the analysis for standard fire, external fire, smoldering fire and hydrocarbon fire.

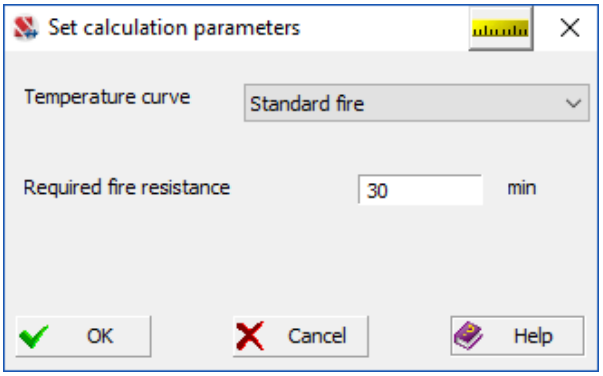


Figure 21.4-2. *The* **Parameters of Fire Resistance Analysis** *dialog box*

Creating a New Group of Bar Elements

Elements with different types of rigidity, reinforcement parameters and belonging to different structural groups can be included in one group of bar elements. The main condition is that all elements of a group must be subjected to the same type of fire.

In order to create a group, click the button — create groups of bar elements, select the elements in the model and click the button . The **Fire Resistance Group** (bars) dialog box (Fig. 21.4-3) will appear.

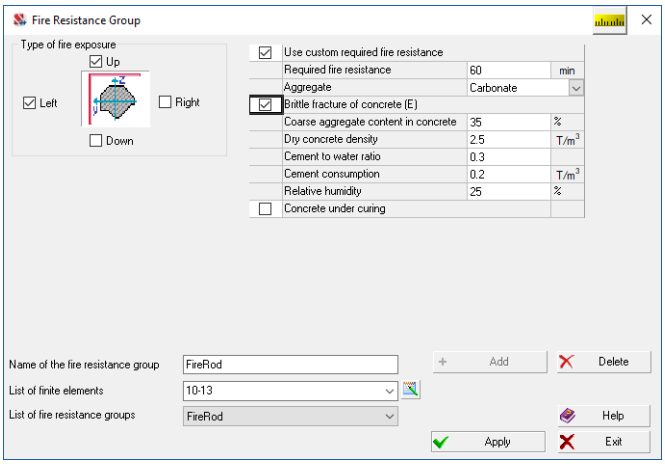


Figure. 21.4-3. *The* **Fire Resistance Group** *dialog box* (bars)

Specify the name of the group, select the position of the heat source with respect to the local axes of the elements of the group (using checkboxes **Up**, **Down**, **Left**, **Right**), and click the **Apply** button in this dialog box.

Moreover, you can specify the custom required fire resistance for a group using the **Use custom control required fire resistance** checkbox and the respective edit field. If this checkbox is not checked, the required fire resistance specified in the general parameters of the fire resistance analysis will be used. If you want to perform the brittle fracture analysis of concrete, check the respective checkbox and specify the necessary data (cement to water ratio, cement consumption, coarse aggregate content in concrete, ...).

Close the dialog box by clicking the **Close** button and confirm the specified data in the information window (Fig. 21.4-4).

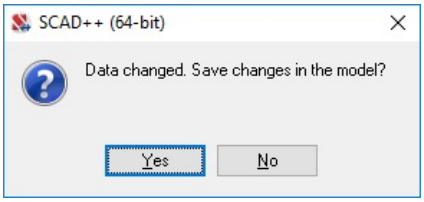


Figure 21.4-4. Information window

Modifying the Existing Fire Resistance Groups

In order to modify a fire resistance group, select it from the list in the toolbar and click the button . Make the necessary changes in the **Fire Resistance Group** (bars) dialog box and click the **Apply** button. If you want to modify other groups, they can be selected directly from the list of fire resistance groups in the dialog box. Click the **Exit** button once you have finished and confirm the changes in the information window (Fig.).

If the list of elements of the group is large, you can use the **Edit the list** dialog box, which is invoked by clicking the respective button — .

Deleting the Fire Resistance Groups

In order to delete a group, it is necessary to select it in the list of groups in the **Fire Resistance Group** dialog box and click the **Delete** button.

Creating and Modifying Fire Resistance Groups of Plate Elements

The fire resistance groups of plate elements are created in the **Fire Resistance Group** (plates) dialog box according to the same rules as the bars (button —). The procedure for modifying groups also coincides with that described for bars. Plates with different types of rigidity can be included in one group.

If you check the **Thermal insulation capacity** checkbox and specify the initial temperature in the case of the one-sided heating, you can perform the respective analysis.

Control of the Fire Resistance Groups

The control of the fire resistance groups is performed in the respective dialog box (Fig. 21.4-5) according to the same rules as the groups of steel and reinforced concrete structural members.

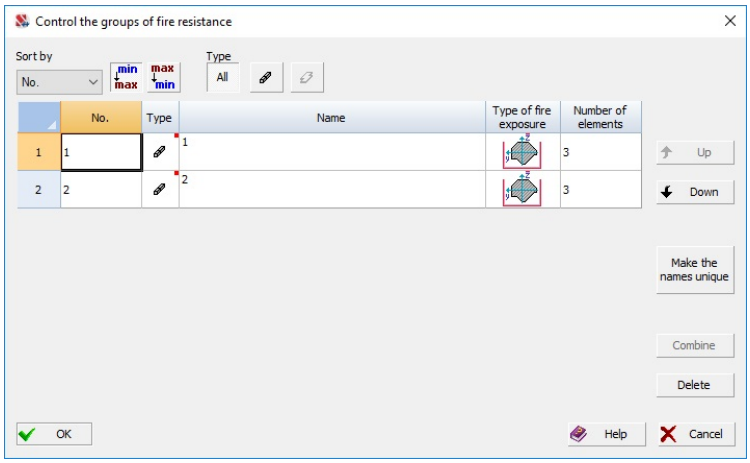


Figure. 21.4-5. *The* **Control of the Fire Resistance Groups** *dialog box*

Selecting the Design Codes

The buttons and are indicators of the selected codes and enable to invoke the **Setting the Design Codes** dialog boxes (Fig. 21.4-6, 21.4-7), where they can be selected. Fire design standards for steel structures EN 1993-1-2:2005 and the respective Ukrainian standard DSTU-N B V.2.6-211:2016, as well as the standard RF STO 36554501-006-2006 for reinforced concrete structures are implemented in the current version of the software.

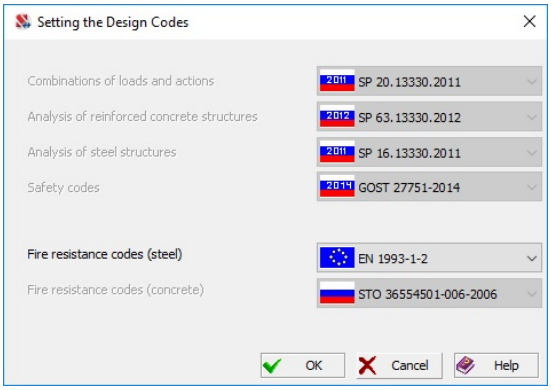


Figure. 21.4-6. Setting the design codes (steel)

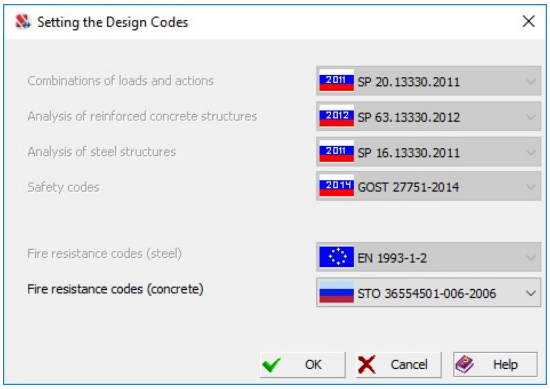


Figure. 21.4-7. Setting the design codes (reinforced concrete)

21.5. Fire Resistance Analysis of Elements

Like in the case of the analysis of steel or reinforced concrete structural members, results of the fire resistance analysis can be displayed as a 3-color or as a full color scale .

If not a single fire resistance group is selected, the visualization of one of the factors selected in the drop-down list is performed:

- critical factor K_{max};
- critical factor for load-bearing capacity;
- factor for brittle fracture of concrete;

- factor for thermal insulation capacity.

After performing the calculation, you can also perform the fire resistance analysis of an individual element with the help of the **Information on the Element** mode. If the fire resistance analysis was performed for the element included in one of the groups of reinforced concrete structural members, the **Fire Resistance** tab will appear in the **Reinforced Concrete. Check** dialog box (Fig. 21.5-1).

You should keep in mind that the fire resistance analysis is performed not for all types of steel sections. If this analysis can be performed for the section of the considered element, you can check the **Fire Resistance** checkbox in the **Section** tab of the **Resistance of Sections** dialog box, and the **Fire Resistance** tab will appear (Fig. 21.5-2).

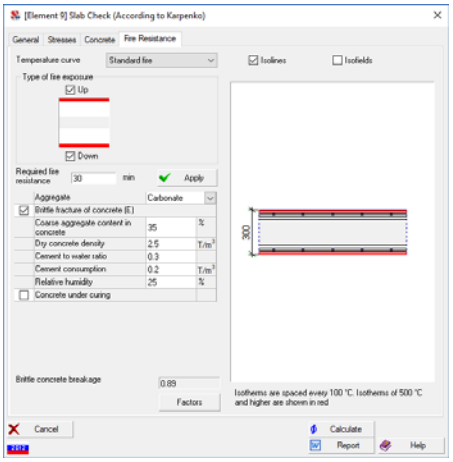


Figure. 21.5-1.*The* **Fire Resistance** *tab for reinforced concrete structural elements*

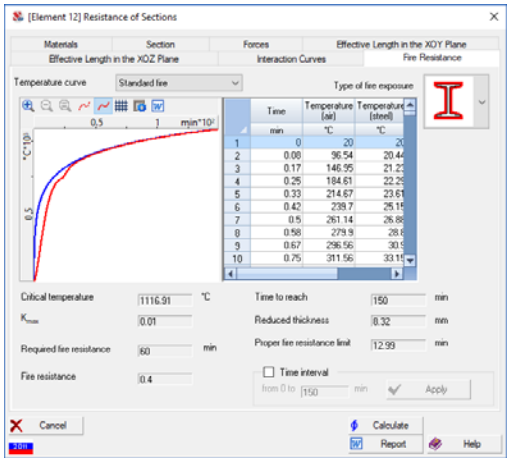


Figure. 21.5-2. *The* **Fire Resistance** *tab for steel structural elements*

The results of the calculation for steel structural elements include the value of the critical temperature, time to reach this temperature, maximum value of the utilization factor of restrictions obtained in the result of the strength analysis, required fire resistance specified for the analysis and its respective fire resistance coefficient (the ratio of the required fire resistance to the time it takes to reach the critical temperature of this element). Moreover, the program also generates the graphs and the table of the variation of the air temperature and of the temperature in the section of the element in the given time range.

The results for reinforced concrete structural elements include the maximum value of the utilization factor of restrictions for fire resistance, and the name of this factor, the factors diagram (invoked by clicking the respective button), as well as the temperature distribution in a section (isotherms), given as isofields or isolines.

Color scales of the reduced thickness and the proper fire resistance limit can be obtained for the steel structural elements, and a report on the steel members can be generated.

The reduced thickness is the ratio of the cross-sectional area to the heated perimeter; the proper fire resistance limit without fire protection is the time it takes to reach the temperature of 500°C in a standard fire, which causes significant change in strength and deformability (see Table 6 of the Manual for determining the limits of fire resistance of building structures, fire hazard parameters of materials. Procedure for the design of fire protection. M. 2013 or Annex B STO APCC 11251254.001-018-03).

The report contains information on the reduced thicknesses and the area of fire exposure. This report can be used to assign a method of fire protection of steel members and determine the amount of material needed for this.

More information about the **Fire Resistance** tab can be found in the descriptions to **ARBAT** and **Kristall**.

22.Calculation of Loads from the Structural Fragment

SCAD enables to output the values of the reactions in nodes with constraints. However, the detailing is required in many cases for large structural models when the behavior of individual loaded fragments each of which requires different detailing is studied. A typical example is a structure with strong stress concentrators for the calculation of which a refined finite element mesh should be used in the area of the concentrator. In such cases you should perform the general calculation of the system using a coarser finite element meshing, and then separately consider the fragment with the stress concentrator.

A similar situation arises in those cases when a coarse mesh is suitable for some purposes (for example, for the determination of the loads on the foundations), and a more detailed approach should be used for other purposes (for example, a detailed study of the stress-strain state of a standard floor slab and the selection of reinforcement in this part of the structure).

The use of the fragmentation implies that the action of the dropped part of the system is modeled by loads transferred by it to the selected fragment, i.e. if we separate a part of the structure, its action on the remaining part is reduced to some loads. In order to improve the accuracy of the solution it is recommended to create a fragment "with a reserve" of 2-3 rows of finite elements from the considered border (Fig. 22-1).

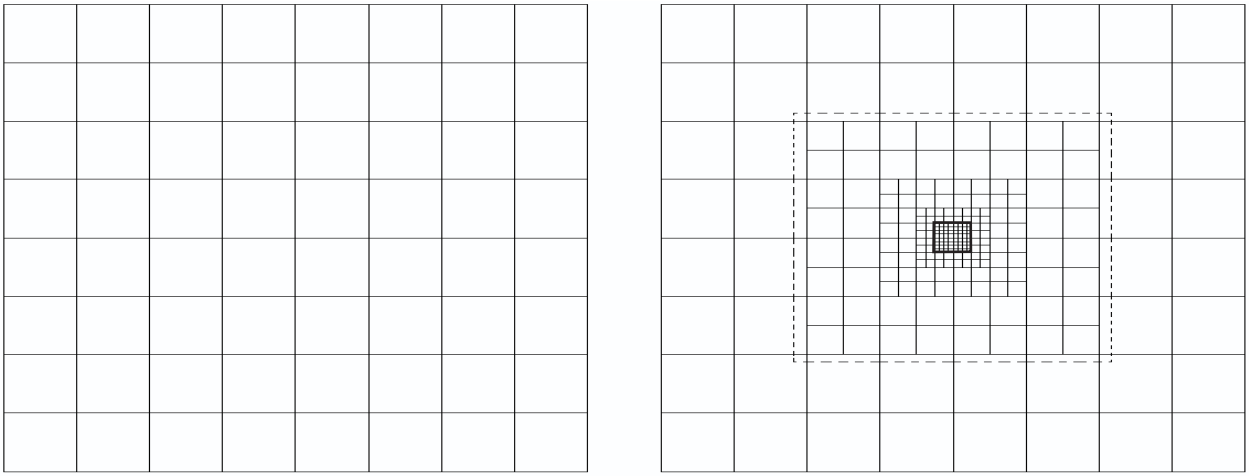


Figure 22-1. *Selection of the fragment*:
on the left — initial system; *on the right* — fragment with a mesh refinement area on the distance from the selection border

The mode of the calculation of loads from the structural fragment enables to determine the loads in the given nodes from the separated part of the model, for example, from the superstructure on the foundations. The loads can be saved as a model. It means that a system will be created consisting only of nodes and without any elements, and the loads on these nodes will be known. Then a detailed design model can be created on these nodes (and on the other additionally included ones) and its calculation can be performed. The analysis of the results of the calculation of loads from the structural fragment is described in Sec. 22.3.

Groups of nodes where the loads have to be determined and groups of elements adjacent to these nodes serve as the initial data for the calculation.

When calculating the loads on the foundations the rotation angle of the part of the foundation about the Z axis is also specified if the axes of inertia of columns do not coincide with the axes of the foundation under these columns. The loads can be calculated for each loading or for the combinations of loadings.

At the analysis of forces (reactions) acting on the fragment of the design model when there are some dynamic loadings (for example, at the calculation for a seismic action) a situation may arise when the obtained results do not satisfy the conditions of equilibrium and their values are much greater than the expected ones. The thing is that the result of the

calculation for this dynamic loading is obtained with the help of a nonlinear procedure for displacements or forces of all the considered modes of natural oscillations. The squares of the results are summed for each mode, and then the square root is taken of this sum. Forces in individual sections and displacements of individual points which occur in the process of seismic oscillations and are obtained according to the "root of the sum of squares" rule ***correspond to different moments in time*** and, therefore, violate the conditions of equilibrium.

22.1. Input Initial Data

Data for the calculation can be prepared during the generation of the design model or in the process of the graphical analysis of the results by specifying the groups of nodes and elements in the **Groups** toolbar. Additional information is specified in the **Special Initial Data** section of the **Project tree** with the help of the **Loads from the structural fragment** operation. Moreover, this information can be specified in the preprocessor (**Special** tab). In this case, if the user has previously selected nodes/elements, then the *Selected* item will appear in the list of groups of nodes/elements (see below).

There are two methods for specifying the initial data. The first method involves the specification of elements from which the loads have to be obtained (cut elements) and the explicit specification of nodes in which the loads will be obtained. In the second case the whole isolated fragment is selected. The loads from it on the rest of the structure including the reactions in constraints will be obtained in all nodes lying on the boundary of the selected fragment. Nodes lying on the boundary of the fragment include the nodes adjacent to the part of the model not included in the fragment as well as the nodes with constraints.

When the first method is used the following procedure for the data preparation is recommended:

- invoke the function of the specification of the groups of elements in the **Groups** section — ;
- select the elements in the model (for example, all or only the elements adjacent to the nodes in which the loads will be obtained);
- save the group of elements having specified its name — ;
- invoke the function of the specification of the groups of nodes — ;
- select the nodes in model in which you have to obtain the loads;
- save the group of nodes having specified its name — ;
- repeat the two previous operations for each group of nodes belonging to other parts of the model.

It should be noted that there can be several groups of nodes. This is due to the fact that the parts of the foundation can have different rotation angles.

The procedure for the second method repeats the first three operations described above. However, when saving the group of elements the **Create a group of connected nodes** operation is invoked and the group of nodes lying on the boundary of the selected fragment is automatically generated.

Moreover, when adding a group of elements, you can use the **Create a group of connected nodes** checkbox, which enables to automatically add nodes connected to the group of added elements to the list of nodes.

The calculation of loads can be performed in the general cycle of solving the problem, or independently in the postprocessing mode. In the first case the initial data for the calculation must be prepared before performing the main calculation. In the second case they can be also prepared after completing the main calculation.

22.2. Description of Fragments

In order to perform the calculation of loads from the structural fragment, you have to specify the additional information which defines the calculation mode (for loadings and/or for combinations of loadings), includes the list of elements coming into the nodes in which the loads are calculated, and the lists of nodes.

Additional information is specified in the **Special Initial Data** section of the **Project tree** in the **Loads from the Structural Fragment** dialog box (Fig. 22.2-1), where the following procedure is recommended:

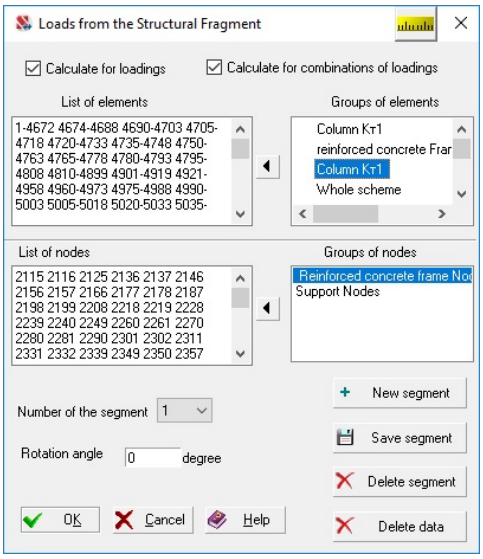


Figure 22.2-1. *The* **Loads from the Structural Fragment** *dialog box*

- select the calculation mode — for loadings and/or for combinations of loadings;
- select a group which is specified for the calculation of loads from the structural fragment in the list of groups of elements;
- click the button [+ <<], and the numbers of elements of the selected group will be input into the **List of elements** field (the list can be edited directly in this field);
- click the **New segment** button (here we deal, for example, with a part of the foundation in the nodes of which the loads are calculated);
- select the name of the group belonging to this segment in the **Groups of nodes** list;
- click the button [+ <<], and the list of nodes of the selected group will be input into the **List of nodes** field;
- specify the rotation angle of the part of the foundation to which the nodes of the first group belong about the Z axis of the global coordinate system (0° by default).

Different segments can not have the same node numbers.

If it is necessary to calculate several parts of the foundation with different angles of orientation with respect to the Z axis of the global coordinate system, click the **New segment** button and repeat all the above actions with the groups of nodes for the new part having specified its angle of orientation, etc.

Finish the input of data by clicking the **OK** button in the dialog box.

The set of nodes of each segment can be edited in the **List of nodes** field. Moreover, you can delete the incorrectly specified segments. In order to do it select the number of the segment you want to delete in the **Number of the segment** drop-down list and click the **Delete segment** button. If it is necessary to exclude the data for the calculation of loads from the structural fragment from the model, click the **Delete data** button.

22.3. Graphical Analysis of the Results of the Calculation of Loads from the Structural Fragment

The mode enables to display on the design model the nodal loads obtained in the result of the calculation of loads from the structural fragment , and to save the nodes with the loads applied to them as a separate model.

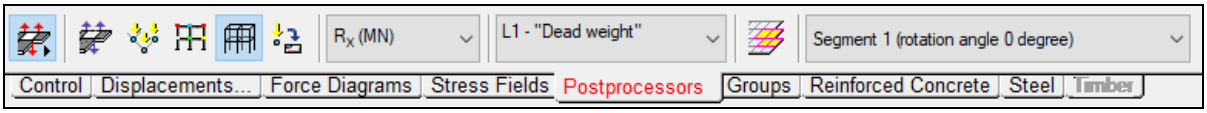


Figure 22.3-1. *The toolbar of the* **Analysis of loads from the structural fragment** *mode*

The toolbar of this mode (Figure. 22.3-1) includes the following operations:

— display loads from the structural fragment for the selected loading;

— display the color map of loads from the structural fragment;

— display the design model;

— save the nodes with the calculated loads as a separate model;

— show cut elements;

— change initial data.

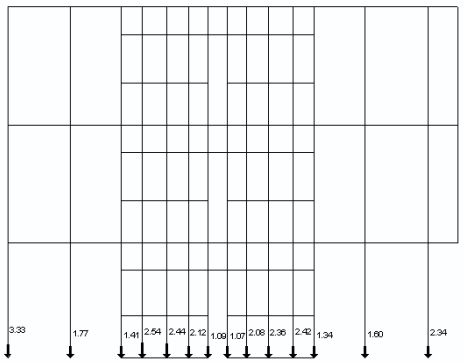


Figure 22.3-2. *Loads from the fragment in the nodes of the subgrade*

The toolbar also contains two lists — for selecting the type of load and for selecting the number of loading or the combination of loadings.

Since the work in this mode does not differ from the modes of the analysis of the results described above, let's focus only on the operation of saving nodes with the calculated loads as a separate model. This operation enables to save the model which includes only the nodes of the initial model where the loads from the fragment are obtained under a new name. All loadings and their combinations for which the loads were calculated will be automatically generated in the new model.

You can attach the obtained model to another one created earlier (the assembly mode), or generate a model on the basis of its nodes.

The filters button enables to output the values of loads. The loads can be output for all nodes or only for the nodes included in a certain segment (in this case you have to select the respective segment in the drop-down list of the toolbar, and if you want to output the data for all segments select the **All nodes** item in this list).

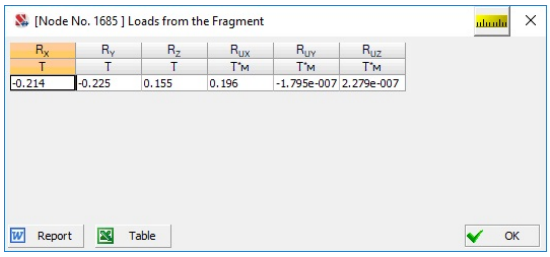


Figure 22.3-3. *Loads from the structural fragment for the selected node*

The button in the toolbar enables to highlight the elements listed in the initial data as "dropped".

Clicking the **Loads from the fragment** button in the **Information on the Node** dialog box enables to obtain the table with the values of loads from the structural fragment for the selected node for the current loading (or combination of loadings) (Fig. 22.3-3).

Button activates dialog box for changing initial data. After usage of this operation you have to go to the project tree and calculate *Loads from the Structural Fragment***.**

23. Progressive Collapse Analysis

The "progressive collapse " concept refers to a situation when the failure or damage of any small part of the structure leads to a complete or nearly complete destruction of the whole structure. Emergencies can be caused by the human activities or by the natural phenomena. The first case includes gas explosions, terrorist attacks, fires, defects in design, construction, and maintenance of buildings, their unqualified reconstruction (with superstructures, extensions, replanning of premises), resulting in weakening or overloading the bearing elements and subgrade etc. Emergencies can be caused by such natural phenomena as earthquakes, hurricanes, landslides, non-uniform deformation of the subgrade.

Since it is impossible to completely eliminate the risk of emergencies or emergency impacts, it is necessary to provide a certain degree of safety of people in buildings and of their property by reducing the possibility of the progressive collapse at the local failures of load-bearing structures.

In order to prevent the progressive collapse the general strengthening, local strengthening, and interconnection between elements can be provided. Most US codes prefer the general strengthening when the failure of one of the elements of the building does not lead to the destruction of the whole structure. The local strengthening, i.e. strengthening of the most sensitive places, is very difficult to standardize for including in the design codes, since it is necessary to understand the nature of the possible actions on the building, including the terrorist attacks. Structural interconnections between the elements or the continuity of the structure are other methods of the general or local strengthening.

The main provisions of the current regulations for the prevention of progressive collapse can be given as follows:

1. The load-bearing system of residential buildings must be resistant to the progressive (chain) collapse in the case of the local failure of individual structures at the emergency impacts (gas explosion, fire, etc.).

2. Local failures of individual load-bearing structures are allowed, but these primary failures must not lead to the collapse of the adjacent structures which receive the load previously taken by the elements damaged in the result of the emergency action.

3. The structural system of the building must provide its strength and stability, at least for the time necessary to evacuate people. The displacements of structures and the opening of cracks in them are not limited.

4. The resistance to the progressive collapse is checked by the calculation for the special combination of loads and actions which includes permanent and temporary long-term loads, and also the effect of hypothetical local failures of the load-bearing structures.

5. The design characteristics of the materials are enhanced with the help of special safety factors. Moreover, the values of the design strength are multiplied by the service factors which take into account the low probability of the emergency impacts and the growth of concrete strength after the erection of the building, and also the post-yield behavior of the reinforcement.

The mode is implemented in **SCAD** in accordance with the current regulations and is designed to model the behavior of buildings and structures in the case of emergency impacts which caused the local failures of individual vertical load-bearing elements. The authors also took into account the apparent arbitrariness of the initial assumptions:

- there is no reliable information on the location and cause of the process and its history;
- real failure parameters can be very different from the strength conditions given in the codes, since it is known that the design values of the strength parameters may differ significantly from those observed in nature.

Structures cannot be completely free from the risk of collapse because of the uncertainties of the requirements to the system, dispersion of mechanical properties of building materials, difficulties of adequate modeling of the behavior of the system even when the modern software are used. Moreover, it is impossible to design and build an absolutely safe structure without taking into account the cost of the prevention of emergencies.

Thus, we can obtain a qualitative assessment of the stability characteristics of the structure with respect to the progressive collapse in the result of the numerical modeling, and compare several possible collapse scenarios in order to detect weak areas in the structure.

The progressive collapse analysis is based on the following provisions:

- a model obtained in the result of the structural analysis and the subsequent selection of the reinforcement in the elements of reinforced concrete structures and sections in the elements of steel structures is used as the initial model of the building for the progressive collapse analysis;
- elements of the design model which model the suddenly removed structural members are combined into groups; the number of structural members which failed simultaneously (collapsed) is not limited;
- the calculation is performed for the combination of loadings which includes the characteristic values of permanent loads and temporary loads (sustained) with coefficient 1;
- dynamic amplification factors are introduced to take into account the suddenness of the removal of the structural members and the effect of falling of the collapsed structures;
- the check of the elements of reinforced concrete and steel structures included in the design model after the sudden removal of members is performed taking into account only the ultimate limit state;
- design strength and strain characteristics of the materials are assumed equal to their characteristic values;
- since large displacements usually occur in the result of the progressive collapse analysis, it is recommended to perform the calculation in the geometrically nonlinear formulation.

23.1. Data Preparation

The progressive collapse analysis is performed in two stages. The first stage includes the following actions:

- static and dynamic (if necessary) calculations in order to determine the stress-strain state of the structure under normal conditions of operation;
- selection of the design combinations of forces (DCF);
- selection of reinforcement in the elements of reinforced concrete structures taking into account the ultimate and serviceability (crack resistance) limit states;
- check and selection of rolled sections of the elements of steel structures.

The Progressive Collapse Dialog Box

In order to perform the second stage it is necessary to specify the additional data in the **Progressive Collapse** dialog box (Fig. 23.1-1). The dialog box appears after invoking the respective operation in the **Special Initial Data** section of the project tree.

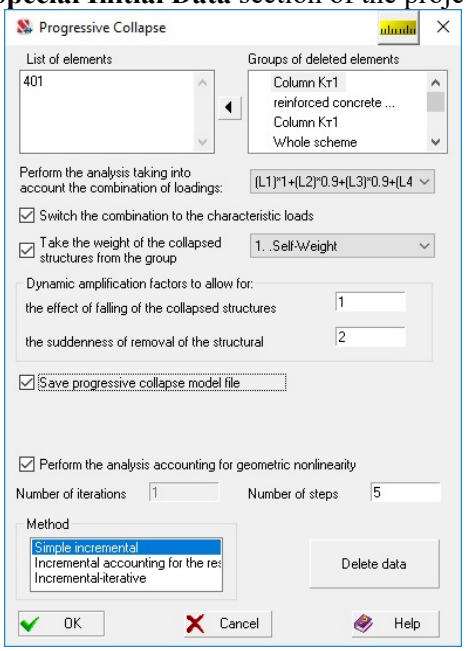


Figure 23.1-1. The **Progressive Collapse** *dialog box*

Moreover, this information can be specified in the preprocessor (**Special** tab). In this case, if the user has previously selected elements, then the *Selected* item will appear in the list of groups of elements (see below).

The additional data include:

list of finite elements included in the suddenly removed structural member;

control combination of loadings;

group of loads defining the weight of the collapsed structures (if the respective checkbox is checked);

overload factor (dynamic amplification factor) — Kf for adjusting the reaction of the system at the sudden removal of a structural member (Kf = 2 by default);

overload factor — Kg for adjusting the reaction of the system to the sudden collapse of the failed structures (Kg = 1 by default).

If a nonlinear calculation is performed, it is necessary to specify the method of the calculation and the corresponding parameters (number of steps, number of iterations).

Numbers of the suddenly removed elements are specified in the **List of elements** field. If such elements have been combined in one or several groups, they can be loaded by selecting the respective groups in the **Groups of deleted elements** list and transferring them into to the **List of elements** field by clicking the button «.

The list of groups includes all the generated groups of elements. The suddenly removed elements can be included in any group. In this case the adjustment of the list of elements is performed after they are loaded into the **List of elements** field.

The **Progressive Collapse** dialog box will be available only if the initial data includes groups of elements (assuming that among these groups of elements there are groups containing the lists of suddenly removed elements), and combinations of loadings (assuming that among these combinations there is such a combination which includes the necessary set of permanent and long-term loads for the action of which the damaged structure is checked). If the coefficients of the combination of loadings are generated in such a way so that the result corresponds to the action of characteristic loads, it is necessary to check the **Characteristic combination** checkbox to obtain the correct solution. Otherwise the transition to the characteristic values will be performed automatically (by dividing by the respective safety factor for load).

If the group of loads characterizing the weight of structures which collapsed in the result of the sudden removal of elements is prepared in advance, it will be taken into account if the **Take the weight of the collapsed structures from the group** checkbox is checked.

The progressive collapse analysis can be invoked from the respective section of the **ANALYSIS** group of the project tree. If the **Perform the analysis accounting for geometric nonlinearity** checkbox is checked in the **Progressive Collapse** dialog box, the specified type of analysis will be automatically performed (in this case, it is necessary to include elements taking into account geometric non-linearity in the design scheme).

In the case of the progressive collapse analysis **SCAD** creates a special file with a model which does not include any removed elements, includes the respective applied forces, ... This file is deleted by default after the calculation, but the **Save progressive collapse model file** checkbox enables to save this file, and, if necessary, open it, perform the calculation and analyze, for example, the deformed model. This file is created in the directory of the main file, and its name has an additional suffix [Destr]. The *Dynamic Reaction* group of loads is also created in this file. It contains the loads corresponding to the reactions of the removed structural elements taking into account the suddenness of removal by the user-defined dynamic amplification factor.

Peculiarities of Using the Results of the Reinforcement Selection

As mentioned above the progressive collapse analysis of the elements of reinforced concrete structures is performed by checking them for the ultimate limit state (strength). Thus, the reinforcement must be specified for the structural members before this check.

The reinforcement layout (arrangement of the reinforcement in different segments of the bar) for each structural group of bar elements can be either completely specified by the user or completely taken in accordance with the results of the automatic selection. The **Specify reinforcement based on the results of the reinforcement selection** operation in the **Selection of reinforcement** section of the **Reinforced Concrete** toolbar is used to transform the results of the automatic selection. The reinforcement layout can be specified by the user with the help of the respective operation in the **Check of reinforced concrete** section of the **Reinforced Concrete** toolbar or the **Assign** toolbar of the preprocessor.

The work with plate elements is organized in a similar way. However, unlike bars in this case you can specify the rules for transforming the selected reinforcement into the specified one (redistribution of the reinforcement in the group) when performing the **Specify reinforcement based on the results of the reinforcement selection** operation.

Only the specified reinforcement is considered in the mode of the progressive collapse analysis.

Data on the reinforcement of each finite element used in the progressive collapse analysis are available in the **Specified reinforcement** section of the **Information on the Element** dialog box.

23.2. Calculation

The following procedure for the calculation is used in the software:

- the reactions from the control combination of loads are determined in the nodes of the failed elements adjacent to the rest of the model;
- the obtained values of the reactions are added in the design combination with the coefficient K_f;
- a group of loads from the weight of the collapsed structures is added in the control combination with the coefficient K_g (if such a group has been specified);
- a new design model with inactive failed elements is generated;
- the obtained model is calculated for the control combination;
- design combinations of forces are generated;
- the check of the load-bearing capacity of the elements of steel and reinforced concrete structures is performed.

A special importance factor γ_a is used when checking the steel and reinforced concrete elements. If GOST 27751, GOST R 54257-2010, or GOST 27751-2014 is selected as the safety code, then γ_a=1,0. If DBN B.1.2-14-2009 is selected, the importance factor is specified by the user for each structural member or group of structural members.

23.3. Analysis of Results

The results of the progressive collapse analysis are displayed in the graphical form in a three-color color scale. The **Progressive collapse** section of the **Postprocessors** toolbar contains the controls of the display.

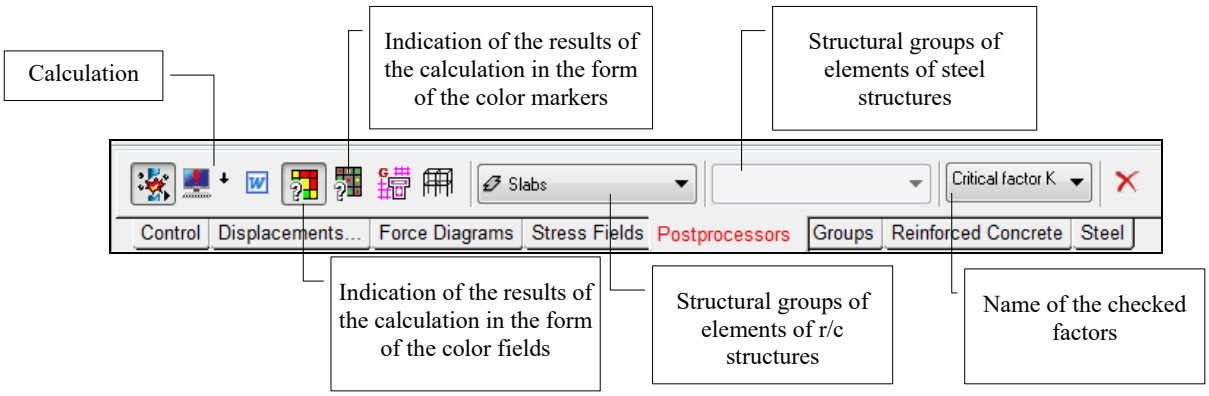


Figure 23.3-1. The **Progressive collapse** section of the toolbar

The operation enables to select the elements in the model which are listed as deleted in the initial data. These elements will be "highlighted" in the design model in purple.

In the three-color scale the elements are divided by color into working elements ($K_{max} < 1$) with the maximum utilization factor of restrictions K_{max} less than one (green color is used for them by default), and the failed ones ($K_{max} \geq 1$) — red color is used for them. The third color (yellow by default) is used to indicate elements in the uncertainty interval, i.e. elements which in the opinion of the engineer can be failed or working with equal probability. The value of the interval of uncertainty (in % from K_{max}) is specified by the user with the help of the **Color scale settings** operation in the scale dialog box. The results can be output in the form of color fields or color markers (buttons and respectively). The button allows you to get extreme values of factors (similar to how it is done in the analysis of elements of steel and reinforced concrete structures).

If the color indication of the results of the calculation (values of the utilization factor of restrictions K_{max}) is performed without indicating a specific structural group, then a color corresponding to the value of the maximal factor is selected for each finite element without indicating a factor which defined this value.

In those cases when a certain group is selected from the list of structural groups, a color scale can be obtained for it not only for the values of K_{max}, but also for the utilization factors of restrictions of any of the available factors. Factors can be selected from the respective list in the toolbar (Fig. 23.3-1).

It should be noted that the detected failed elements are elements which failed on the very first step of the progressive collapse process. If you include them in the list of finite elements included in the suddenly removed structural member and determine where the load is transferred from these elements after they fail, you can obtain the failure pattern on the second step etc. However, more often it will be necessary to strengthen the elements (maybe not all) included in the members which failed on the first step, and repeat the calculation for the strengthened structure. The strengthened elements should be combined into the respective groups of reinforcement or groups of steel elements.

After performing the calculation for a certain finite element you can use the **Information on the Element** dialog box which is invoked by clicking the filters button . The **DCF (of the progressive collapse)** and **Progressive collapse** buttons in this dialog box enable to obtain the data on the design combinations of forces which were used in the calculation (Fig. 23.3-2) and factors calculated when checking the element (Fig. 23.3-3). The **Show forces** checkbox enables to get an alternative presentation of information showing forces/stresses in an element (Fig. 20.19-2*b*). Moreover, similar to the analysis of elements of steel and reinforced concrete structures, operation **Resistance sections (progressive collapse)** is implemented.

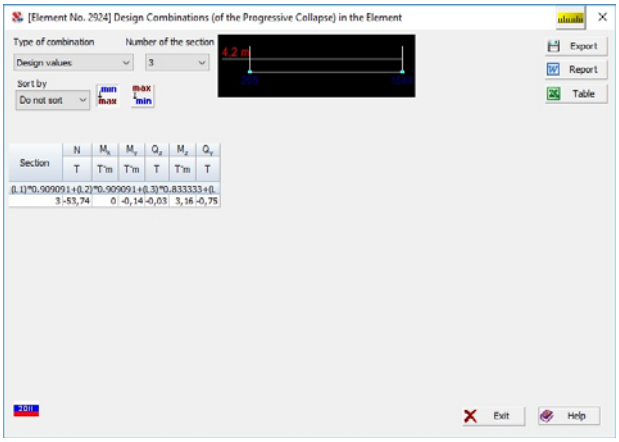


Figure 23.3-2. *The* **Design Combinations (of the Progressive Collapse) in the Element** *dialog box*

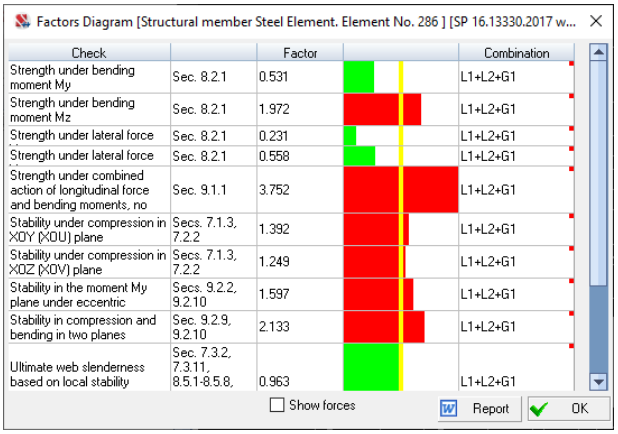


Figure 23.3-3. *The* **Factors Diagram** *dialog box*

References to Chapter 23

1. Recommendations on the protection of residential frame buildings during emergencies/ Moskomarkhitektura. – M.: GUP NIAC, 2002.– 24 p.
2. SP 296.1325800.2017 Buildings and structures. Special actions. — M.: 2017. — 27 p.
3. SP 385.1325800.2018 Protection of buildings and structures against progressive collapse. Design code. Basic statements

24. Stability Analysis

The following values are determined in this postprocessor for each selected loading (or combination of loadings):

- stability factors of safety;
- buckling modes;
- unsupported lengths of bar elements.

24.1. Problem Formulation

The stability problem is solved in the classical formulation for the elastic system (see Sec. 2.5) and assuming that all the external loads applied to the system (hence the internal ones as well) increase in proportion to the same parameter λ. The value of the parameter λ, at which the stiffness matrix of the system $K(\lambda)$ stops being positive is called critical, and the corresponding value λ is called *the stability factor of safety* (SFS). Stiffness matrix $K(\lambda) = K_0 - K_1(\lambda)$ consists from an "ordinary" stiffness matrix K_0 and a matrix of "pushing" reactions $K_1(\lambda)$, which are defined by compressive forces in bars, compressive stresses in shell finite elements etc. (see Sec. 2.5.7). It should be noted that if the stiffness matrix is positive definite, it means that the elastic potential is positive regardless of the values of nodal displacements and rotations (it means that it is necessary to expend energy for deforming the system and, consequently, it resists the deformation).

If the system loses stability, its stiffness matrix becomes degenerate (with a zero determinant), and if the system is in the supercritical state (the accumulated potential energy of "pushing" reactions is released at its forced deformation), its stiffness matrix becomes indefinite.

Thus, the problem of evaluating the stability of equilibrium comes down to checking the positive definiteness of the stiffness matrix at the trial value of the factor λ.

It should be noted that only the critical states at which the so-called buckling mode occurs, when not all nodal displacements and rotations are equal to zero can be found by performing the checks of the stiffness matrix. It is also checked that a hidden buckling mode, which takes place within one finite element and does not cause nodal displacements and rotations, does not occur at the trial value λ. Since the respective critical values λ_{cr} are known for all types of finite elements (they are calculated by simple formulas), it means, that it is also necessary to check the inequality $\lambda > \lambda_{cr}$ for all finite elements.

If necessary, the specified number of factors λ and their respective buckling modes can be calculated. It is especially important, for example, if there are multiple modes.

It should be noted, that the whole system loses stability, i.e. all its elements. In order to illustrate this fundamental principle, let's imagine a compressed bar the design model of which is made up from a chain of finite elements of the "space bar" type. If this bar is compressed by the force P, it may lose stability at its certain value k times greater than the specified one (i.e. k — stability factor of safety). The question arises which of the finite elements the longitudinal force in which is equal to $kN = kP$ at the moment of buckling has lost stability? Naturally, all the elements that make up the bar. This conclusion applies to any design model. Although buckling characterizes the entire system, it does not mean that all its elements are equally responsible for the occurrence of the critical state. Their specific role can be estimated by analyzing the energy pattern of the phenomenon (see Chapter 25).

Classical theory of the stability of equilibrium considered in most courses of structural mechanics and usually used in the structural analysis assumes that all internal forces increase in proportion to one parameter, and the ratio between them remains constant. However, even for linear systems the stability factor of safety obtained by the classical method might not have a clear physical meaning. Indeed, imagine a structure with elements highly compressed by the constant load G_0 from the self-weight, and in addition loaded by the temporary load P_0 (Fig. 24.1-1).

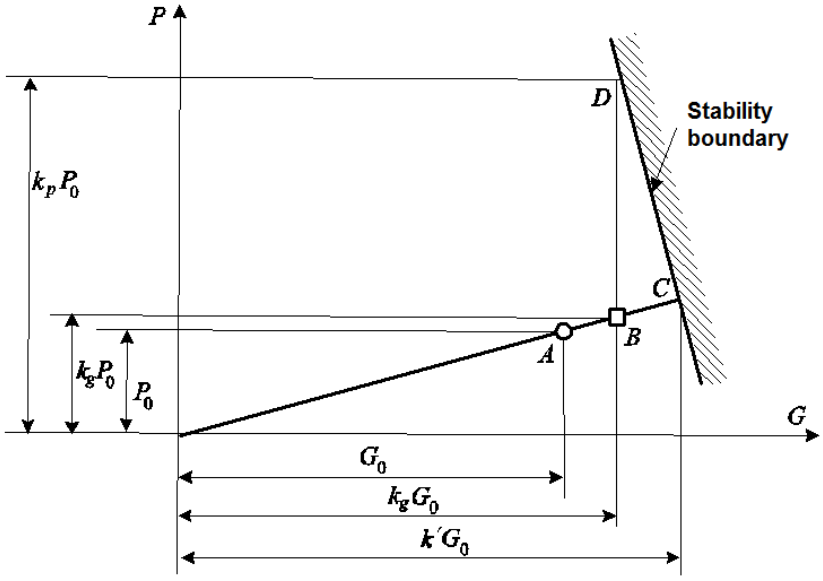


Figure 24.1-1. *Disproportionate growth of loads*

Factor of safety $k' = 1{,}25$ for the total load corresponds to the clearly unrealistic growth of the self-weight by 25%. If we consider the growth of the self-weight, for example, by 10% (i.e. take $k_g = 1{,}1$), the temporary load has to increase much more to achieve the critical state. Naturally, under such reasoning, a graphical illustration of which is given in Fig. 24.1-1, a rather modest factor of safety of 1,25 appears in a completely different light ($k_p > 1{,}25$). The result will obviously greatly depend on the type of the stability boundary, and if it has another configuration, all the factors of safety can turn out to be such that the value $k_p P_0$ will be much lower. It is important to note the insufficient accuracy of the analysis of the system with the usual interpretation of the stability factor of safety.

Therefore, the variant when two loadings act on the system only one of which changes is implemented (see Sec. 2.5).

24.2. Search of the Stability Factor of Safety

The search of the stability factors of safety is performed in the range $[0,\Lambda]$, where Λ — the specified number corresponding to the SFS value, which is considered to be not important for the assessment of the quality of the system, and with the specified accuracy ε. The problem of determining the minimal value of λ, and then the next larger one, at which the degeneration of the matrix $K(\lambda)$ takes place is solved.

Matrix $K(\lambda)$ is made up of the stability matrices of individual finite elements. If the system has no elements that can buckle (for example, all bars are in tension in the bar system), a message is generated that the system is "*absolutely stable*".

Then the stability of the system is checked at $\lambda = \Lambda$ (i. e. positive definiteness of matrix $K(\Lambda)$). If this condition is satisfied, a message that the SFS is greater than the specified maximum value is generated.

If the condition of the positive definiteness of $K(\Lambda)$ is not satisfied (this is evidenced by the negative values on the main diagonal of the stiffness matrix transformed in the process of solving the system of equations), the analysis of the positive definiteness of the matrix $K(\Lambda/2)$,... is performed, i. e. a standard bisection method is used. This process continues until such an interval (λ_1, λ_2) is determined so that $\lambda_2 - \lambda_1 \le \varepsilon$ (ε — the accuracy of the determination of the SFS specified by the user) and the matrix $K(\lambda_1)$ is positive definite, while the matrix $K(\lambda_2)$ does not have this property. The value λ_1 is considered to be the SFS.

When generating a stability matrix for each finite element (that can buckle) on this step the value of λ_{cr} which leads to the loss of stability of the element is calculated. If $\min \lambda_{cr} < \Lambda$, the search interval is reduced, and the number of the element for which the minimum value of λ_{cr} is achieved is output in the log.

24.3. Buckling Modes

Assuming that the stability factors of safety determined on the first stage are accurate, the problem of the determination of the eigenvector at the known *i*-th eigenvalue of the problem is solved (see Sec. 2.5.7)

$$[K_0 - K_i(\lambda_i)]\ \mathbf{u} = 0. \qquad (24.3.1)$$

It should be noted that the right-hand sides of the system are equal to zero, i.e. the values of nodal displacements and rotations **u** caused only by the internal compressive stresses and forces are determined. As you know, the transverse loads have no effect on the values of critical forces and buckling modes. Since the system of equations (24.3.1) is solved at the zero right-hand side, the buckling mode is determined only up to the factor.

Knowing the buckling mode, you can use the energy postprocessor (see Sec. 25) and determine the role of individual parts of the system in the process of buckling. Use the **Elements with nonpositive energy** button to do it. It must be emphasized that the moment of buckling is considered, i. e. the result of the analysis does not depend on the value of the stability factor of safety. Thus, if you increase the intensity (but not the type) of loading by increasing all loads, for example, N times, the factor of safety will decrease N times, while the buckling mode and the energy pattern will remain the same.

24.4. Unsupported Lengths

If the system has bar elements, it is possible to determine their unsupported lengths, i.e. lengths of the same but hinged bars the critical force N_{cr} of which coincides with the longitudinal force in the bar of the system at the moment of buckling. Since for the simply supported bar of the length *l* according to the Euler's formula

$$N_{cr} = \pi^2 EJ / l^2,$$

the unsupported length factor (ratio of the unsupported length to the length of the bar) will be equal to

$$\mu = [(\pi^2 EJ/(N_{cr}\, l^2)]^{1/2}, \qquad (24.4.1)$$

where EJ — rigidities of bars in the principal planes of inertia (for a spatial problem — two for each bar).

We can illustrate it with an example of a variable cross-section bar (Fig. 24.4-1), compressed by forces F = 5T and F = 10T. The calculation of this system gives the value k = 24,38.

The unsupported length of the first element L_1 = 25,45 m, and of the second one — L_2 = 12,72 m. Thus, the unsupported length factor of the first element μ_1 = 25,45/12 = 2,12, and of the second one — μ_2 = 12,72/2 = 6,36. Calculation according to SNiP II-23-81*[1] (see Annex 6, Table 67) gives values

$$n = (2000\times12)/(24000\times2) = 0{,}5;\ \beta = (10 + 5)/5 = 3;\ \alpha_1 = (2/12)\times[24000/(2000\times3)]^{1/2} = 0{,}333.$$

If we interpolate the values from the table 67, we will obtain μ_2 = 2,136 and μ_1 = 2,136/0.333 = 6,236.

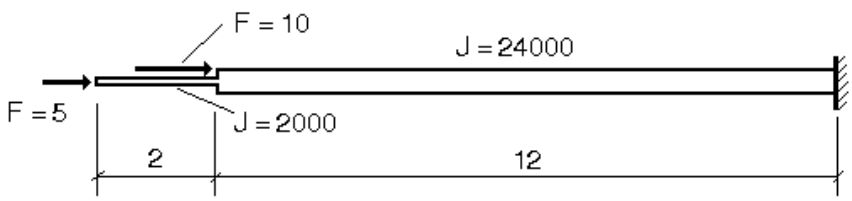


Figure 24.4-1.

It can be seen from the formula (24.4.1) that if the system has heavily and lightly loaded bars with the same section and the same boundary conditions (for example, columns of a multi-storey building, where the compression on the upper floors is less than on the lower ones), the lightly loaded bars have significantly greater unsupported lengths. This fact has long been well known and was a subject of lively discussions many times. It was found in these debates that the strength and stability checks of such "too

[1] Hereinafter in this chapter we refer to SNiP II-23-81*, but it equally applies to SP 16.13330.2011 and DBN B.2.6-163:2010 as well.

long" bars are performed quite correctly, but the check of the limit slenderness should not be performed with such lengths. Unfortunately, this conclusion is not given in the codes.

It should be noted that when a real bar is divided into finite elements, the unsupported length of each of these finite elements is output in the result of the calculation. Thus, in the above example, if we divide the first element, for example, into four equal parts, the unsupported length of each of them will be equal to 25,45 m in the result of the calculation, and the checks of any of these parts would be performed under the same conditions.

24.5. Unsupported Lengths and Effective Length Factors

It should be noted that the use of the unsupported lengths calculated in **SCAD** does not allow to **automatically** specify the effective lengths (or the effective length factors) in the subsystems of the calculation of steel and/or reinforced concrete structures (see Chapters 19 and 20):

1. The unsupported length of the bar element depends on the loading. Design combinations of forces are used in the calculations of the elements of steel and reinforced concrete structures. These combinations are basically combinations of several loadings and this combination can be different in different sections of the same element. Naturally, a problem arises — which unsupported length (from which loading or which combination of loadings) should be used.
2. A concept of a *structural member* — a sequence of finite elements, is used in the **SCAD** subsystem of the calculation of elements of steel structures. Unsupported lengths of each finite element (not the structural member) are calculated in the stability analysis.
3. It is necessary to know the effective length in order to determine the slenderness according to SNiP II-23-81*. If the element is in tension, the concept of the unsupported length loses its meaning (and it is not calculated). And the effective length factor is necessary to check the slenderness limitations.
4. As far as we know the concept of the unsupported length introduced in SNiP is not the classic definition from structural mechanics. The developers of SNiP assumed that the structure is rigid in one plane, and believed that the effective length characterizes the stability of the bar under this condition. The effective lengths are determined in two planes in **SCAD** assuming that the rigidities in both planes are finite.

24.6. Design Codes and Stability Analysis of Bars

The moment of buckling is determined in **SCAD** for a **strictly rectilinear, perfectly elastic** bar. It might turn out that a certain bar is overstressed and has already lost its stability at, for example, 80% of the current load.

Formulas given in the codes allow for the **elastoplastic behavior of a bar with initial imperfections** (this is the way the buckling coefficient curve $\varphi(\bar{\lambda})$ given in them is plotted). It might turn out that the same bar is also overstressed and has lost its bearing capacity at, for example, 76% of the current load. In this case 76% is not very different from 80%. But in other cases the difference can be very large. An exemplary plot shown in Fig. 24.6-1 can be used to explain this phenomenon. Plots of this type can be found in textbooks and guides on steel structures (see, for example, [1, p. 324, Fig. 6.13] or [2, p. 209-210]).

It can be seen from the plots given in the figure, that if the slenderness is small the values of φ differ significantly. This difference can be seen by comparing the results obtained with the help of **Kristall**, which implements the lower curve of Fig. 24.6-1 (in accordance with the recommendations of the codes), and **SCAD** where the classical Euler solution is implemented, which is given by formula (24.4.1) when there are hinges at the ends of the bar.

As an example, let's consider a design model made up from four bar finite elements of ring section. The critical load of such a compressed column turns out to be 44% greater than it should be, for example, from formula 7 of SNiP II-23-81*, since the buckling coefficient φ is given in SNiP allowing for the additional margin due to the possible imperfections of the compressed bar. To verify this, consider, for example, a bar with the conventional slenderness $\bar{\lambda} > 4{,}5$, for which formula 10 is given in SNiP II-23-81*: $\varphi = 332/[\bar{\lambda}^2(51-\bar{\lambda})]$. Let's compare this result with the Euler solution: $\sigma_{cr} = \pi^2 R_y/\bar{\lambda}^2$. Determine the factor of safety equal to $\sigma_{cr}/(\varphi R_y)$. It varies from 1,38 at $\bar{\lambda} = 4{,}5$ to 1,10 at $\bar{\lambda} = 14$.

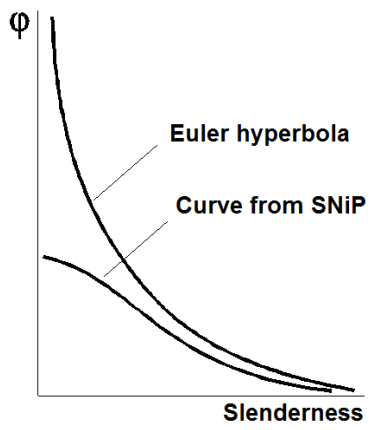


Figure 24.6-1. *Relationship* φ(λ)

It should also be noted that if the slenderness is small, one more difference comes into play — the formulas given in the codes take into account the plastic stage of the material at buckling, while the software provides the solution of the stability problem of an elastic system.

SCAD does not take plasticity into account for the following reasons:

- firstly, it is not known what to do for the elements from the materials different from those given in the code;
- secondly, it is not known whether the user is interested in, for example, SNiP II-23-81* "Steel Structures" or SNiP 2.05.03-84 "Bridges and Pipes". Curves φ(λ) in the last document are different even for steel structures, because the assumptions on the value of the initial imperfections are different.

If we remember that there are reinforced concrete, timber, plastic and other kinds of structures, accounting for all the details dealing with their check according to the codes distracts from the *general* design software.

24.7. Input Data

The data for the check of the overall stability of the system is input in the dialog box (Fig. 24.7-1), which can be invoked from the **Special Initial Data** section of the **Project tree**. It is necessary to perform the following operations to input the data:

- specify the necessary check modes — calculation of the stability factor of safety, buckling modes and unsupported lengths of bar elements, with the help of the respective checkboxes;
- specify the respective values in the **Upper search limit** and **Accuracy of calculations** fields (they are equal to 2 and 0,01 by default, and they can be adjusted when specifying the calculation parameters (see Chapter 11);

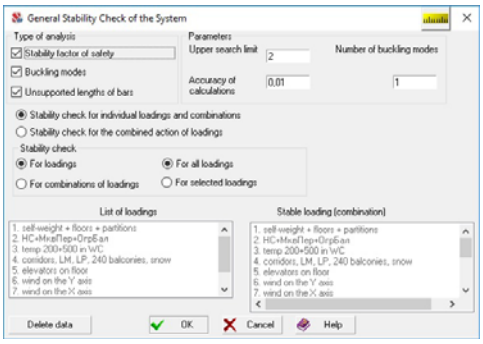


Figure 24.7-1. *The* **General Stability Check of the System** *dialog box*

- enable the options which determine the type of data for which the stability check is performed: for loadings or for combinations of loadings, for all loadings (combinations) or only for the selected ones; in the last case the necessary data are selected in the list of loadings (combinations);
- if necessary you can select the **Stability check for the combined action of loadings** radio button, the **Stable loading (combination)** list will become available where you can select the necessary data.

Accuracy of calculations — defines the criterion of the end of the iterative process of searching the stability factor of safety. If the values of this parameter are very small, the calculation time can increase significantly. If the buckling moment is determined inaccurately, the condition that the elastic potential is equal to zero can be performed with an unacceptable error. Therefore, in order to analyze the buckling modes in the energy processor, it is necessary to increase the accuracy of the calculations.

Upper search limit — is used to limit the search interval of the stability factor of safety. If its value is greater that the upper limit, the system is considered to be stable and the analysis log (see Chapter 13) will contain the following message:

```
Stability factor of safety of the system is greater than the upper search
limit. Its value is not determined.
```

In this case formula (24.4.1) can not be used, the unsupported lengths will not be calculated, and the following message will be generated:

```
Unsupported lengths cannot be calculated for loadings NNN because the
stability factor of safety of the system is not calculated for them.
```

The **Number of buckling modes** parameter enables to obtain several buckling modes and to study the behavior of the structure in more detail, and in some cases to use it when making design decisions (see Sec. 9.2 of [3]).

In order to obtain the values of the unsupported lengths, and the buckling modes which can not be determined if the value of λ_{cr} is not calculated, it is necessary to specify a larger value of the upper search limit and repeat the calculation.

Check the respective checkboxes in the **Stability** tab of the **Calculation Parameters** dialog box (see Chapter 11) if you want to allow for tangential displacements in the geometric stiffness matrix of shells and for possible buckling at the bending of bars.

The **Delete data** button enables to delete the prepared data and cancel the stability analysis.

24.8. Peculiarities of the Analysis for the Dynamic Loadings

The stability check for the combinations of loadings which include the dynamic ones should be performed with caution due to the following facts:

- the results of the dynamic loading are alternating;
- in those cases when the results of the dynamic analysis are generated as a root of the sum of squares (RSS) of the modal contributions, the conditions of equilibrium and the geometric consistency of the nodal displacements are violated (see, for example, Sec. 13.4).

Let's consider them separately.

Since it is not known in advance which sign of the result of the dynamic analysis is more dangerous according to the stability analysis, both possibilities should be considered. For example, if the combination (L1)*1+(L2)*0.9+(L3)*0.7+(L4)*0.8, where the loading (L3) is a dynamic one, is considered in the stability analysis, it is also necessary to perform the check for the combination (L1)*1+(L2)*0.9+(L3)*(-0.7)+(L4)*0.8. If there are several dynamic loadings, it is necessary to try all the options of the signs for their results.

Actually the inconsistency of forces related to the use of the RSS (obviously, if not just one natural oscillation mode is taken into account) does not cause any problems. You just have to check the stability factor of safety at the proportional growth of all the internal forces ignoring the fact that these forces disturb the equilibrium conditions.

However the inconsistency of nodal displacements leads to the distortions of the rigid bodies (in fact, they deform, which contradicts their physical nature). The difficulty lies in the fact that instead of being defined by the internal forces or stresses the matrix $K_l(\lambda)$ for these elements is calculated on the basis of the values of the nodal displacements, which define the variation of the spatial orientation of an infinitely rigid finite element (see 9.5.1 in [3]). The inconsistency of the nodal displacements can lead to a fatal error in the calculation $K_l(\lambda)$.

Therefore the software enables to reduce the matrix $K_l(\lambda)$ when checking the stability for combinations containing a dynamic loading with more than one natural oscillation mode taken into account. A respective warning appears in the analysis log.

24.9. Analysis of Results

Buckling modes can be analyzed at the graphical analysis of the results in the **Displacements** section (see Fig. 12.1-5), where they are identified by the letter **S** in the list of loadings. All results can be obtained with the help of the **Table generator in the text format** or **Documenter** (see Sec. 13.17).

References to Chapter 24

1. V.V. Gorev, B.Yu. Uvarov, V.V. Filippov and others, Steel structures. In 3 v. — V. 1. Elements of steel structures. — M.: High school, 1997.— 527 p.
2. V.V. Kuznetsov, Steel structures. In 3 v. — V. 1. General. (Designer's handbook) — M.: ASV, 1998.— 576 p.
3. A.V. Perelmuter, V.I. Slivker, Design models of structures and possibilities of their analysis. — M, DMK-Press, 2007. — 600 p.

25. Energy Postprocessor

The energy postprocessor can be considered as a tool for obtaining the display of the peculiarities of the stress-strain state of the system in the form of the strain energy distribution pattern.

If the energy pattern is generated taking into account the effect of the geometric stiffness matrix, it becomes possible to classify the individual parts of the system (up to its individual elements) into the following classes: the class of *restraining* and the class of *pushing* elements (or parts) of the system. Restraining elements help to preserve the stability of equilibrium of the system, while pushing elements contribute to its buckling.

The check of the role of individual subsystems is performed during the calculation of the value of energy accumulated in different parts of the system at its deformation according to the buckling mode. This energy is equal to zero for the whole system, and its parts where the energy is non-positive are classified as pushing, while the parts of the system with positive strain energy can be classified as restraining ones.

On the basis of the numerical values of the energy, pushing elements of the system can be ranked according to their degree of responsibility for the critical state of the system, since the contribution of each element of the system in its total energy balance can serve as a convenient quantitative measure of its responsibility for the stability of equilibrium .

Figure 25-1. *The* **Energy postprocessor** *section of the* **Postprocessors** *toolbar*

The mode of the analysis of the results of the energy postprocessor is invoked by clicking the button in the **Postprocessors** toolbar.

The toolbar of this mode contains the following buttons:

— display the distribution of energy as color fields;

— display the distribution of energy as color markers;

— display the distribution of specific energy as color fields;

— display the distribution of specific energy as color markers;

— display elements with positive, zero and negative values of energy;

— display the distribution of energy characteristic as color fields;

— display the distribution of sensitivity coefficients as color fields;

— display the design model.

The results of the energy processor are displayed as color fields or markers. The distribution of the energy in the system can be obtained as reduced to the volume of the finite element or to the unit volume (specific energy).

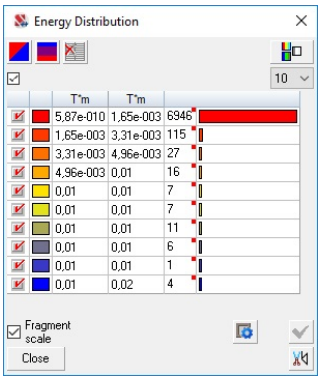


Figure 25-2. *The* **Energy Distribution** *color scale*

In the case when the color scale of the distribution of the positive and negative energy is generated for the selected buckling mode on the fragment of the design model and the **Fragment scale** checkbox is checked, a message on the sign of the total energy of the given fragment is generated.

It enables to make a conclusion on whether the subsystem (selected fragment) is *restraining*, *pushing* or *neutral*[1].

Energy indicators of the system make sense not only at the stability analysis, but for the static loadings and their combinations as well. In this case the distribution of the specific energy characterizes the energy "loading" of the elements of the design model, and the distribution of the energy reduced to the whole volume of the FE can indirectly serve as an indicator of the quality of the finite element mesh.

You can also display information on the distribution of the energy characteristic:

$$\eta = 1 - \lambda \frac{\langle L_g v, v \rangle}{\langle L v, v \rangle},$$

where L_g is the geometrical stiffness matrix of the element, L is the stiffness matrix of the element, λ is the stability factor of safety, $\boldsymbol{v}$ are "displacements" of a finite element obtained when solving the stability problem (see Sec. 16.4.2 of *Stability of Structural Equilibrium and Related Problems* by A.V.Perelmuter and V.I.Slivker, vol. 2 — World Scientific, 2013).

Moreover, the energy postprocessor enables to obtain a color scale of the sensitivity factors, which are used to assess the role of the system parameters (design variables) when their changes are relatively small, i.e. partial derivatives of a certain result with respect to the design variable. In this case, we consider the sensitivity factors of the critical parameter (stability factor of safety) with respect to the stiffness of the finite elements.

[1] See Perelmuter A.V., Slivker V.I. Design models of structures and possibilities of their analysis. — M., SCAD SOFT, ASV, DMK-Press, 2011. — 709 p.

26. Principal and Equivalent Stresses

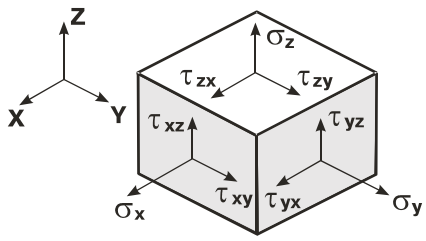


Figure 26-1. *Stress components*

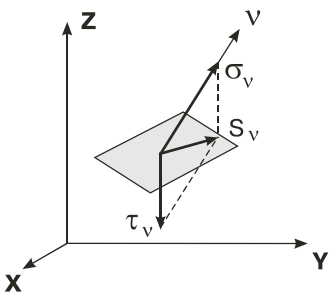


Figure 26-2. *Stresses on an arbitrary plane*

Let's outline the fundamental principles of the stress theory usually considered in the course of the theory of elasticity or the strength of materials.

If you select an elementary volume in the form of an infinitesimally small parallelepiped from the body in the vicinity of a certain point (Fig. 26-1), the environmental action on it can be replaced by the stresses the components of which act on the faces of the parallelepiped.

According to the *law of pairing* of shear stresses

$$\tau_{xy} = \tau_{yx}\,,\ \tau_{yz} = \tau_{zy}\,,\ \tau_{zx} = \tau_{xz}\,. \tag{26.1}$$

In the general case there are only six independent stress components in a point which form a symmetric *stress tensor*

$$T_\sigma = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}. \tag{26.2}$$

Normal stress σ_ν and shear stress τ_ν (Fig. 26-2) with the resultant S_ν act on an arbitrarily oriented plane passing through the same point the normal to which ν has direction cosines *l, m, n* with the axes x, y, z. The projections of this resultant on the coordinate axes $S_{\nu x}$, $S_{\nu y}$, $S_{\nu z}$ are related to the stress components by the equilibrium conditions (Cauchy's formula):

$$\begin{aligned} S_{\nu x} &= \sigma_x l + \tau_{xy} m + \tau_{xz} n \\ S_{\nu y} &= \tau_{xy} l + \sigma_y m + \tau_{yz} n\,. \\ S_{\nu z} &= \tau_{xz} l + \tau_{yz} m + \sigma_z n \end{aligned} \tag{26.3}$$

There are such three mutually perpendicular planes on which there are no shear stresses. *Principal stresses* σ_1, σ_2 and σ_3 ($\sigma_1 \geq \sigma_2 \geq \sigma_3$) act on these so-called *principal planes*. It is also known that the principal stresses have extreme properties, i.e. the resulting stress on any plane $S_\nu \leq \sigma_1$ and $S_\nu \geq \sigma_3$.

The direction cosines l_k , m_k and n_k of the normals to the principal planes ν_k are determined by solving the system of equations:

$$\begin{cases} (\sigma_x - \sigma_k) l_k + \tau_{xy} m_k + \tau_{xz} n_k = 0; \\ \tau_{xy} l_k + (\sigma_y - \sigma_k) m_k + \tau_{yz} n_k = 0; \\ \tau_{xz} l_k + \tau_{yz} m_k + (\sigma_z - \sigma_k) n_k = 0; \end{cases} \tag{26.4}$$

a nonzero solution of which exists when its determinant is equal to zero.
And $l_k^2 + m_k^2 + n_k^2 = 1$.

It follows from (26.4) that the principal stresses σ_k (k=1, 2, 3) are the roots of the cubic equation

$$\det \begin{bmatrix} \sigma_x - \sigma & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y - \sigma & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z - \sigma \end{bmatrix} = 0\,. \tag{26.5}$$

Expanded form of the equation (26.5):

$$\sigma^3 - I_1(T_\sigma)\sigma^2 - I_2(T_\sigma)\sigma - I_3(T_\sigma) = 0\,, \tag{26.6}$$

Its coefficients are invariants (i.e. they do not depend on the selected coordinate system). The first invariant $I_1(T_\sigma) = \sigma_x + \sigma_y + \sigma_z$ is equal to three times the average stress (hydrostatic pressure) σ_0.

The direction of the principal planes can be defined not only by the nine direction cosines but also by the three Euler angles (precession angle ψ, nutation angle θ and the pure rotation angle φ). Any plane parallel to the coordinate plane (XOY, XOZ or YOZ) can be set to any position with their help.

The Lode-Nadai coefficient is used to characterize the stress-strain state (SSS).

$$\mu_0 = 2\frac{\sigma_2 - \sigma_3}{\sigma_1 - \sigma_3} - 1$$

It takes values $\mu_0 = 1$ at pure compression, $\mu_0 = 0$ at pure shear, $\mu_0 = -1$ at pure tension.

When outputting the results of the calculation the stress tensor (26.2) in the general case has the following form

$$T_\sigma = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}. \tag{26.7}$$

26.1. Principal Stresses for Different Types of Finite Elements

Each type of element has certain peculiarities of the stress-strain state (SSS), which in turn defines the peculiarities of the position of the principal planes.

Depending on the considered type of element principal stresses and angles which characterize the position of the principal planes are calculated in each point where the forces (stresses) are determined. The results of the calculation can be obtained only in the center of gravity of the element (center of gravity of the cross-section of the body of revolution for axisymmetric elements) or in the center of gravity and in nodes.

Spatial Problem of the Theory of Elasticity

Solid elements are used for solving the spatial problem of the theory of elasticity, and axisymmetric elements as a special case. The following values are calculated for them:

- principal stresses σ_1, σ_2 and σ_3;
- Euler angles —θ, ψ and φ;
- Lode-Nadai coefficient μ_0;
- angle of the principal stress σ_1 to the X_1 axis.

Deep Beam Elements

The stress tensor in the case of the plane SSS (deep beam) has the following form

$$T_\sigma = \begin{bmatrix} \sigma_x & 0 & \tau_{xz} \\ 0 & 0 & 0 \\ \tau_{xz} & 0 & \sigma_z \end{bmatrix}, \tag{26.1.1}$$

where σ_x, σ_z, τ_{xz} — stress components in the midplane of the element in the local coordinates X_1OZ_1. In this case only two principal stresses can be calculated by the formula

$$\sigma_{1,3} = \frac{\sigma_x + \sigma_z}{2} \pm \sqrt{\left(\frac{\sigma_x - \sigma_z}{2}\right)^2 + \tau_{xz}^2}\,. \tag{26.1.2}$$

The position of the principal planes is characterized by the angle of the principal stress σ_1 to the X_1 axis

$$\varphi = arctg\frac{\sigma_1 - \sigma_x}{\tau_{xz}}\,. \tag{26.1.3}$$

If $\tau_{xz} = 0$, it is considered that $\varphi = 0$, and in this case the directions of the principal planes coincide with the axes of the coordinate system of the output of stresses in the element.

Plates and Shells

The following forces on the midplane are calculated for plates:

- moments — M_x, M_y and M_{xy};
- shear forces — Q_x and Q_y.

The following stresses are also calculated for shells: σ_x , σ_y and τ_{xy}. Stress tensor has the following form:

$$T_\sigma = \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{xy} & \sigma_y & 0 \\ 0 & 0 & 0 \end{bmatrix}, \tag{26.1.4}$$

because shear stresses $\tau_{xz} = 1/5 Q_x / h$, $\tau_{yz} = 1/5 Q_y / h$ are not taken into account.

Principal stresses are determined on the lower (L), middle (M) and upper (U) surfaces for each point where the forces are calculated. And

$$\begin{aligned} \sigma_x^{U/L} &= \sigma_x \pm 6M_x/h^2, \\ \sigma_y^{U/L} &= \sigma_y \pm 6M_y/h^2, \\ \tau_{xy}^{U/L} &= \tau_{xy} \pm 6M_{xy}/h^2, \end{aligned} \tag{26.1.5}$$

where h — thickness of the element.

Then the principal planes for the upper and lower surfaces are parallel, and the principal stresses are determined by the formula

$$\sigma_{1,2} = \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2} \ . \tag{26.1.6}$$

The position of the principal planes is characterized by the angle of the principal stress σ_1 to the X_1 axis

$$\varphi = \mathrm{arctg}\frac{\sigma_1 - \sigma_x}{\tau_{xy}} \ . \tag{26.1.7}$$

If $\tau_{xy} = 0$, it is considered that $\varphi = 0$, and in this case the directions of the principal planes coincide with the axes of the coordinate system of the output of stresses in the element.

Bar Elements

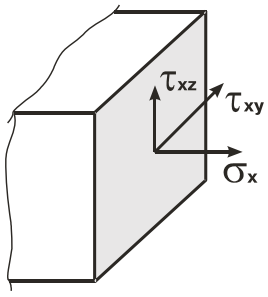


Figure 26.1-1

Principal stresses in bar elements are determined by the formula

$$\sigma_{1,2} = \frac{\sigma_x}{2} \pm \sqrt{\left(\frac{\sigma_x}{2}\right)^2 + \tau_{xy}^2 + \tau_{xz}^2} \ . \tag{26.1.8}$$

Here σ_x, τ_{xy} and τ_{xz} are normal and shear stresses respectively in the characteristic points of the bar cross-section (Fig. 26.1-1).

Principal stresses are calculated for bar elements the sections of which are defined parametrically or selected from the assortment of rolled profiles. Fig. 26.1-2 and Fig. 26.1-3 show the characteristic points of the sections for which the calculations are performed.

Values of the principal stresses are not calculated in the points which are not located on the material part of the cross-section (for example, point 9 for the hollow section).

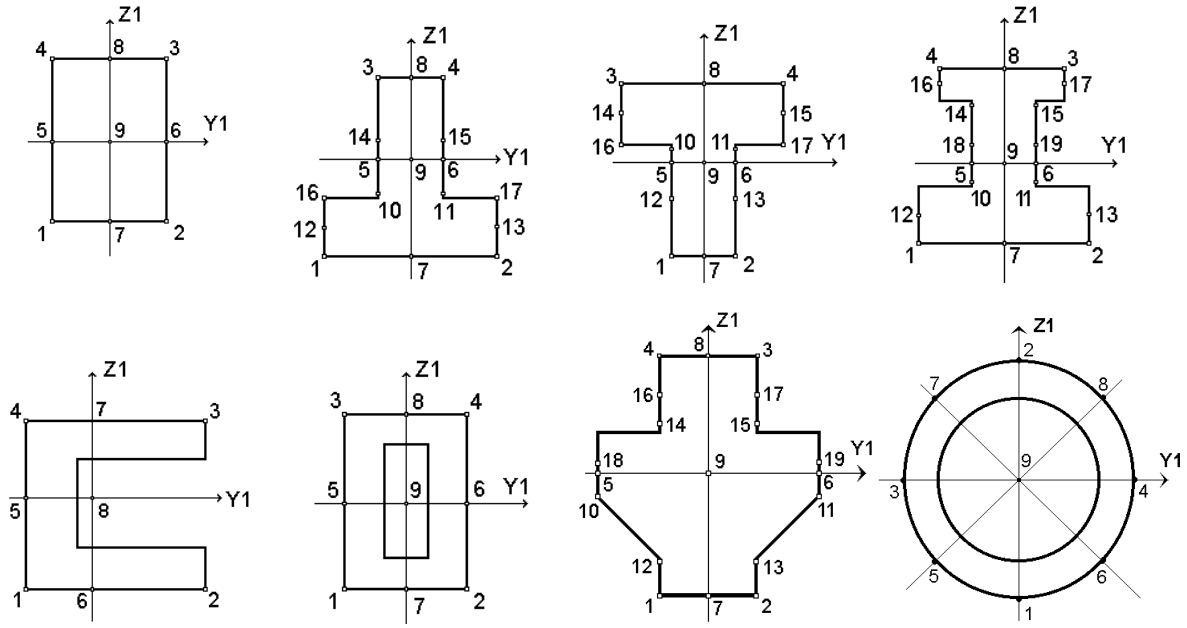


Figure 26.1-2. *Parametric sections*

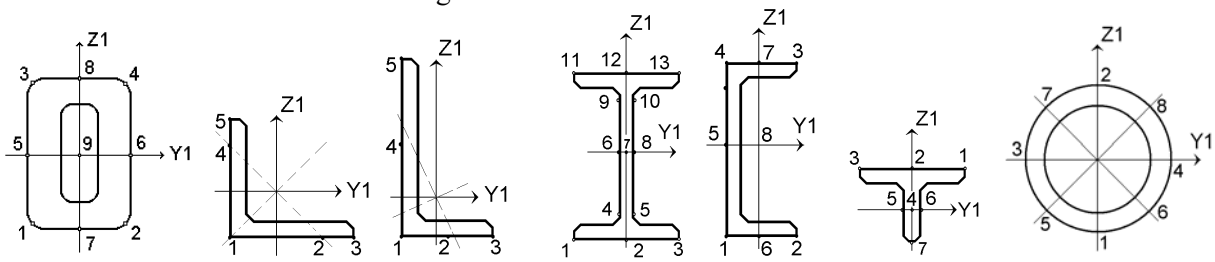


Figure 26.1-3. *Rolled steel profiles (with the numbers of characteristic points)*

26.2. Calculating Equivalent Stresses

In the case of simple deformation types, in particular the uniaxial stress state, the danger of the current stresses is estimated by comparing them to the experimentally determined value (to the yield stress for plastic materials or to the fracture strength for brittle materials). In the case of a complex stress state characterized by the principal stresses σ_1, σ_2 and σ_3, a certain hypothesis (failure theory) on the primary effect of a certain factor on the strength of material is used. A certain equivalent stress σ_e can be compared to the limit value σ_0^+, which corresponds to the simple uniaxial tension. The condition of non-occurrence of the limit state in the material is written as follows

$$\sigma_e = f\left(\sigma_1, \sigma_2, \sigma_3, k_1, ..., k_n\right) \le \sigma_0^+ ,$$

where $k_1,...,k_n$ — material constants, which can be absent.

Here are some designations of the used constants:

$\sigma_0 = \frac{1}{3}(\sigma_1 + \sigma_2 + \sigma_3)$ — average stress (hydrostatic pressure);

$\sigma_i = \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]}$ — stress intensity;

σ_0^+, σ_0^-, τ_0 — ultimate stresses of the material at the uniaxial tension, uniaxial compression, and pure shear, respectively;

$\chi = \sigma_0^+ / \sigma_0^-$; $\varphi = \sigma_0^+ / \tau_0$; $\psi = \sigma_0^- / \tau_0$; $\lambda = 1/\sigma_0^-$.

Sometimes it is more convenient to compare the equivalent stress to the limit σ_0^- corresponding to the strength of the sample material in the simple uniaxial compression. The respective equivalent stress is designated as σ_S.

The software implements four failure theories the information on which is given in Table 26.2-1. All of them belong to the isotropic materials and the conditions of static loading, when the history of the behavior of the structure does not affect the formulation of the failure conditions.

Table 26.2-1

No.	Failure theory	Expressions for the calculation of the equivalent tensile σ_e and compressive stress σ_s	Scope
1	Maximum normal stress theory	$\sigma_e = \sigma_1$ $\sigma_s = \sigma_3$	For brittle homogeneous materials (ceramics, glass)
2	Maximum linear strain theory	$\sigma_e = \sigma_1 - \mu\,(\sigma_2+\sigma_3)$ $\sigma_s = \sigma_3 - \mu\,(\sigma_1+\sigma_2)$	For brittle materials under simple compression
3	Maximum shear stress theory	$\sigma_e = \sigma_1 - \sigma_3$ $\sigma_s = \sigma_e$	For plastic materials with low hardening, for which the local plastic deformations in the form of slip lines are typical (tempered steel)
4	The theory of octahedral shear stresses or the specific distortion energy	$\sigma_e = \sigma_i = \left\{ \frac{1}{2}\left[(\sigma_1-\sigma_2)^2 + (\sigma_2-\sigma_3)^2 + (\sigma_3-\sigma_1)^2\right]\right\}^{1/2}$ $\sigma_s = \sigma_e$	For most plastic materials (steel, copper, nickel)

26.3. Preparing Data for the Calculation of the Principal and Equivalent Stresses

The initial data for the calculation of the principal and equivalent stresses are prepared in the respective dialog box (Fig. 26.3-1), which can be invoked from the **Special Initial Data** section of the **Project tree**. The calculation can be performed for individual loadings or for their combinations.

The type of the analysis is specified with the help of the checkboxes in the upper part of the dialog box.

The radio buttons of the **Failure theory** group enable to select the theory according to which the equivalent stresses are calculated.

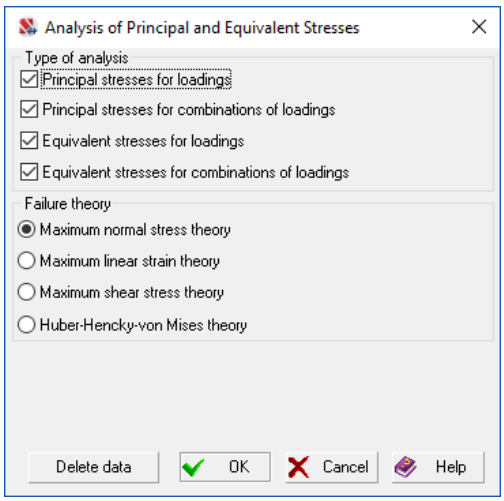


Figure 26.3-1. *The* **Analysis of Principal and Equivalent Stresses** *dialog box*

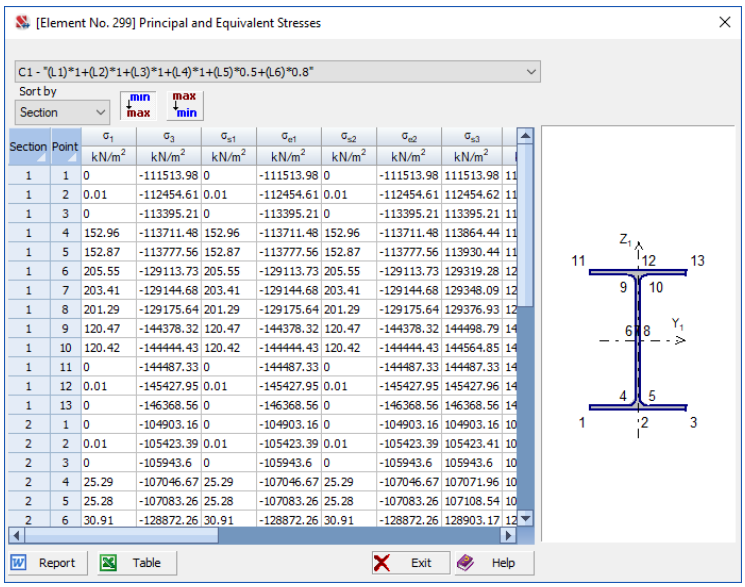


Figure 26.3-2. *Table of principal and equivalent stresses in the selected element*

If you click the **Delete data** button, the information necessary for the calculation of the principal and equivalent stresses will be deleted from the initial data, and the calculation will not be performed.

SCAD does not perform the calculation of the principal and equivalent stresses for the design combinations of forces. Since the design combinations are selected based on the search of the extreme value of a certain component of the stress state, it can not be connected to the search of the principal and equivalent stresses, which are calculated using all the components of the stress state.

The results of the calculation can be printed as tables with the help of the **Print Tables** section of the **Project tree** or the **Documenter** (see Chapter 13). Moreover, the **Information on the Element** dialog box enables to display the principal and equivalent stresses in the tabular form (Fig. 26.3-2).

26.4. Graphical Display of the Results of the Calculation of the Principal and Equivalent Stresses

The **Analysis of principal and equivalent stresses** mode of the **Postprocessors** toolbar is used for the graphical display of the results of the calculation of the principal and equivalent stresses. The mode of the graphical analysis of the results enables to plot isolines and isofields of the principal and equivalent stresses for plate elements, and to display the directions of one of the principal planes. Isosurfaces of stresses can be displayed for solid elements. Diagrams or a color scale of diagrams can be also obtained for bars (26.4-2).

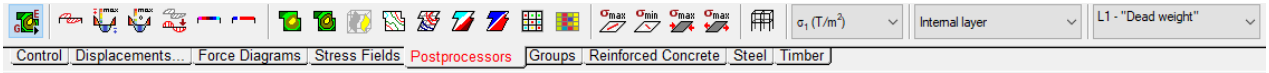


Figure 26.4-1. *The toolbar of the* **Analysis of principal and equivalent stresses** *mode*

The toolbar of this mode contains the following functions:

— stress diagrams;

— change the display axis of diagrams;

— color display of the values of principal/equivalent stresses;

— color display of values of principal/equivalent stresses with the selected value;

— color display of the diagrams of principal/equivalent stresses;

— color display of the diagrams of principal/equivalent stresses with the selected value;

— display isolines of principal/equivalent stresses;

— display isofields of principal/equivalent stresses;

— display isofields and isolines of principal/equivalent stresses;

— display isosurfaces of stresses in solid elements;

— display isolines of principal/equivalent stresses with a constant step;

— display isofields of principal/equivalent stresses with a constant step;

— display isofields of principal/equivalent stresses with the selected value;

— color display of stresses;

— color indication of stresses at the centers of elements;

— display directions of principal planes with the maximum stress;

— display directions of principal planes with the minimum stress;

— display directions of principal planes with the maximum compressive stress;

— display directions of principal planes with the maximum tensile stress;

— display design model.

The toolbar also contains the following lists: **Select type of stress**; **Select layer**, where the selected stresses of plates are analyzed; **Select loading** or combination of loadings.

The following designations are used when determining the type of the output information:

σ_1, σ_2, σ_3 — principal stresses;

σ_{E1} — equivalent stress (reduced to the equivalent tension) according to the first failure theory;

σ_{S1} — equivalent stress (reduced to the equivalent compression) according to the first failure theory;

σ_{E2}, σ_{E3}, σ_{E4} — equivalent stresses (reduced to the equivalent tension) according to the 2 – 4 failure theories;

σ_{S2}, σ_{S3}, σ_{S4} — equivalent stresses (reduced to the equivalent compression) according to the 2 – 4 failure theories.

The following numbering of the failure theories is used:

1 — maximum normal stress theory;
2 — maximum linear strain theory;
3 — maximum shear stress theory;
4 — Huber-Hencky-von Mises theory.

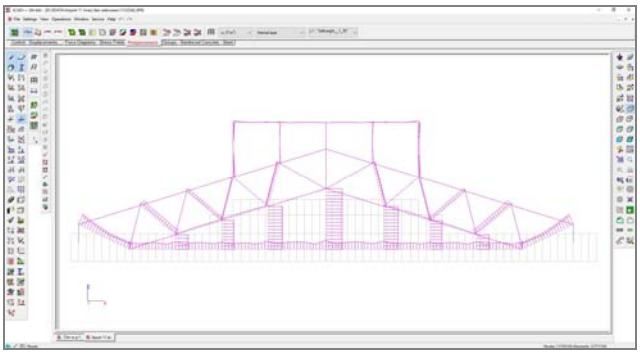
The control of the display is performed with the help of the buttons of the toolbar (Fig. 26.4-1) similarly to the output of forces and stresses, therefore only the description of the operations which are unique or have certain differences is given below.

Color Display of Principal/Equivalent Stresses in Bars

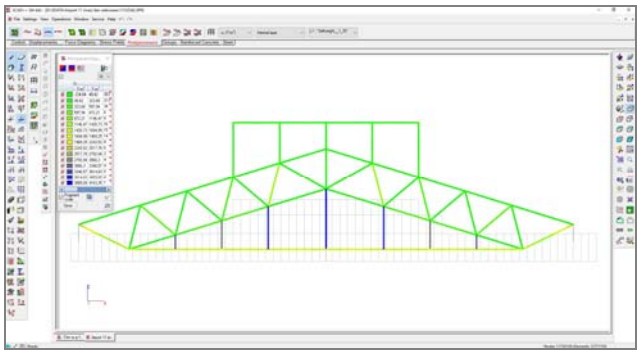
The software calculates principal/equivalent stresses for bar elements in a number of points specific to each cross-sectional shape. At the display of diagrams (mode) or the color display of diagrams (modes) a diagram of the minimal value for all points is plotted for the stresses σ_1, σ_{s1}, σ_{s2}, σ_{s3}, σ_{s4}; and a diagram of the maximal value for all points is plotted for the stresses σ_3, σ_{e1}, σ_{e2}, σ_{e3}, σ_{e4}.

Operations (**Color display of diagrams of principal/equivalent stresses**) and (**Color display of diagrams of principal/equivalent stresses with selected value**) are similar to operations , , but they provide gradient coloring of diagrams:

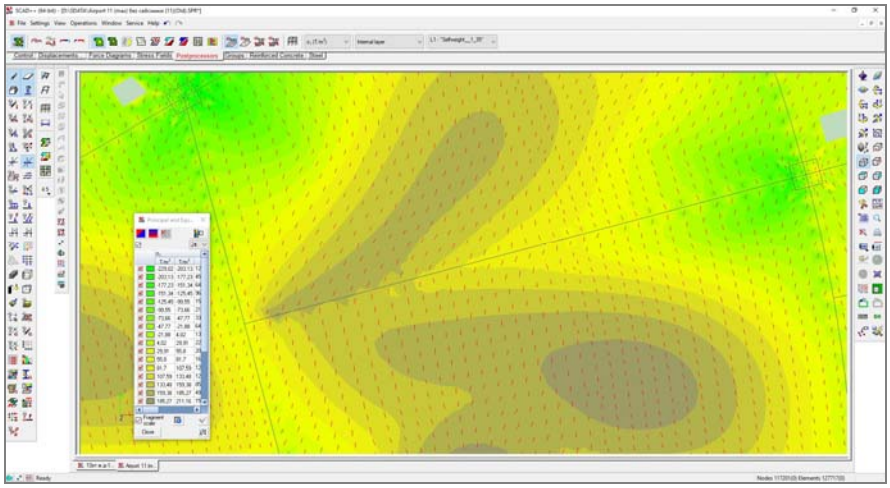
The button enables to change the display axis of diagrams and thus select a variant of the display with a "better visibility" (see Fig. 26.4-2).



Diagrams



Color display of diagrams



Isofields + direction of principal planes

Figure 26.4-2. *Graphical display of the principal/equivalent stresses in bars.*

Display of Directions of Principal Planes

The direction of the selected principal plane for plate elements is displayed as a segment with the center in the geometric center of the finite element. There are 4 modes for drawing the principal planes on the plate elements:

— a plane corresponding to the stress σ_1 is drawn;

— a plane corresponding to the stress σ_3 is drawn;

— if $\sigma_3<0$, then a plane corresponding to the stress σ_3 is drawn;

— if $\sigma_1>0$, then a plane corresponding to the stress σ_1 is drawn.

27. Nonlinear Calculations in SCAD

27.1. General Information

The problems of the analysis of load-bearing structures focused on the refined prediction of the peculiarities of the behavior of the system at all stages of its operation including the stages prior to the failure can not be usually solved by the methods of the linear structural mechanics. Educational literature [1, 2] and the majority of software consider the following "set of nonlinearities": deviation from Hooke's law (*physical nonlinearity*), failure to consider the equilibrium conditions in geometrical terms of the non-deformed state (*geometric nonlinearity*), accounting for the possible changes in the design model during the deformation process (*structural nonlinearity*). The nonlinearity related to the accumulation of stress and strain during the changes of the structure as it is created (*genetic nonlinearity*) should be added to them.

Without going into details, let's formulate some general warnings on the issue of the nonlinear analysis . Some of them (but not all) are given in Table 27.1-1.

Table 27.1-1

Peculiarity	Linear problems	Nonlinear problems
Relationship between displacement and load	Displacements are linearly dependent on the applied load	The relationship between displacement and load is nonlinear
Relationship between stress and strain	A relationship between stress and strain is assumed to be linear	In the problems where the physical nonlinearity is considered the "stress-strain" relationship can be a nonlinear function of stress or strain
Value of displacement	The change in geometry due to the displacement is assumed to be small, and is ignored when checking the equilibrium	Displacements can be not small, it is necessary to use the deformed state to check the equilibrium
Reversibility	All deformations are completely reversible and disappear when the system is unloaded	After the load is removed, the state of the system can differ from the initial one
Boundary conditions	Boundary conditions remain unchanged during the calculation	Boundary conditions can change, for example, the contact areas change
Sequence of load application	The sequence of load application is not important, the final state does not depend on it	The state of the structure can depend on the sequence of load application
Use of the results	The results of the calculation for different loads can be added or multiplied by certain coefficients in order to combine the design states	The decomposition of the problem into components with the subsequent combination of the results is impossible
Initial stress-strain state	The initial stress-strain state is insignificant	It is usually required to specify the initial stress-strain state, especially in the case of the nonlinearity related to the behavior of the material

You should be careful when using the concept of "physical nonlinearity", which is often identified with the plastic behavior of the material. However, many programs consider the case when a linear relationship between stress and strain is replaced by a nonlinear function, the same for loading and unloading. Such nonlinear elastic materials almost never occur in nature. Plastic materials usually have residual deformations after unloading.

SCAD enables to perform the calculation of the geometrically non-linear systems, cable-stayed structures, and problems with unilateral constraints.

27.2. Geometric Nonlinearity

Geometrically nonlinear behavior of the elastic system is related to the need to take into account the changes in the geometry of the system at its deformation under the load. Thus, technically, different geometric distortions can be considered, but **SCAD** enables to take into account only the so-called "von Karman approximation" when it is assumed that the squares of the rotation angles are of the same order of magnitude as the relative elongation of the material, which in turn is assumed to be small compared to one. This issue is considered in more detail in Sec. 3.5 of [3].

The interaction between bending effects and longitudinal forces (the phenomenon of buckling in bars) is considered, when, for example, the linear equation for a flexible plate of thickness h

$$\frac{D}{h}\nabla^2\nabla^2 w = \frac{q}{h}, \tag{27.2.1}$$

is replaced by the following equations

$$\begin{gathered}\frac{D}{h}\nabla^2\nabla^2 w = \frac{\partial^2\Phi}{\partial y^2}\cdot\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2\Phi}{\partial x^2}\cdot\frac{\partial^2 w}{\partial y^2} - 2\frac{\partial^2\Phi}{\partial x\partial y}\cdot\frac{\partial^2 w}{\partial x\partial y} + \frac{q}{h};\\ \nabla^2\nabla^2\Phi = E\left[\left(\frac{\partial^2 w}{\partial x\partial y}\right)^2 - \frac{\partial^2 w}{\partial x^2}\cdot\frac{\partial^2 w}{\partial y^2}\right]\end{gathered} \tag{27.2.2}$$

with respect to the deflection function $w(x,y)$ and the stress function $\Phi(x,y)$, through which the membrane forces are determined as

$$\sigma_x = \frac{\partial^2\Phi}{\partial y^2};\quad \sigma_y = \frac{\partial^2\Phi}{\partial x^2};\quad \tau_{xy} = \frac{\partial^2\Phi}{\partial x\partial y}.$$

The following designation of the differential operator is used in (27.2.1) and (27.2.2)

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2},$$

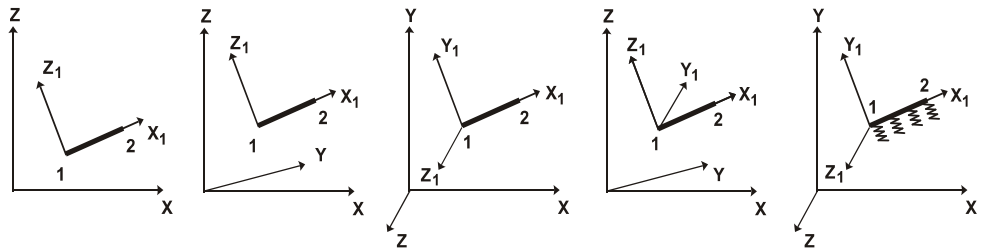
q — load value.

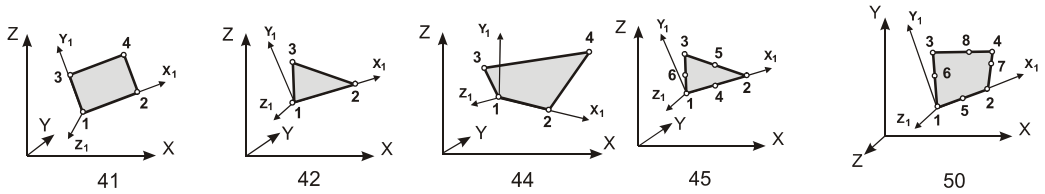
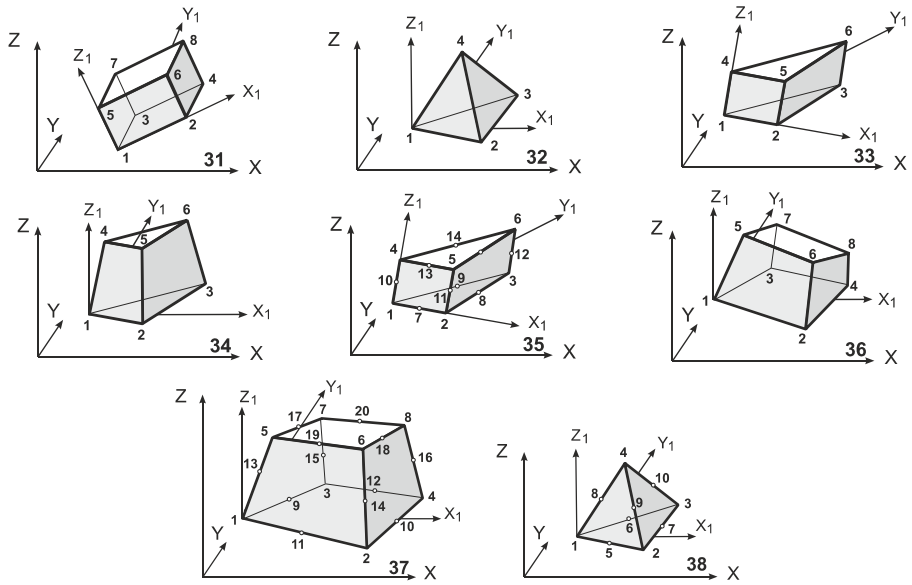
Geometrically nonlinear effects can be taken into account if the finite elements given in Table 27.2-1 are used in the model.

Elements 301–310, 601-610 are analogous to bar elements, and 341–350, 391-399, 641-650, 691-699, 741-750, 791-799 — to shell elements, and 331-338 — to solid elements. They are considered in detail in Sec. 3.3-3.5. The initial information and loads are specified for them in the same way. The calculated forces and sign conventions coincide as well (see Sec. 13.5).

If you want to change the type of a linear element to a nonlinear one to take the geometric nonlinearity into account, invoke the **Specify elements to take into account the geometric nonlinearity** operation in the **Assign** toolbar, select the respective elements in the model and click the **OK** button. A similar result can be obtained with the help of the **Specify the type of finite elements** operation. Before selecting the type of a shell or bar element (linear) in the respective dialog box (Fig. 27.2-1), you should first check the **Accounting for geometric nonlinearity** checkbox. The selected type is assigned to the elements in the traditional way.

Table 27.2-1

FE type	Linear analog	Description
301-310	1-10	
601-610	101-110	

341-350	41-50	
391-399	91-99	
641-650	141-150	
691-699	191-199	
741-750	541-550	
791-799	591-599	
331-338	31-38	

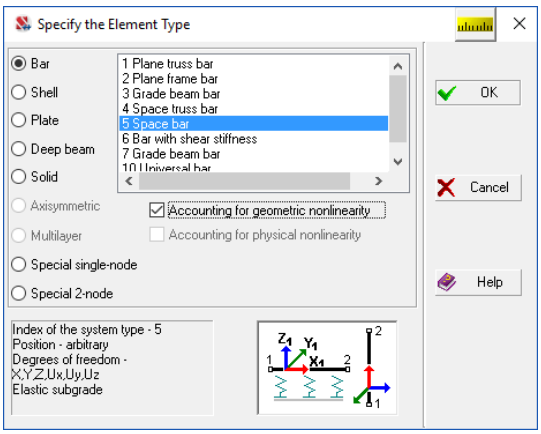


Figure 27.2-1. *The* **Specify the Element Type** *dialog box*

Cable-Stayed Elements

The design model of a cable-stayed element is a flexible thread with a small sag which in the initial state is characterized by the temperature T_0 and strain H_0, is loaded by a certain transverse load $q_0(x)$, and for

which the strain becomes equal to H when the chord is elongated by Δ and the transverse load changes to $q(x)$, and the temperature – to T.

The equation that relates these values for flat threads:

$$\Delta = \frac{(H-H_0)L}{EA} - \frac{D}{2H^2} + \frac{D_0}{2H_0^2} + \alpha(T-T_0)L\,, \tag{27.2.1}$$

where

$$D = \int_0^L Q^2(x)dx\,, \qquad D_0 = \int_0^L Q_0^2(x)dx\,,$$

$Q(x)$ — shear force arising in a simply supported beam of the span L from the action of load $q(x)$, directed across the chord of the cable-stayed element;

$Q_0(x)$— the same from the load $q_0(x)$;

α — thermal expansion coefficient;

E — elastic modulus of the thread material;

A — cross-sectional area of the thread.

The equation (0.1) can be expressed in the dimensionless variables $\delta = \Delta/L$, $\delta_0 = \Delta_0/L$, $t = H/(EA)$, and the following designations

$$\Delta_0 = \frac{H_0 L}{EA} - \alpha(T-T_0)L - \frac{D_0}{2H_0^2}\ , \quad K = \frac{D}{2L(EA)^2}$$

as:

$$\delta = t - \frac{K}{t^2} - \delta_0\,. \tag{27.2.2}$$

Diagram of the δ vs. t relationship at $\delta_0 = 0$ is given in Fig. 0-1, the value δ_0 can be easily taken into account by moving the origin. It can be seen from the figure that as K decreases the curve $t = t(\delta)$ approaches its asymptotes. The X-axis is an asymptote when $\delta < 0$, and the line with the tangent of the angle of EA/L – when δ are positive. Instant compliance of the cable-stayed element in the vicinity of a certain stress state characterized by the strain H can be easily determined by differentiating the equation of state (27.2.1):

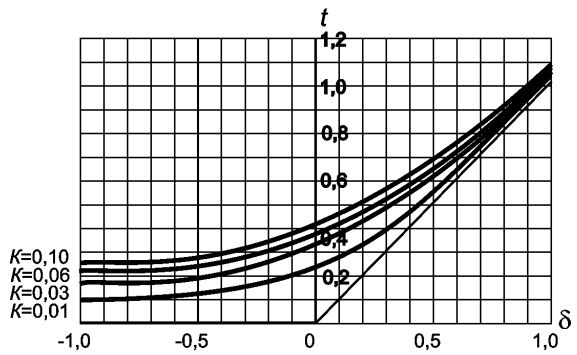


Figure 27.2-2. *Dimensionless relationship between the strain and the increase of the chord*

$$\frac{d\Delta}{dH} = \frac{L}{EA} + \frac{D}{H^3}\,. \tag{27.2.3}$$

When performing the analysis of cable-stayed systems it is important to specify the reference state correctly, which is characterized by the initial values of the shear loads on cable-stayed elements q_0 (it is usually a component of the load from the self-weight of the cable stay perpendicular to the chord of the cable-stayed element) and the values of the strains $H_0 \geq 0$.

The main difficulty is that all values of H_0 must correspond to the equilibrium state of the system in which the prestressing creates a certain pattern of internal forces corresponding to the initial external load q_0, which can be absent. There are usually many possible equilibrium conditions of the system each of which corresponds to its distribution of prestressing forces in cable stays $H_{0,i}$ (i = 1,…,m), where m — number of cable stays in the system. Therefore, when designing the cable-stayed structures the engineer has a task of selecting a rational initial stress state from the set of allowable ones based on the certain criteria of optimality and structural limitations. These criteria and limitations may vary depending on the particular design problem.

The following operation is provided in **SCAD** in order to facilitate these calculations when working with cable-stayed elements — cable stays are deleted, and the prestressing forces are applied to the remaining part of the structure. The system might turn out to be unstable after deleting the cable stays,

which is typical for masts with a hinged trunk. In order to avoid this problem it is sometimes necessary to install additional elastic constraints with such a small stiffness that their effect on the general stress-strain state can be neglected.

The parameters of a cable-stayed element can be specified after clicking the **Create cable-stayed elements** button in the **Special elements** toolbar (see Sec. 6.4). The **Rigidity of Bar Elements** dialog box (Fig. 27.2-3) appears after clicking this button. If the input method other than the numerical and parametric description is selected, the only difference from the specification of stiffness properties of bar elements is the fact that it is necessary to specify the value of prestressing. The dialog box for the numerical and parametric description is shown in Fig. 27.2-4.

When specifying the stiffness properties by selecting the section from the assortment of rolled profiles the **Zinc coated steel wire ropes** assortment is available only for cable-stayed elements.

If the assortment of zinc coated steel wire ropes is used to specify the stiffness properties of cable-stayed elements, ***the elastic modulus of steel is specified in accordance with the recommendations of the respective codes*** and can be changed by the user.

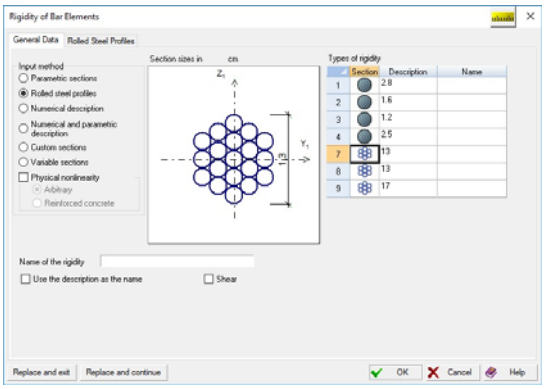


Figure 27.2-3. *The* **General Data** *tab of the* **Rigidity of Bar Elements** *dialog box*

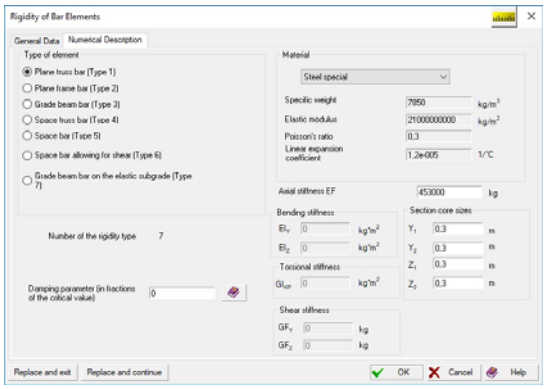


Figure 27.2-4. *The* **Numerical Description** *tab of the* **Rigidity of Bar Elements** *dialog box*

In addition to performing the static calculations it is also necessary to carry out the calculations of the free oscillations, because otherwise it is impossible to determine the pulsation component of the wind load, perform the seismic analysis, etc. The spectral analysis of the system (determination of frequencies and its natural oscillation modes) makes sense only in the linear formulation. In the case of the nonlinear problems we can talk about the expansion of the motion of the system by the natural oscillation modes in the vicinity of the considered equilibrium position at the linearization of the behavior of the system in this vicinity. The determination of the stress-strain state (SSS) of the system at its static loading should be performed taking into account the geometrically nonlinear effects mentioned above. In the case of the dynamic analysis, knowing the SSS of the structure from the acting static loads, it is necessary to construct a dynamic model of the so-called linearized system.

This concept refers to a structure working in the geometrically linear formulation but taking into account the achieved level of forces in the cable stays in the state prior to the linearization, as well as the achieved level of compressive forces in the respective compressed elements of the design model.

A linearized system usually refers to the initial system all the constituent elements of which are considered in the linear formulation but with tangential (instantaneous) stiffness matrices.

Such a system can be built by replacing cable-stayed elements by bars, the rigidity of which is calculated by the following formula

$$\overline{EA} = \frac{1}{\dfrac{1}{EA} + \dfrac{D_y + D_z}{LH^3}}, \qquad (27.2.4)$$

which is easily obtained from (27.2.3) and takes into account the fact that a cable-stayed element can be loaded in two planes.

A cable-stayed element has type 308 in the finite element library. It allows the same types of loads as the element of type 305 — bar of the general position described in Sec. 3.3. Strain of the cable stay N_X is obtained in the result of the calculation. More detailed information on the analysis of cable stays is given in [4].

Note: **If the "self-weight" load is specified in the nonlinear loading, it will be ignored on the cable-stayed elements, since it is taken into account automatically. The following warning will appear in the analysis log:**

The additionally specified "self-weight" load on the following cable-stayed elements will be ignored in the nonlinear loading (No. of the loading): (list of elements)

Unilateral Constraints

Unilateral constraints are constraints which can take only the forces of a certain sign and are disabled when the forces of the opposite sign appear, or, which is the same, allow displacements only in a certain direction and not restrain the displacements in the opposite direction. The respective finite element implemented in the software models the behavior of a perfect unilateral constraint (without friction).

Like the general type elastic constraints (see Sec. 3.8, 6.4), unilateral constraints can be single-node (between the node and the "ground") or two-node (between two nodes). In order to specify the characteristics of the unilateral constraint click the button in the **Nodes and Elements** tab.

The **Unilateral Constraints** dialog box (Fig. 0-1) will appear, where you can specify the performance of the unilateral constraint (constraint is disabled at compression or at tension), its direction and rigidity (force required for the unit displacement).

The parameters of the initial state of a unilateral constraint are its distinctive feature. For example, the gap Δ_0 in a unilateral constraint or the prestressing which can be interpreted as the negative gap, i.e. $\Delta_0 < 0$, can be taken into account for the simplest model of the bar which can take tension and is disabled when compression appears (Fig. 27.2-6).

If you want to enable a constraint when working in the system, it is necessary to select the gap first, or if you want to disable it, it is necessary to overcome the prestress.

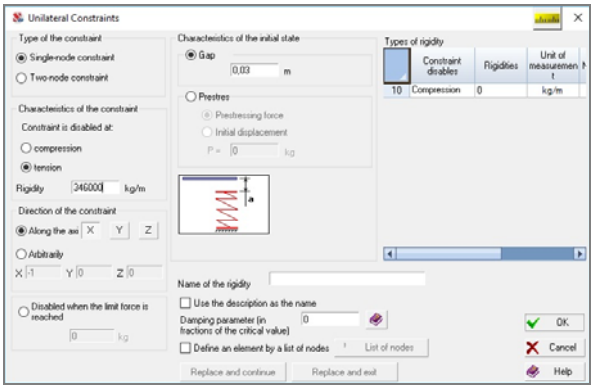


Figure 27.2-5. *The* **Unilateral Constraints** *dialog box*

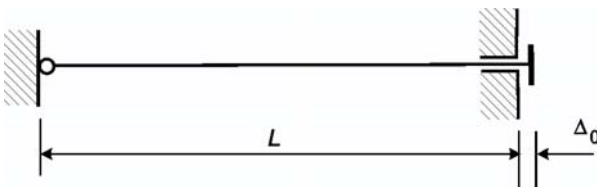


Figure 27.2-6. *Unilateral bar*

Two variants of the diagram of the behavior of such an element as the relationship between the longitudinal force H and the increase of the distance between the end points Δ are given in Fig. 27.2-7, where a variant with prestressing (equal to H_0) corresponds to the case *a*, and a variant of the system with a gap Δ_0 corresponds to the case *b*.

Slope of the diagram is equal to EA/L (EA — tensile stiffness of the bar, L — length of the bar).

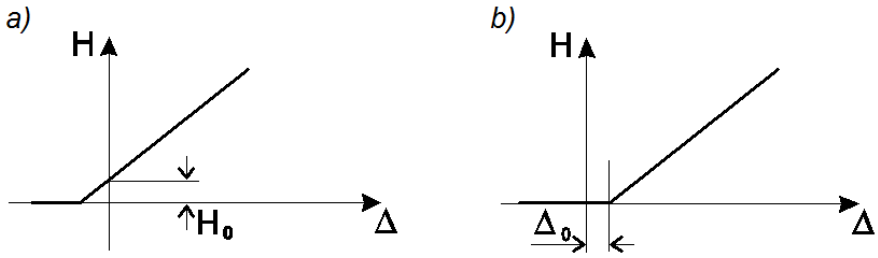


Figure 27.2-7. *Variants of the diagram H — Δ*

A single-node element of the unilateral constraint has type 351 in the finite element library, and the two-node element has type 352. Their linear analogues are elements modeling the constraints of finite rigidity, 51 and 55, respectively. A force (reaction in the spring) corresponding to the degree of freedom in the direction of which the constraint of finite rigidity acts when it is enabled is obtained in the result of the calculation.

27.3. Physical Nonlinearity

General Information

Nonlinear calculations are required in analyses of systems with stress-strain relationships different from Hooke's law (the simplest examples are the tensile failure of concrete, plastic yielding that may occur in elements of steel structures).

It should be noted that there are many problems that have not been solved yet, and the lack of these solutions prevents the full use of nonlinear analysis in the design (see [11]).

Many software packages enable to analyze systems taking into account the nonlinear behavior of the material. The user has to specify a σ-ε diagram for each material as the initial data in many of them. **SCAD** uses a different approach. The user has to select a physical model of the behavior of the material for the calculation. He (the user) can affect (by setting certain options) the preliminary diagram, but the final one will be automatically generated by the program. The reasons for this approach include:

- Not every diagram that can be specified by the user has a physical meaning;
- Many ideal diagrams (for example, the Prandtl diagram) do not always provide a numerically stable solution, because there is a danger of looping at break points or significantly slowing the convergence of a nonlinear algorithm based on the Newton-Raphson method and intended to solve a system of nonlinear algebraic equations of the finite element method (this served as a stimulus for the use of the Pade approximation);
- Engineers with no experience in nonlinear calculations tend to use diagrams of concrete, reinforcement and steel behavior, which are given in SNiP. But they do not take into account the fact that the diagrams given in SNiP, DBN, ... are intended for other purposes (for example, the analysis of the behavior of normal sections of reinforced concrete elements based on a nonlinear deformation model). There are, however, exceptions. Section 3.1.7 of EN1992 provides diagrams for the design of *cross-sections* of reinforced concrete elements, and Section 3.1.5 of EN1992 provides a diagram for nonlinear *structural* analysis, when it is necessary to take into account the descending branch of the σ-ε curve (these diagrams are different!). Similarly, Section 5.4.3 of EN 1993-1-1 provides diagrams for steel structures, and Section C6 of Annex C Finite Element Methods of Analysis of EN 1993-1-5 provides diagrams for the finite element analysis.
- The diagrams given in the codes do not fully take into account the behavior of concrete and reinforcement. For example, they do not allow for the degradation of concrete in the tension area during cracking. The reduction in the strength of concrete in the compression area after reaching the yield point is not taken into account as well.

An important feature of the used algorithm is overcoming the problem of divergence of the numerical solution after passing the yield point of the σ-ε diagram for concrete in the case of mesh refinement. Here, the factor that prevents the numerical instability is the presence of reinforcement, the σ-ε diagram of which does not have a descending branch. But even in this case the angle between the descending branch and the

strain axis is limited, because if the drop is too rapid, the behavior of the concrete – reinforcement system is unstable as well.

The design theory and finite element procedures are described in sufficient detail in [10]. We will only briefly describe how to specify the necessary initial data.

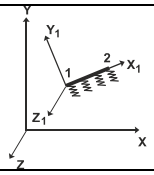
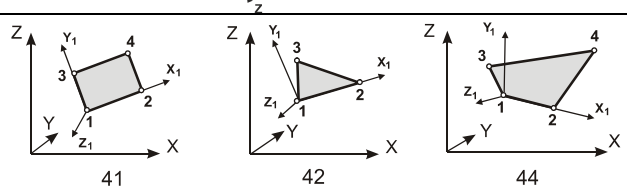
Physically Nonlinear Finite Elements

Physically nonlinear effects can be taken into account if the finite elements given in Table 27.3-1 are used in the model.

Elements 405, 410 are analogous to bar elements, and 441, 442, 444 — to shell elements. They are considered in detail in Sec. 3.3-3.4. The initial information and loads are specified for them in the same way. The calculated forces and sign conventions coincide as well (see Sec. 13.5).

If you want to change the type of a linear element to a nonlinear one to take the physical nonlinearity into account, invoke the **Specify elements to take into account the nonlinearity** operation in the **Assign** toolbar, select the respective elements in the model and click the **OK** button. A similar result can be obtained with the help of the **Specify the type of finite elements** operation. Before selecting the type of a shell or bar element (linear) in the respective dialog box (Fig. 27.3-1) you should first check the **Accounting for physical nonlinearity** checkbox. The selected type is assigned to the elements in the traditional way, and the number of the type of the nonlinear element will be automatically modified by adding 400.

Table 27.3-1

FE type	Linear analog	Description
405, 410	5,10	
441 442 444	41 42 44	 41 42 44

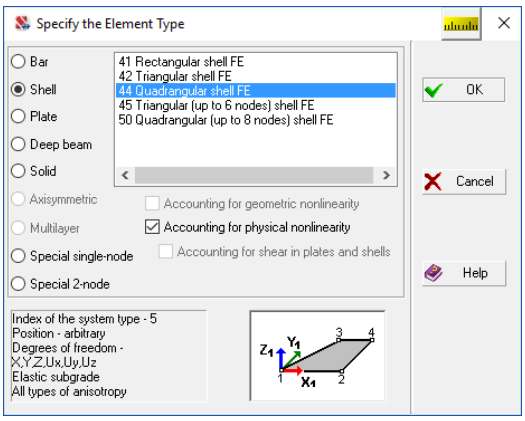


Figure 27.3-1. *The* **Specify the Element Type** *dialog box*

Material Models

When specifying the initial data for physically nonlinear finite elements, it is necessary to specify the data on the nature of the (nonlinear) behavior of the material. In order to do this, check the **Physical nonlinearity** checkbox in the dialog boxes for specifying stiffness parameters of bars and plates (see Fig. 27.3-2).

In this case it will be possible to specify the type of the material. Two types of nonlinear materials are implemented in **SCAD**. If the first one called *Arbitrary* is selected, it means that the structure is made of mono-material. The second option is reinforced concrete, i.e. a bimaterial structure consisting of concrete and reinforcement.

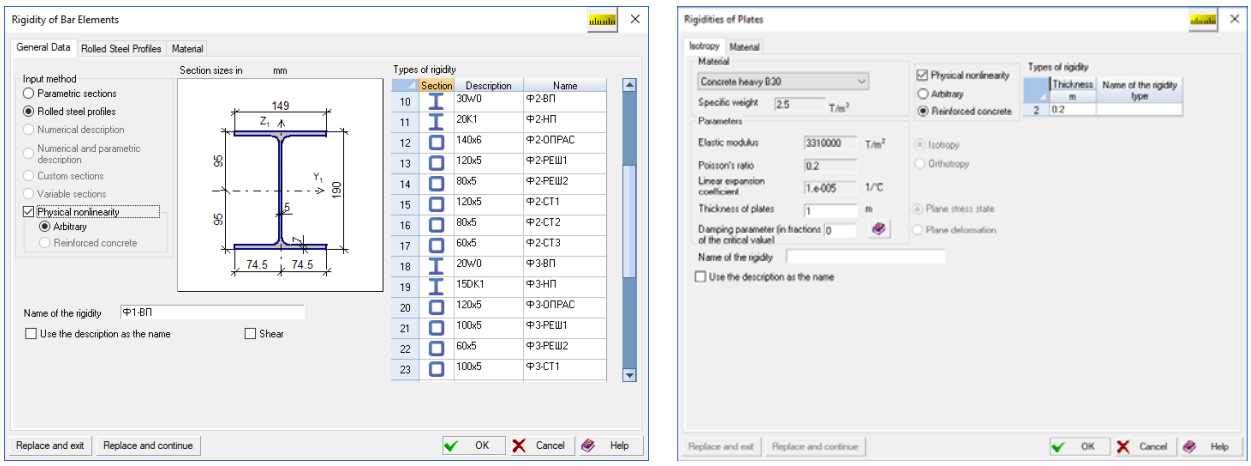


Figure 27.3-2. Dialog boxes for specifying stiffness parameters of bars and plates

When specifying the data for bar elements the physical nonlinearity is implemented for parametric sections and rolled steel. If rolled steel is selected, the **Reinforced concrete** option is obviously not available.

The properties of the material have to be specified in the **Material** tab.

Arbitrary Material

Let's consider the data input for the case of an arbitrary material (Fig. 27.3-3).

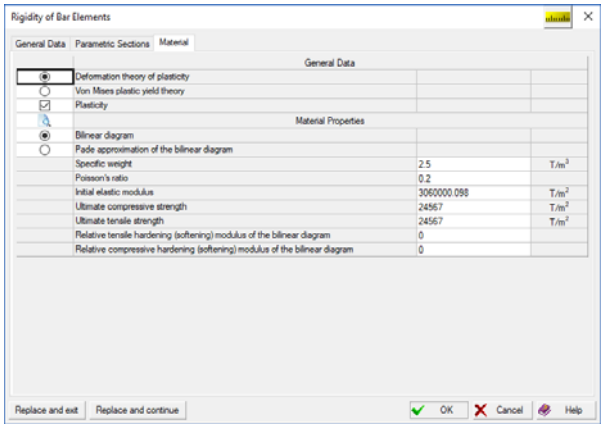


Figure 27.3-3. The **Material** tab for the case of an arbitrary material

First you have to select a theory which describes the behavior of the material. The *deformation theory of plasticity* [1], [4] and *von Mises plastic yield theory* [12] are implemented. The **Plasticity** checkbox enables to specify whether the plastic behavior should be taken into account. If it is not checked, nonlinear elasticity will be assumed for the material; otherwise such plasticity effects as the residual deformations after unloading will be taken into account.

In the case of the deformation theory of plasticity you can select the type of the σ-ε diagram (bilinear diagram or the Pade approximation of the bilinear diagram). The figure below shows an example of a symmetric bilinear diagram and a Pade approximation of this diagram.

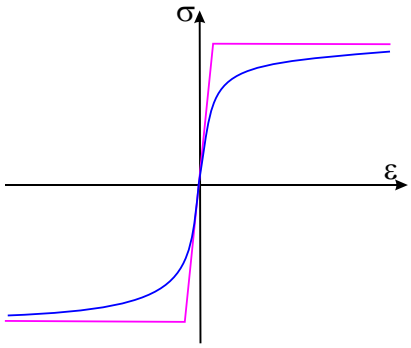


Figure 27.3-4. Bilinear diagram and the Pade approximation of this diagram

Depending on the selected theory the user also has to specify the following data:

Table 27.3-2

Type of data	Comments
specific weight	When creating a new type of rigidity, these data will be copied from the data for the linear analysis, but can be modified for the nonlinear analysis
Poisson's ratio	
initial elastic modulus	
ultimate compressive and tensile strength of the material	Specified only when the deformation theory of plasticity is used. Specifying different values of ultimate compressive and tensile strength of the material enables to set asymmetric diagrams
relative tensile hardening (softening) modulus of the bilinear diagram	
relative compressive hardening (softening) modulus of the bilinear diagram	
yield stress	Specified only when the von Mises plastic yield theory is used.
relative hardening (softening) modulus of the bilinear diagram	
Maximum tensile strain ($\varepsilon_{t,max} > 0$)	**Used only for the plastic yield theory.** When $\varepsilon_e \geq \varepsilon_{t,max}$, where ε_e is the reduced strain, the material fiber degradation is simulated using the elements of fracture mechanics. If $\varepsilon_{t,max}$ is set very large, for example, $\varepsilon_{t,max} = 1000$, the tensile strain of the fiber is not limited.
Maximum compressive strain ($\varepsilon_{c,min} < 0$)	**Used only for the plastic yield theory.** When $\varepsilon_e \leq \varepsilon_{c,min}$, where ε_e is the reduced strain, the material fiber degradation is simulated using the elements of fracture mechanics. If the absolute value of $\varepsilon_{c,min}$ is set very large, for example, $\varepsilon_{c,min} = -1000$, the compressive strain of the fiber is not limited.

The effect of relative hardening (softening) moduli θ on the material model can best be demonstrated with an example of idealized diagrams σ-ε (Fig. 27.3-5).

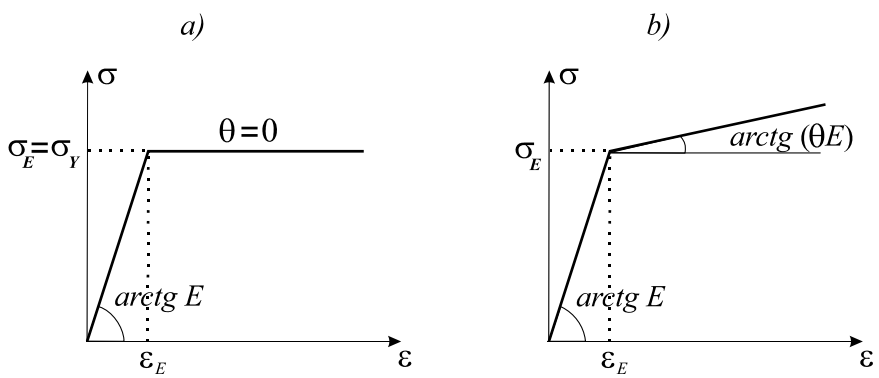


Figure 27.3-5. Idealized diagrams σ-ε:
a) Prandtl diagram; *b*) Prandtl diagram with linear hardening

After specifying all the necessary information you can use the button to preview the σ-ε diagram (Fig. 27.3-6).

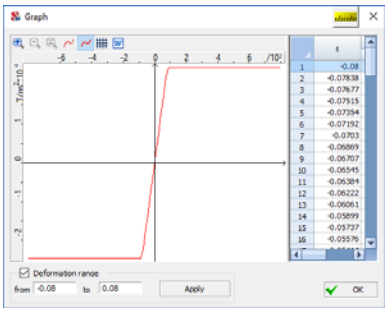
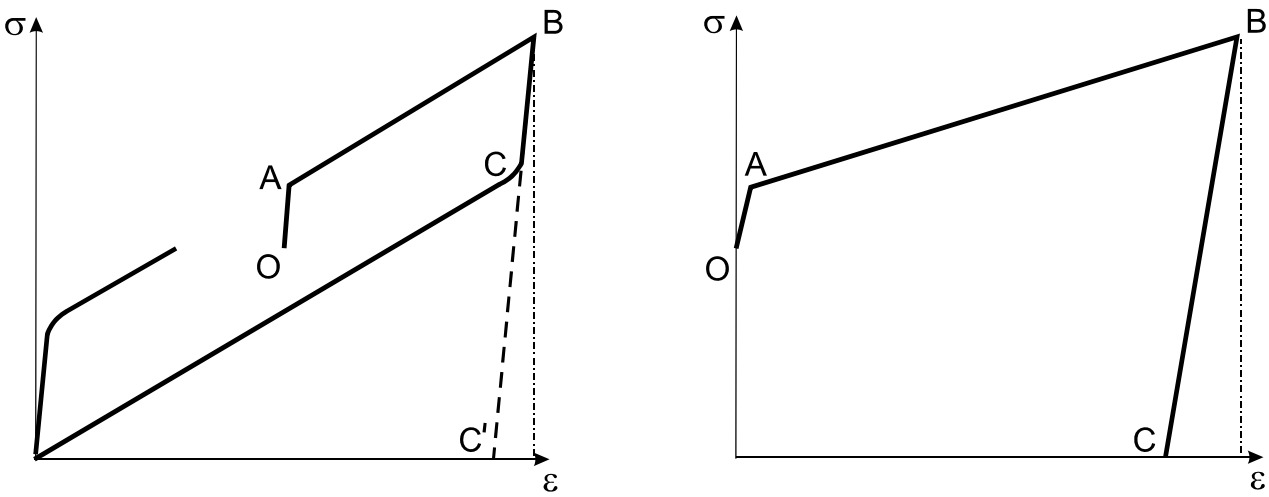


Figure 27.3-6. Preview of the σ-ε diagram

The **Strain range** checkbox in this dialog box enables to specify the range of strain ε variation, which will be applied after clicking the respective button.

In the case of the plastic yield theory for an arbitrary material the σ – ε diagram is symmetric bilinear – tensile and compressive yield points, as well as the tensile and compressive hardening/softening moduli coincide. Moreover, the **Take into account the Bauschinger effect** checkbox appears. The Bauschinger effect refers to a property of materials where the yield strength is reduced under the deformation of the material in the opposite direction after being subjected to plastic deformations in the given direction. Let us explain this phenomenon with the following example. Consider a steel bar subjected to an axial force. The bar is in a uniaxial stress state ($\sigma_x \neq 0$, other strain tensor components are equal to zero). First, the force increases from zero to a given value – the bar is stretched, then the force decreases to zero, then to a given value with the opposite sign, and then increases to zero. Thus, a simple constant amplitude loading cycle is set. Two material models are considered. The first model takes into account the Bauschinger effect, while the second one does not. Figure 27.3-7 shows the corresponding σ – ε diagrams obtained directly by solving this test problem.



Kinematic hardening – Bauschinger effect is taken into account

Isotropic hardening – Bauschinger effect is ignored

Figure 27.3-7.

When the Bauschinger effect is taken into account (Fig. 27.3-7 on the left), the segment OA corresponds to the elastic behavior of the material, and the segment AB corresponds to the hardening elastoplastic behavior. The force reaches its peak value at the point B and then decreases. Elastic unloading occurs in the bar first, and then it is loaded until the compressive yield point is reached (linear segment BC). When the Bauschinger effect is taken into account, the compressive yield point of the material moves from the point C′ to the point C. And when it is not taken into account (Fig. 27.3-7 on the right), the point C remains in the same place. The compressive load reaches its peak value at this point, and as the load

decreases the representative point moves along the straight line BC from the point C to zero, performing elastic unloading.

We see that we obtain a completely different behavior of the material depending on whether we take into account the Bauschinger effect or not. The σ – ε diagram has a pronounced hysteresis character in the first case unlike the second one. Kinematic hardening is used in the plastic yield theory in the first case, where the yield surface does not expand once the representative point has left it, but moves in the space of principal stresses as a rigid body parallel to the hydrostatic axis ($\sigma_1 = \sigma_2 = \sigma_3$). Isotropic hardening is used in the second case, where the yield surface expands uniformly.

A material model should be selected based on the properties of this material. The Bauschinger effect occurs in steel and most metals, and it does not occur in concrete and masonry. **It is especially important to take into account the Bauschinger effect in the case of cyclic and dynamic loading.**

Reinforced Concrete

Let's now consider the case of reinforced concrete (Fig. 27.3-8). The general procedure for specifying the data is similar to that described above for the arbitrary material. But in this case the data has to be specified both for concrete and for reinforcement.

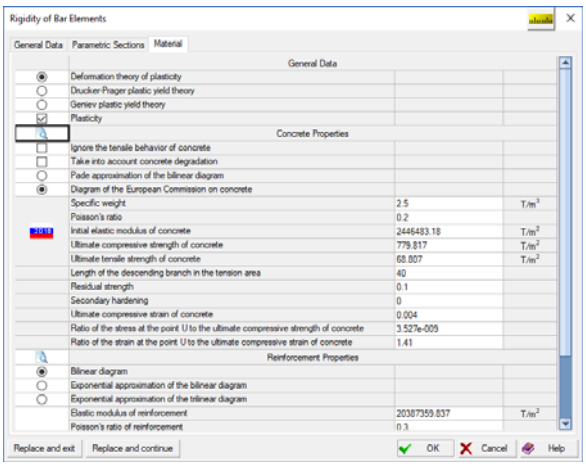


Figure 27.3-8. The **Material** tab for the case of reinforced concrete

The behavior of concrete in the implemented calculations is described on the basis of the deformation theory of plasticity [1][4], the Drucker-Prager plastic yield theory [13] or the Geniev plastic yield theory [2].

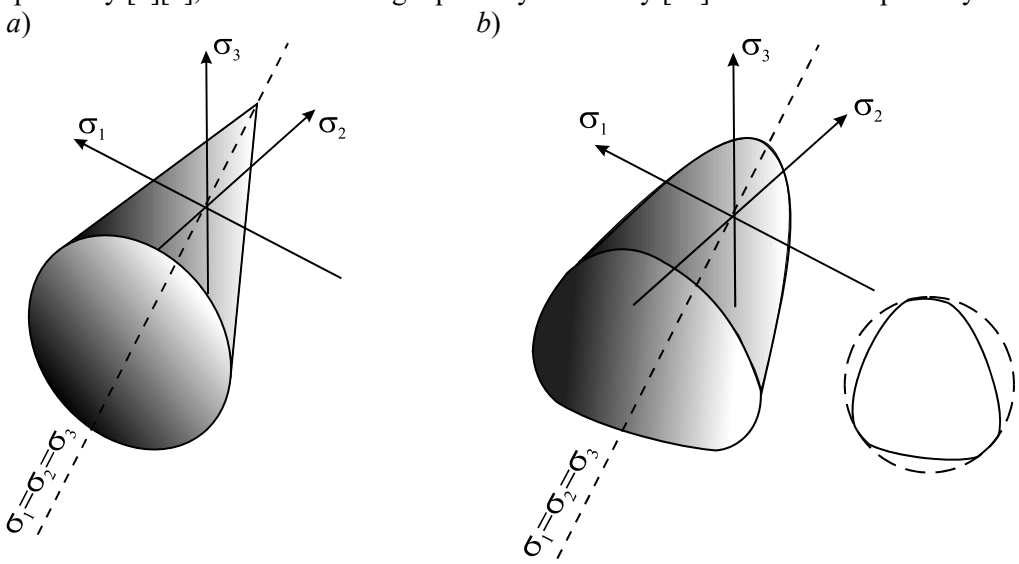


Figure 27.3-9. Yield surface: *a* - Drucker-Prager, *b* - Geniev

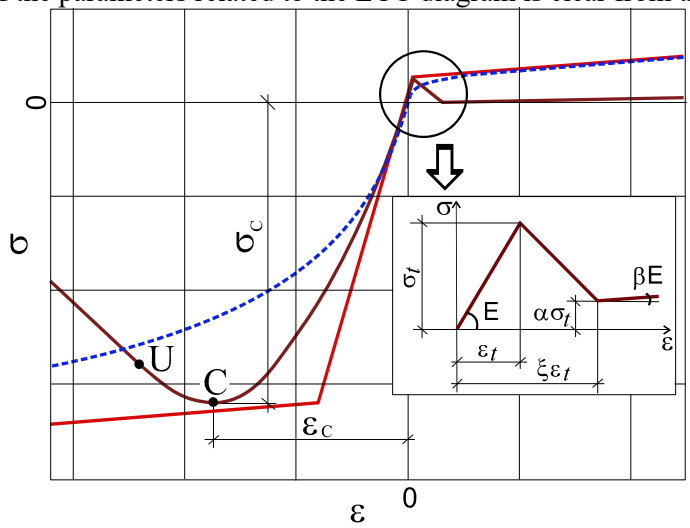
You can specify for concrete whether to ignore the tensile behavior of concrete, to take into account the plasticity and/or concrete degradation caused by the formation and development of cracks.

A complete list of data about the properties of concrete that can be requested from the user is given in Table 27.3-3.

Table 27.3-3

Type of data	Comments
Type of diagram: • Pade approximation of the bilinear diagram • Diagram of the European Commission on concrete (**ECC**)	A diagram can be selected only when the deformation theory of plasticity is used.
Options: • Ignore the tensile behavior of concrete • Take into account concrete degradation	The *Ignore the tensile behavior of concrete* option can be used only when the deformation theory of plasticity is used. If the deformation theory of plasticity is used, the *Take into account concrete degradation* option enables to take into account only the residual strength of concrete, specified by the parameter α, during the transition from the compression to the tension area in the ECC diagram. If the plastic yield theory is used, this option allows you to select the model of concrete degradation caused by the formation and development of cracks (see description below).
specific weight	When creating a new type of rigidity, these data will be copied from the data for the linear analysis, but can be modified for the nonlinear analysis
Poisson's ratio	
Initial elastic modulus of concrete	
Ultimate compressive strength of concrete	
Ultimate tensile strength of concrete	
Relative tensile hardening (softening) modulus of the bilinear diagram	When the plastic yield theory is used, it enables to specify the concrete softening in the tension area. If the *Take into account concrete degradation* option is enabled, concrete softening is ignored, but the concrete degradation during cracking is modeled using the damage parameter.
Relative compressive hardening (softening) modulus of the bilinear diagram	When the plastic yield theory is used, it enables to specify the concrete softening in the compression area. In this case, the yield surface is compressed once the representative point reaches it.
Ultimate compressive strain of concrete	
Residual strength α	Limits the decrease of the reduced stress σ_e during the degradation of concrete: $\sigma_e = \begin{cases} \sigma_e, & \sigma_e > \alpha\sigma_t \\ \alpha\sigma_t, & \sigma_e \le \alpha\sigma_t \end{cases}, \quad t \leftrightarrow c$
Length of the descending branch in the tension area	These parameters are required only when the ECC diagram is used
Secondary hardening	
Strain corresponding to the ultimate compressive strength of concrete	
Ratio of the stress at the point U to the ultimate compressive strength of concrete	These parameters are required only when the ECC diagram is used. The default values are 0.85 and 1.41, respectively
Ratio of the strain at the point U to the ultimate compressive strain of concrete	
Out-of-roundness of the Geniev paraboloid	This parameter is required only when the Geniev plastic yield theory is used (the default value is 0.577).

The physical meaning of the parameters related to the ECC diagram is clear from the Figure 27.3-10



where: bilinear; Pade approximation; ECC diagram.

Figure 27.3-10. Concrete behavior diagrams

The ECC diagram is represented by the following relationships

$$\sigma(\varepsilon)=\begin{cases} E\varepsilon, \quad 0\le\varepsilon\le\varepsilon_t, \\ \alpha\sigma_t+\dfrac{(1-\alpha)E}{1-\xi}(\varepsilon-\xi\varepsilon_t), \quad \varepsilon_t\le\varepsilon\le\xi\varepsilon_t, \xi>1, \\ \alpha\sigma_t+\beta E(\varepsilon-\xi\varepsilon_t), \quad \varepsilon>\xi\varepsilon_t, \\ \dfrac{\dfrac{E}{E_r}\varepsilon_r\sigma_c}{1+A\varepsilon_r+B\varepsilon_r^2+C\varepsilon_r^3}, \quad \varepsilon_u\le\varepsilon<0. \end{cases}$$

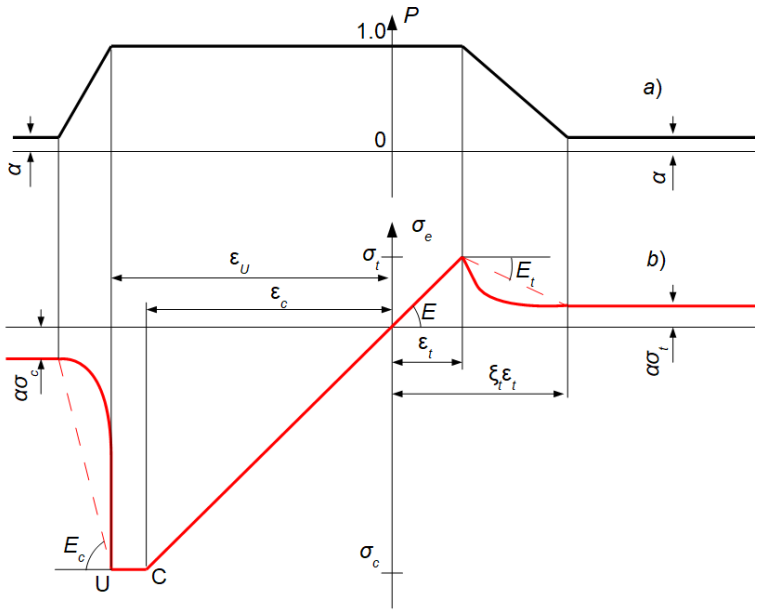
where point C(ε_c, σ_c) corresponds to the ultimate compressive strength, point U(ε_u, σ_u) is the limit point limiting the strain variation range for the ECC curve, and according to the recommendations [5], $\sigma_u = 0.85\cdot\sigma_c$ and $\varepsilon_u = 1.41\cdot\varepsilon_c$. Other parameters correspond to those given in [11]:

$$E_r=\sigma_c/\varepsilon_c, \quad \varepsilon_r=\varepsilon/\varepsilon_c, \quad E_u=\sigma_u/\varepsilon_u, \quad p=\varepsilon_u/\varepsilon_c,$$

$$A=\left[\frac{E}{E_u}+\left(p^3-2p^2\right)\frac{E}{E_r}-\left(2p^3-3p^2+1\right)\right]/\left[p\left(p^2-2p+1\right)\right],$$

$$B=2\frac{E}{E_r}-3-2A, \quad C=2-\frac{E}{E_r}+A.$$

When the plastic yield theory is used, the *Take into account concrete degradation* option allows you to select a method allowing for the reduction in concrete strength caused by the formation and development of cracks. If the respective checkbox is unchecked, then when the image point reaches the yield surface, it is compressed – a softening mechanism is implemented. In this case, the compression program is determined by the descending branch of the σ – ε diagram and is described in detail in [10, pp. 23–26], and the results of solving test problems are presented in Section 3 [10].



$$P(\varepsilon_e)=\begin{cases}1.0, & \varepsilon_e\in[\varepsilon_U,\varepsilon_t],\\ 1-\dfrac{E_t}{E}\left(1-\dfrac{\varepsilon_e}{\varepsilon_t}\right), & \varepsilon_e\in[\varepsilon_t,\xi_t\varepsilon_t],\\ 1-\dfrac{E_c}{E}\left(1-\dfrac{\varepsilon_e}{\varepsilon_c}\right), & \varepsilon_e\in[\xi_c\varepsilon_c,\varepsilon_c],\\ \alpha, & \varepsilon_e<\xi_c\varepsilon_c,\quad \varepsilon_e>\xi_t\varepsilon_t\end{cases}$$

Figure. 27.3-11. Relationship between the parameter P and the reduced strain ε_e (*a*) and the corresponding diagram $\sigma_e - \varepsilon_e$ (*b*)

If the *Take into account concrete degradation* checkbox is checked, the yield surface remains frozen, and elements of fracture mechanics are used to simulate the degradation of concrete caused by the formation and development of cracks. The relationship between the parameter P and the reduced strain ε_e is shown in Fig. 27.3-11 (a) and the approximate $\sigma - \varepsilon$ diagram obtained from the analysis is shown in Fig. 27.3-11 (b). Points C, U correspond to the respective points of the ECC diagram (Fig. 27.3-10).

Steepness of the descending branch in Fig. 27.3-11 (a) is determined by the E_t/E ratio in the tension area and by the E_c/E ratio in the compression area: $\xi_t = 1 - (1 - \alpha)E/E_t$ ($t \leftrightarrow c$).

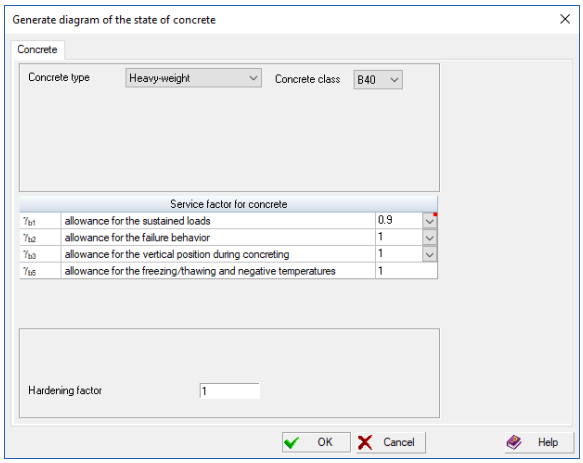


Figure 27.3-12. *The* **Generate a Concrete State Diagram** *dialog box*

Clicking the button 2012 with an icon of the selected reinforced concrete design code invokes the dialog box (Fig. 27.3-12), where you can specify the concrete class and service factors. Clicking the **OK** button will automatically fill in the data on the concrete properties (initial elastic modulus, ultimate compressive strength and ultimate tensile strength), taken from the codes. These data can be changed by the user.

The developers have deliberately left this possibility, because in many cases, for example, in the seismic analysis it is necessary to use additional service factors taking into account the dynamic hardening of the material.

Required data about the reinforcement behavior:

Table 27.3-4

Type of data	Comments
Type of diagram: • bilinear diagram • Exponential approximation of the bilinear diagram • Exponential approximation of the trilinear diagram	Exponential approximations of the bilinear and the trilinear diagrams are available only when the deformation theory of plasticity is used. If the plastic yield theory is selected, the symmetric bilinear diagram and von Mises yield criterion are assumed.
Elastic modulus of reinforcement	
Poisson's ratio of reinforcement	
Yield tensile strength of reinforcement	
Yield compressive strength of reinforcement	
Relative hardening (softening) modulus in the tension area for the reinforcement	
Relative hardening (softening) modulus in the compression area for the reinforcement	
Ultimate strain for tensile reinforcement $\varepsilon_{ul,t}$	If $\varepsilon_e > \varepsilon_{ul,t}$, the instantaneous rupture of the rebar is modeled. In order to disable the rupture of the rebar specify $\varepsilon_{ul,t} = 1000$ (or any other large positive number).
Ultimate strain for compressive reinforcement $\varepsilon_{ul,c}$	If $\varepsilon_e > \varepsilon_{ul,c}$, the instantaneous rupture of the rebar is modeled. In order to disable the rupture of the rebar specify $\varepsilon_{ul,t} = -1000$ (or any other large positive number).
Take into account the Bauschinger effect (plastic yield)	This option is recommended for steel reinforcement and relative hardening greater than zero.

It should be noted that once the reinforcement rupture occurs, the iterative solution of the nonlinear static problem usually begins to diverge. The computational procedure for nonlinear dynamic analysis is more stable and usually enables to obtain a solution for multiple statically indeterminate systems. **Therefore, if it is fundamentally important to take into account the reinforcement rupture, we recommend to solve a dynamic problem instead of a static one with a load slowly varying in time and the dissipation parameters somewhat lower than those at critical damping.**

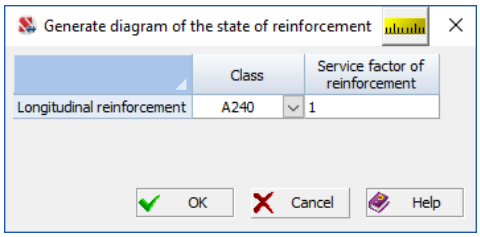


Figure 27.3-13. *The* **Generate a Reinforcement State Diagram** *dialog box*

Clicking the button 2012 with an icon of the selected reinforced concrete design code invokes the dialog box (Fig. 27.3-13), where you can specify the class of longitudinal reinforcement and the service factor. Clicking the **OK** button will automatically fill in the data on the reinforcement properties, taken from the codes. These data can be changed by the user.

The above information is not enough to perform a physically nonlinear calculation of the reinforced concrete elements. The reinforcement layout has to be specified for them (see Sec. 19.14), and these elements must be included in one of the groups of reinforced concrete structural members. In particular, information on the concrete covers will be taken from these groups.

In a numerical solution, shell finite elements are divided into layers along the thickness (their default number is 20), plus there are reinforcement layers as well.

The cross-sections of bar elements are automatically divided into triangles. In the case of default meshing the area of each triangle does not exceed 5% of the total cross-sectional area. However, the **Specify the additional data for elements** operation (see Sec. 7.3) enables to change these data (number of layers and meshing density) for certain elements (to increase the accuracy of the calculation). For example, if you set the meshing density to 5 (five times greater than the default value) the generated triangles will have an area not greater than 1% of the cross-sectional area.

Shell finite elements used today cannot simulate the elastic subgrade behavior. However, this problem can be solved as follows. Mesh the foundation slab into finite elements of type 442 or 444, specify the properties of the material and the reinforcement, and copy the foundation slab with an offset convenient for working with a graphic preprocessor. Next, change the element types to 42 and 44, respectively, in the resulting copy, which accurately describes the geometry of the foundation slab, create a new type of rigidity, specify a small thickness (for example, 0.0001 m) and a small modulus of elasticity (for example, $0.001 \cdot E_c$, where E_c is the initial modulus of elasticity of concrete), and specify the parameters of the elastic subgrade. We can create a separate group of finite elements to be able to get a quick access to this thin "film" in the future. We then return the thin "film" to its initial position and merge the duplicate nodes. We have thus "glued" the thin "film" with the elastic subgrade to the foundation slab.

Accounting for Geometric Nonlinearity

If you have selected the analysis based on the plastic flow theory, the **Accounting for geometric nonlinearity** checkbox will be available.

You can simultaneously take into account both geometric and physical nonlinearity for bar finite elements of type 405 and 2D shell finite elements of types 442 and 444. It can be especially crucial for steel structural members. Since it is natural to use the plastic flow theory in the analysis of elastic-plastic behavior of steel structures, the current version of **SCAD** provides this option both for a homogeneous material and for reinforced concrete only for the plastic flow theory.

In the case of reinforced concrete the geometric nonlinearity is accounted both for concrete and reinforcement. The following expressions for membrane deformations are used for shell finite elements:

$$\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x} + \frac{1}{2}\left(\frac{\partial u}{\partial x}\right)^2 + \frac{1}{2}\left(\frac{\partial v}{\partial x}\right)^2 + \frac{1}{2}\left(\frac{\partial w}{\partial x}\right)^2 + z\kappa_x \\
\varepsilon_y &= \frac{\partial v}{\partial y} + \frac{1}{2}\left(\frac{\partial u}{\partial y}\right)^2 + \frac{1}{2}\left(\frac{\partial v}{\partial y}\right)^2 + \frac{1}{2}\left(\frac{\partial w}{\partial y}\right)^2 + z\kappa_y \quad , \\
\gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} + \frac{\partial u}{\partial x}\cdot\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\cdot\frac{\partial v}{\partial y} + \frac{\partial w}{\partial x}\cdot\frac{\partial w}{\partial y} + 2z\chi
\end{aligned} \tag{27.3.1}$$

Here u, v, w are displacements of the midplane points, κ_x, κ_y are changes in curvatures in the direction of the OX, OY axes of the local coordinate system of the finite element, and χ is the torsion of the midplane. These expressions differ from the von Karman equations, since they contain terms $\frac{1}{2}\left(\frac{\partial u}{\partial x}\right)^2 + \frac{1}{2}\left(\frac{\partial v}{\partial x}\right)^2$ in the first equation, $\frac{1}{2}\left(\frac{\partial u}{\partial y}\right)^2 + \frac{1}{2}\left(\frac{\partial v}{\partial y}\right)^2$ in the second and $\frac{\partial u}{\partial x}\cdot\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\cdot\frac{\partial v}{\partial y}$ in the third one, which is **critical for the simulation of the behavior of flexible steel structural members when using 2D shell elements.**

For a type 405 bar finite element, the expression for the longitudinal strain ε_x contains the Taylor expansion terms up to the eighth order, which makes it possible to simulate the initial postcritical behavior.

When interpreting the results, you should keep in mind that in this implementation we use Mindlin-Reissner plates and shells and Timoshenko bars to simulate the geometric nonlinearity instead of the Kirchhoff-Love plates and shells and Kirchhoff-Clebsch bars, which can affect the analysis results.

27.4. Control of the Nonlinear Analysis

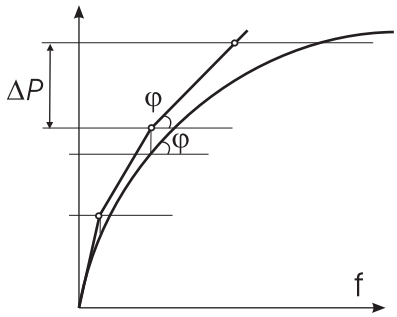


Figure 27.4-1. *Graphic illustration of the modifications of the incremental method (simple)*

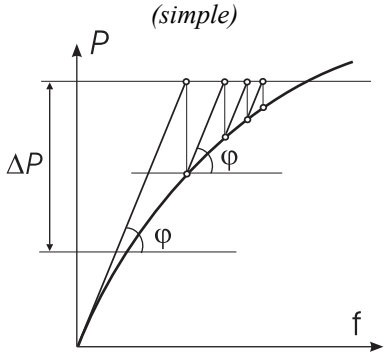


Figure 27.4-2. *Graphic illustration of the modifications of the incremental method (accounting for the residual)*

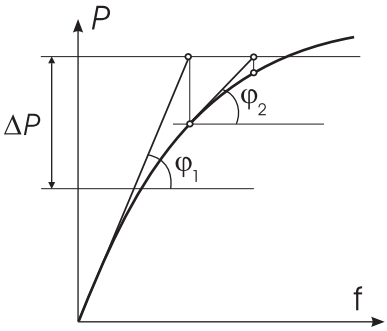


Figure 27.4-3. *Graphic illustration of the modifications of the incremental method (incremental-iterative)*

Nonlinear calculations are performed using the incremental method, which is based on the monitoring of the behavior of the system at relatively small load increments. A linearized system of governing equations is solved at each step for the current increment of the vector of nodal loads generated for the considered loading.

The calculation is focused on solving nonlinear problems in several modifications of the incremental method:

- simple incremental method;
- incremental method accounting for the residual;
- incremental-iterative method.

When the simple incremental method is used (Fig. 27.4-1), a linearized problem is solved in each step ΔP, and assuming that this solution is sufficiently accurate, a transition to the next step of the nonlinear loading occurs. The error of the solution of the nonlinear problem is not controlled; the number of steps is specified by the user. The stiffness of the linearized system (slope φ) is taken without accounting for the force residual.

The incremental method accounting for the residual provides the iterative refinement of the nonlinear load at the subsequent step ΔP by taking into account the residual in the equations of equilibrium. Iterations are performed with a constant value of the linearized stiffness matrix (Fig. 27.4-2), which was calculated at the beginning of the subsequent step (angle φ remains unchanged in the step).

Finally, when the incremental-iterative method is used (Fig. 27.4-3), the iterative refinement is performed at each step ΔP with the adjustment of the linearized stiffness matrix at each iteration (angle φ changes within the step).

Incremental process simulates the behavior of the system at the increasing (decreasing if the values of the loading factor are negative) loads acting on the system. It is assumed that all components of the loads related to the given loading increase (decrease) simultaneously in equal proportion.

The history of the nonlinear loading can be studied by specifying a sequence of individual variants of the loading. In this case the beginning of the application of the new nonlinear loading corresponds to the end of the previous one, i.e. the new loading is the continuation of the previous one. In particular, you can use the previously modeled loading, but with the negative value of loading factor, which enables to explore the full “loading-unloading” cycle. This technique enables to evaluate the accuracy of the calculation of the discrepancy between the initial and final states of the system.

The analysis of the stress-strain state of the structure taking into account the nonlinear effects is performed in those cases when at least one nonlinear element is specified in the design model. Different types of bars, three- and four-node shell elements taking into account the geometric nonlinearity, as well as the single- and two-node elements modeling unilateral constraints, and cable-stayed elements can be specified as such elements. Linear and nonlinear finite elements can be combined in one problem.

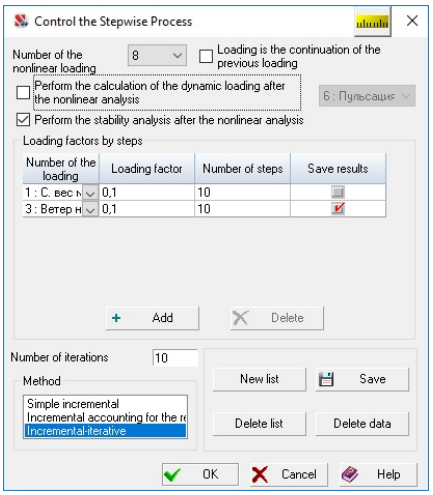


Figure 27.4-4. *The* **Control the Stepwise Process** *dialog box*

Data for the control of the analysis of the stress-strain state of the structure are specified in the **Control the Stepwise Process** dialog box (Fig. 27.4-4), which can be invoked from the **Design Model** section of the project tree (the **Modeling of Nonlinear Loads** operation). The dialog box contains a list of specified nonlinear loadings, a table for modeling the loading of the structure, a list for selecting the modification of the incremental method, a text field for specifying the number of iterations, and a number of control buttons.

Data describing one step are specified in each line of the table. It is necessary to perform the following actions to prepare the data:

- select the number of the linear loading from the list in the **Number of the loading** column;
- specify the loading factor for the current step in the **Loading factor** column as a multiplier to the value of the full load;
- specify the number of steps in the **Number of steps** column which should be performed with the specified factor (if the number is greater than one, the step will consist of several steps, each of which will be performed with the loading factor specified in the previous column);
- check the checkbox in the **Save results** column if you want to analyze the results of the current step (if a number greater than one is specified in the **Number of steps** column, only the resulting information is output without the results of the intermediate steps);
- repeat the above actions for each step of the analysis;
- select the necessary modification of the incremental method in the **Method** list;
- if the incremental-iterative method is selected, it is necessary to specify the limitation on the number of iterations in the **Number of iterations** field;
- click the **Save** button, and the prepared data will appear in the list of nonlinear loadings;
- if it is necessary to prepare several lists, click the **New list** button and repeat the above actions for other loadings;
- if the results of the calculation of the new list are the continuation of the loading of the previous one, check the **Loading is the continuation of the previous loading** checkbox after clicking the **New list** button.

There are also the following options:

- **Perform the calculation of the dynamic loading after the nonlinear analysis.** If this checkbox is checked, a list with a set of dynamic loadings of the model appears. After performing the steps of the nonlinear calculation the dynamic calculation selected in the list will be performed (for the system linearized after the nonlinear calculation).
- **Perform the stability analysis after the nonlinear analysis**. If this checkbox is checked, the stability analysis will be performed after performing the nonlinear calculation (for the system linearized after the nonlinear calculation). There must be data for performing the stability analysis of the linear system.

The **Delete list** button enables to delete the current list (its number corresponds to the number of the nonlinear loading). If you click the **Delete data** button, all the control information will be deleted, and the nonlinear calculation will be blocked.

Additional parameters described in the Chapter 11 are also used in the nonlinear calculation.

When performing the nonlinear calculation the relative residual (deformation and load) is output into the analysis log after each step of the iterative process.

When performing the linear calculation the nonlinear elements of the design model work as their corresponding linear elements.

27.5. Results of the Calculation

In addition to the parameters described in Chapter 12 the number of the step in the loading, at which the results of the calculation are saved, is specified in the line in the “Select loading” window (Fig. 27.5-1) at the graphical analysis of the results of the calculation (deformations, force diagrams etc.). The step is not specified for the results of the dynamic loadings, since the dynamic calculation for the linearized system is performed for the nonlinear loading after completing all the steps of the loading.

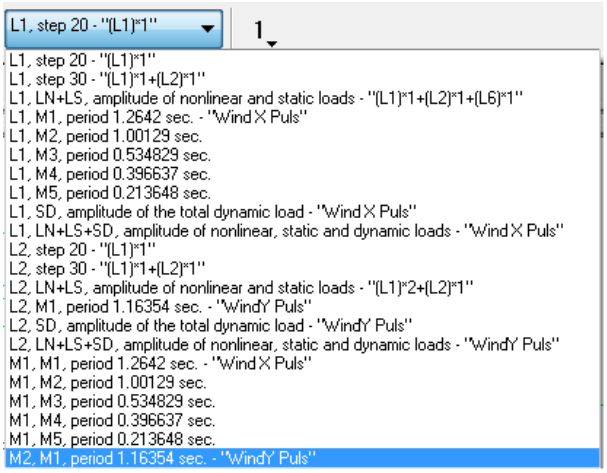


Figure 27.5-1. *The “Select loading” list for the nonlinear analysis*

In the case of the nonlinear calculation the following designations are used in the “Select loading” list (Fig. 27.5-1) for the identification of the output information:

LN + LS – amplitude from the nonlinear and static load (static wind) of the dynamic loading;

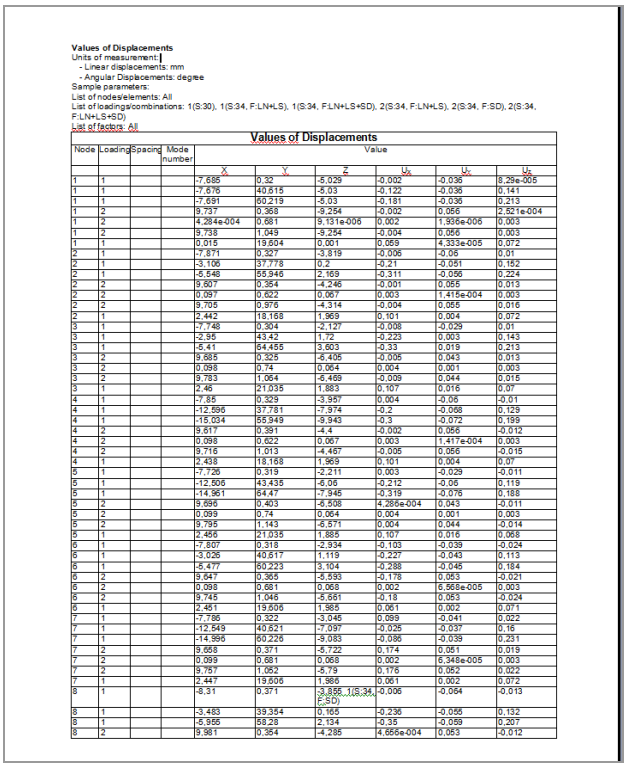
LN + SD – amplitude from the nonlinear and dynamic envelope of the components of the dynamic loading;

LN + LS + SD or **LN + LS – SD** – amplitude from the nonlinear, static and dynamic components of the dynamic loading;

LN + S1, LN – S1, LN + S2, LN – S2 ... – amplitude from the nonlinear, and dynamic envelope from the real and imaginary components of the dynamic loading;

LN + T1, LN + T2, ... – amplitude from the nonlinear and dynamic component at the given moment in time.

A fragment of the table of displacements for the nonlinear problem is given in Fig. 27.5-2. The table is generated by the **Documenter**.



Values of Displacements
Units of measurement:
- Linear displacements: mm
- Angular Displacements: degree

Sample parameters:
List of nodes/elements: All
List of loadings/combinations: 1(S:30), 1(S:34, F:LN+LS), 1(S:34, F:LN+LS+SD), 2(S:34, F:LN+LS), 2(S:34, F:SD), 2(S:34, F:LN+LS+SD)
List of factors: All

Values of Displacements

Node	Loading	Spacing	Mode number	Value					
				X	Y	Z	U_X	U_Y	U_Z
1	1			-7,685	0,32	-5,029	-0,002	-0,036	8,29e-005
1	1			-7,676	40,615	-5,03	-0,122	-0,036	0,141
1	1			-7,691	60,219	-5,03	-0,181	-0,036	0,213
1	2			9,737	0,368	-9,254	-0,002	0,056	2,521e-004
1	2			4,284e-004	0,681	9,131e-006	0,002	1,936e-006	0,003
1	2			9,738	1,049	-9,254	-0,004	0,056	0,003
1	1			0,015	19,504	0,001	0,059	4,333e-005	0,072
2	1			-7,871	0,327	-3,819	-0,006	-0,06	0,01
2	1			-3,106	37,778	0,2	-0,21	-0,051	0,152
2	1			-5,546	55,946	2,169	-0,311	-0,056	0,224
2	2			9,607	0,354	-4,246	-0,001	0,055	0,013
2	2			0,097	0,622	0,067	0,003	1,415e-004	0,003
2	2			9,705	0,976	-4,314	-0,004	0,055	0,016
2	1			2,442	18,168	1,969	0,101	0,004	0,072
3	1			-7,748	0,304	-2,127	-0,008	-0,029	0,01
3	1			-2,95	43,42	1,72	-0,223	0,003	0,143
3	1			-5,41	64,455	3,603	-0,33	0,019	0,213
3	2			9,685	0,325	-5,405	-0,005	0,043	0,013
3	2			0,098	0,74	0,064	0,004	0,001	0,003
3	2			9,783	1,064	-5,469	-0,009	0,044	0,015
3	1			2,46	21,035	1,883	0,107	0,016	0,01
4	1			-7,85	0,329	-3,957	0,004	-0,06	-0,01
4	1			-12,596	37,781	-7,974	-0,2	-0,068	0,129
4	1			-15,034	55,949	-9,943	-0,3	-0,072	0,199
4	2			9,617	0,391	-4,4	-0,002	0,056	-0,012
4	2			0,098	0,622	0,067	0,003	1,417e-004	0,003
4	2			9,716	1,013	-4,467	-0,005	0,056	-0,015
4	1			2,438	18,168	1,969	0,101	0,004	0,07
5	1			-7,726	0,319	-2,211	0,003	-0,029	-0,011
5	1			-12,506	43,435	-6,06	-0,212	-0,06	0,119
5	1			-14,961	64,47	-7,945	-0,319	-0,076	0,188
5	2			9,696	0,403	-6,508	4,286e-004	0,043	-0,011
5	2			0,099	0,74	0,064	0,004	0,001	0,003
5	2			9,795	1,143	-6,571	0,004	0,044	-0,014
5	1			2,456	21,035	1,885	0,107	0,016	0,068
6	1			-7,807	0,318	-2,934	-0,103	-0,039	-0,024
6	1			-3,026	40,617	1,119	-0,227	-0,043	0,113
6	1			-5,477	60,223	3,104	-0,288	-0,045	0,184
6	2			9,647	0,365	-5,593	-0,178	0,053	-0,021
6	2			0,098	0,681	0,068	0,002	6,568e-005	0,003
6	2			9,745	1,046	-5,661	-0,18	0,053	-0,024
6	1			2,451	19,506	1,985	0,061	0,002	0,071
7	1			-7,786	0,322	-3,045	0,099	-0,041	0,022
7	1			-12,549	40,621	-7,097	-0,025	-0,037	0,16
7	1			-14,986	60,226	-9,083	-0,086	-0,039	0,231
7	2			9,658	0,371	-5,722	0,174	0,051	0,019
7	2			0,099	0,681	0,068	0,002	6,348e-005	0,003
7	2			9,757	1,052	-5,79	0,176	0,052	0,022
7	1			2,447	19,506	1,985	0,061	0,002	0,072
8	1			-8,31	0,371	-3,856 (1(S:34, F:SD)	-0,006	-0,064	-0,013
8	1			-3,483	39,354	0,165	-0,236	-0,055	0,132
8	1			-5,956	58,28	2,134	-0,35	-0,059	0,207
8	2			9,981	0,354	-4,285	4,656e-004	0,053	-0,012

Figure 27.5-2. *Fragment of the “Displacements” table for the nonlinear problem*

The design combinations of forces should be calculated (see Chapter 18), so that the results of the nonlinear calculation could be used when selecting/checking steel and/or reinforced concrete structures.

Nonlinear Loading History

The control of the nonlinear loading history of the elements and nodes is performed with the help of the respective operation which can be invoked from the **Information on the Element** and **Information on the Node** dialog boxes respectively.

References to Chapter 27

1. Bezukhov N.I., Fundamentals of the Theory of Elasticity, Plasticity and Creep. — M.: High school, 1961. — 532 p.
2. Geniev G.A., Kissyuk V.N., Tyupin G.A. The Theory of Plasticity of Concrete and Reinforced Concrete. —M.: Stroyizdat, 1974. — 306 p.
3. Grigolyuk E.I., Shalashilin V.I., Problems of nonlinear deformation. — M.: Nauka, 1988.
4. Ilyushin A. A. Plasticity. Part one. Elastic-plastic deformations. — M.-L.: OGIZ, 1948. — 376 p.
5. Klovanych S.F., Finite Element Method in Nonlinear Problems of Engineering Mechanics — Zaporozhye: “IPO” Ltd, 2009. — 400 p.
6. Novozhilov V.V., The theory of elasticity.— Sudpromgiz, 1958.— 370 p.
7. Perelmuter A.V., Slivker V.I., Design models of structures and a possibility of their analysis.— M.:DMK Press, 2007. – 600 p.
8. Perelmuter A.V., SCAD Office. Calculation of guyed masts. — Kiev: SCAD Soft, Ltd. — 47 p.
9. Perelmuter A.V., Tur V.V., Whether we are ready to proceed to a nonlinear analysis at designing? International Journal for Computational Civil and Structural Engineering, 2017, **13**(3), 86-102.
10. Fialko S.Yu., Application of finite element method to analysis of strength and bearing capacity of thin-walled concrete structures, taking into account the physical nonlinearity — M.:ASV, SCAD Soft, 2018. — 256 p.
11. Bathe K. J., Walczak J.,Welch A., Mistry N. Nonlinear analysis of concrete structures. Computers & Structures, 1989, **32**, 563 – 590.
12. Criesfield M. A. Non-linear finite element analysis for solids and structures. Volume 1: Essentials. — John Wiley & Sons Ltd, Chichester New York …, 2000.
13. Criesfield M. A. Non-linear finite element analysis for solids and structures. Volume 2: Advanced topics. — John Wiley & Sons Ltd, Chichester New York …, 2000.

28. Amplitude-Frequency Characteristics

When installing the equipment and machinery which create periodic loads (machines with rotating parts, crank mechanisms, etc.) on the structures, the amplitude of a certain displacement or internal force depends both on the natural frequencies of the structure ω_i ($i = 1, \ldots, n$), and on the frequency θ of the exciting force

$$P = P_0 \sin(\theta t).$$

Peak value P_0 of the harmonic load is defined by the design assignment and depends on the design of the equipment which causes this load. If there are no accurate data, you can use the recommendations of SNiP 2.02.05-87 "Foundations of Machines with Dynamic Loads" and assume

$$P_0 = \mu \sum_{k=1}^{m} Q_k ,$$

where Q_k — weight of each rotor of the machine, and the coefficient μ is determined by the Table 28-1:

Table 28-1

Type of the machine	Coefficient μ
Turbomachines	0,20
Electric machines with the number of revolutions n_{rev} per minute: more than 750 from 750 to 500 less than 500	 0,20 0,15 0,10
Centrifuges with a rotor diameter d, m	$d\,(n_{rev}/1000)^2$
Centrifugal pumps	0,15
Smoke exhaust fans and fans	$0{,}8\,(n_{rev}/1000)^2$, but not less than 0,20

Due to the inaccuracy of determining the natural frequencies caused both by the approximate representation of the design model, and the rounding errors in the arithmetic calculations, as well as the possible instability of the angular frequency of excitation θ, reaching 10%, it is necessary to assess the behavior of the system when the frequency θ varies in a certain range from θ_1 to θ_2. It might be particularly important for the values θ, approaching one of the natural frequencies ω_i (near-resonant mode), that can occur when starting or stopping the machine. *The amplitude-frequency characteristic (AFC)* is a relationship between the peak value of a certain component of the stress-strain state (displacement, stress, internal force) and the frequency of the excitation θ . It is determined by the calculation of the system for the harmonic action at the frequencies of the exciting force changing in the specified range. The peak value of the force P_0 remains unchanged.

28.1. Preparation of the Initial Data

Information for analysis is specified in the **Special Initial Data** section of the **Project tree** with the help of the **Amplitude-Frequency Characteristics** operation. Moreover, this information can be specified in the preprocessor (**Special** tab). In this case, if the user has previously selected nodes, then the *Selected* item will appear in the list of groups of nodes (see below).

The initial data for the calculation of AFC include:

- a group of loads containing one or several harmonic loads P_0, applied to the nodes and/or elements of the design model;
- special data which are specified in the **Amplitude-Frequency Characteristics** dialog box (Fig. 28.1-1).

It should be noted that only synchronous (with the same frequency) and common-mode harmonic loads can be included in one group.

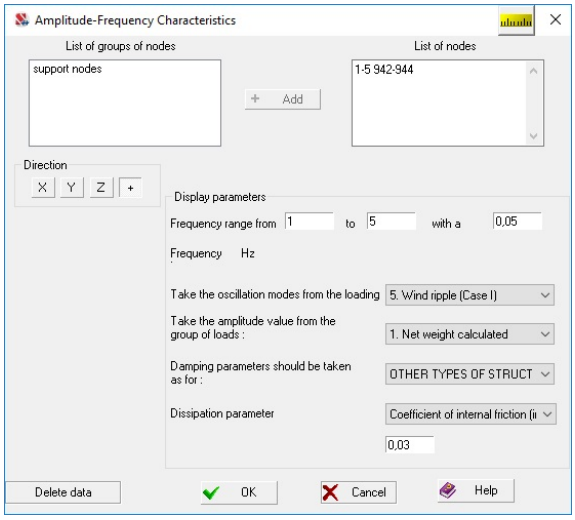


Figure 28.1-1. *The* **Amplitude-Frequency Characteristics** *dialog box*

The following data are specified in the dialog box:

- list of the groups of nodes for which the AFC are calculated, or the list of the numbers of these nodes;
- direction along which the displacement components are determined (X, Y, Z), or the indicator of the full displacement (+);
- value of the maximum allowable amplitude, which is indicated on the graph with the results of the calculation (you do not have to specify it);
- lower and upper frequency limits (frequency range), for which the calculation is performed;
- division step of the frequency range used when determining the amplitudes (when the spectrum of natural frequencies is dense, the step should be decreased). There should be 3–4 division steps between each pair of natural frequencies;
- coefficient of internal friction.

Moreover, it is necessary to select the number of the loading in the list where the natural oscillation modes were determined, and the group of loads.

The calculation is performed after preparing all the data and invoking the respective item in the project tree.

28.2. Results of the Calculation

The results of the calculation in the form of a set of AFC plots for the specified nodes are displayed in the **Amplitude-Frequency Characteristics** dialog box (Fig. 28.2-1).

The dialog box contains a list of nodes for which the amplitude-frequency characteristics are calculated. There are checkboxes next to each node number in the list, which enable to control the display of the plot for the respective node. The **All nodes** checkbox is used to enable/disable the display of plots for all nodes. A separate group of buttons **Direction** X Y Z + enables to select the direction (X, Y, Z or full — button with the + sign) of the AFC for which the drawing is performed.

If you want to narrow or broaden the range of the analyzed frequencies, check the respective checkbox and specify the limits of the new range in the **from** and **to** text fields.

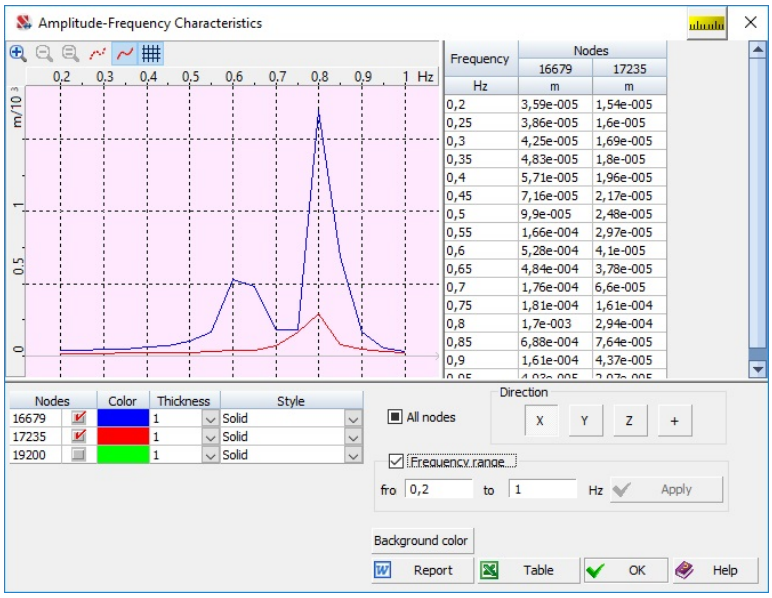


Figure 28.2-1. *Dialog box with the results of the calculation of the* AFC

The **Report** and **Table** buttons enable to generate a report in a word processor format or obtain an Excel table.

If you place the cursor in the field of the graph, the information field with the values in the selected point will appear next to the cursor which enables to explore the graph dynamically.

You can change the color, thickness and style of lines for each type of data, and the **Background color** button enables to change the background for the graphs.

29. Response Spectra

Response spectrum is a set of values of the maximum absolute or relative response accelerations of the linear oscillator at the action given by the accelerogram taking into account the natural damping frequency of the oscillator. They are usually used when designing nuclear power plants (see Sec.5.2 of Design codes for earthquake-resistant nuclear plants NR-031-01).

29.1. Seismic Analysis

We consider the oscillations of linear discrete systems with many degrees of freedom obtained from any continual or combined systems after applying to them the discretization procedure of the finite element method. A system of ordinary differential equations is solved

$$[M]\{\ddot{u}\}+[K]\{u\}=\{0\}, \tag{29.1.1}$$

where $\{u\}$ — displacement vector;

$[M]$ — mass matrix;

$[K]$ — stiffness matrix.

Forced oscillations of a discrete linear system with the resistance proportional to the velocity (so-called viscous damping according to the Voigt-Kelvin hypothesis) are described by a system of ordinary differential equations

$$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=\{F(t)\}, \tag{29.1.2}$$

where $[C]$ — energy dissipation matrix;

$\{F(t)\}$ — load vector.

In the case of the kinematic excitation portable inertial forces are used as the load and the system of equations (29.1.2) can be written as

$$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=-[M]\{I\}\ddot{x}_0(t), \tag{29.1.3}$$

where $\{u\}$ — relative displacement vector (for example, in the XOY coordinate system connected with the ground);

$\{I\}$ — vector the components of which are the cosines of the angles between the directions of the displacements in the coordinates and the ground acceleration vector;

$\ddot{x}_0(t)$ — ground acceleration.

The solution of equation (29.1.3) is found as an expansion by natural oscillation modes of the system (the so-called “modal superposition”)

$$\{u\}=\sum_{j=1}^{n}\{\Phi_j\}\Psi_j(t), \tag{29.1.4}$$

where n — number of the degrees of freedom of the system (of the eigenvalues and eigenvectors taken into account);

Φ_j — j-th natural oscillation mode of the discrete system;

$\Psi_j(t)$ — unknown time functions which have to be determined.

Suppose that the following condition holds for the dissipation matrix [C]:

$$\{\Phi_i\}^T[C]\{\Phi_j\}=\begin{cases}0, & i\neq j\\ 2\phi_i\omega_i\{\Phi_i\}^T[M]\{\Phi_i\}, & i=j\end{cases},$$

where ω_i — i-th natural frequency of the discrete system.

After substituting (29.1.4) in (29.1.3) and multiplying (29.1.3) by the vector $\{\Phi_i\}^T$ in order to find $\Psi_j(t)$ we obtain the following differential equation

$$\ddot{\Psi} + 2\phi_i\omega_i\dot{\Psi}_i + \omega_i^2\Psi_i = -D_i\ddot{x}_0(t)\,, \tag{29.1.5}$$

where

$$D_i = \frac{\{\Phi_i\}^T[M]\{I\}}{\{\Phi_i\}^T[M]\{\Phi_i\}}\ddot{x}_0(t)\,.$$

In order to determine the inertial loads on the structure it is necessary to know the accelerations of its points:

$$\{\ddot{u}_a\} = \{\ddot{u}\} + \{I\}\ddot{x}_0(t) = \sum_{i=1}^{n}\{\Phi_i\}\left(\ddot{\Psi}_j(t) + D_i\ddot{x}_0(t)\right) = \sum_{i=1}^{n}\{\Phi_i\}\ddot{\Psi}_{ja}(t)\,.$$

Seismic oscillations of discrete systems are described by the systems of differential equations with a more general form of the right-hand side:

$$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=-[M](\{\dot{I}_x\}\ddot{x}_0(t)+\{\dot{I}_y\}\ddot{y}_0(t)+\{\dot{I}_z\}\ddot{z}_0(t), \tag{29.1.6}$$

where $\ddot{x}_0(t)$, $\ddot{y}_0(t)$ and $\ddot{z}_0(t)$ — components of the analytical accelerogram.

If any of the components is not taken into account, the respective part of the load from (29.1.6) is excluded.

29.2. Accelerograms and Response Spectra

When analyzing the seismic resistance of the equipment it is necessary to determine the seismic inertial loads acting on it. The method of the separate consideration of the seismic oscillations of the building and equipment using the accelerograms and spectra calculated for the equipment attachment points is assumed.

The calculation is performed as follows:

- forced oscillations of the structure under the seismic action specified by the analytical accelerogram on the ground are determined (calculated);
- the acceleration variation laws for the selected points of the structure are determined;
- taking the accelerograms as an exciting action the forced linear oscillations of the linear non-conservative oscillators are calculated, and the relationships of the modules of their maximum absolute or relative accelerations and their natural frequencies and dissipation factors are determined.

Thus, equation (29.1.2) is solved for each considered point where:

- dissipation factor φ is determined;
- the load is the calculated exciting action from the analytical accelerogram;
- the sets of natural frequencies of the oscillators at the calculation of the response spectrum are determined and given in Table 29.2-1.

Natural frequencies of the calculated structure are added to the values of the frequencies of the non-conservative oscillators given in the Table 29.2-1[1]. This is done in order to allow for the possibility of resonance with them.

Table 29.2-1

Frequency range, Hz	Increments, Hz
0,2…3,0	0,10
3,0…3,6	0,15
3,6…5,0	0,20
5,0…8,0	0,25
8,0…15,0	0,5
15,0…18,0	1,0
18,0…22,0	2,0
22,0…34,0	3,0

[1] Seismic Analysis of Safety-Related Nuclear Structures and Commentary on Standard for Analysis of Safety-Related Nuclear Structures // *ASCE Standard*, Sept. 1986

The solutions are found for each specified oscillator on the whole range of the accelerogram action, and the maximum absolute value is selected, which is the response spectrum of the given point for the given accelerogram.

29.3. Input Data and Analysis of the Results

Information for analysis is specified in the **Special Initial Data** section of the **Project tree** with the help of the **Response Spectra** operation. Moreover, this information can be specified in the preprocessor (**Special** tab). In this case, if the user has previously selected nodes, then the *Selected* item will appear in the list of groups of nodes (see below).

The **Response Spectra** dialog box of the postprocessor (Fig. 29.3-1) contains a list of nodes for which the spectra have to be calculated (the numbers of nodes are specified in the **List of nodes** field with the "space" separator).

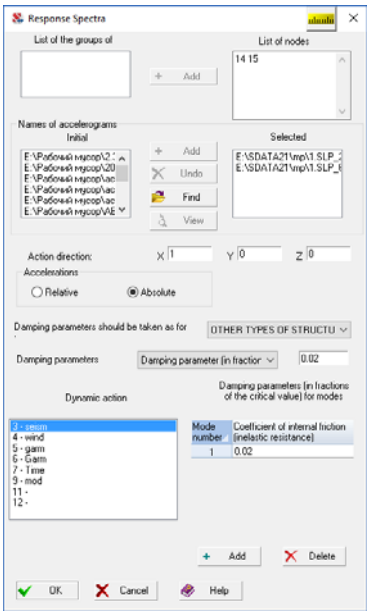


Figure 29.3-1. *The* **Response Spectra** *dialog box*

Accelerograms can be loaded into the **Initial** list by clicking the **Find** button. Names of the accelerograms for which the calculations of spectra have to be performed are selected in the **Initial** list and transferred to the list of accelerograms used in the calculation — **Selected**, by clicking the **Add** button. An accelerogram can be removed from the **Selected** list by clicking the **Remove** button. Moreover, in order to perform the calculation it is necessary to:

- specify the direction of the action;
- select the type of spectra (absolute or relative accelerations);
- specify the value of the dissipation factor in the respective field;
- if the dissipation factor is specified for each mode, its values are separated by spaces in the respective text field; if the number of the specified values of the factor is less than the number of the obtained oscillation modes, the last entered value applies to all the remaining modes;
- select the loading from which the natural oscillation modes are taken in the **Dynamic action** list.

If one (and only one) accelerogram is selected in the **Selected** list, the **View** button enables to preview this accelerogram.

Calculation is performed after preparing all the data and invoking the respective item in the project tree.

The results can be viewed and documented in the dialog box (Fig. 29.3-2), which can be invoked from the project tree.

The following procedure is recommended for generating the spectra:

- select the number of the node for which the spectra are generated in the **Node** list;
- select the line with the number of the loading in the **Loading** list;
- select the name in the **Accelerogram** list;
- specify the minimum and maximum values of the frequencies for which the spectrum is determined in the **from** and **to** fields of the **Frequency range**;
- use the checkboxes to specify the direction for which the response spectra are plotted (the SUM line enables to obtain the graph of the total acceleration).

The button enables to preview the selected accelerogram.

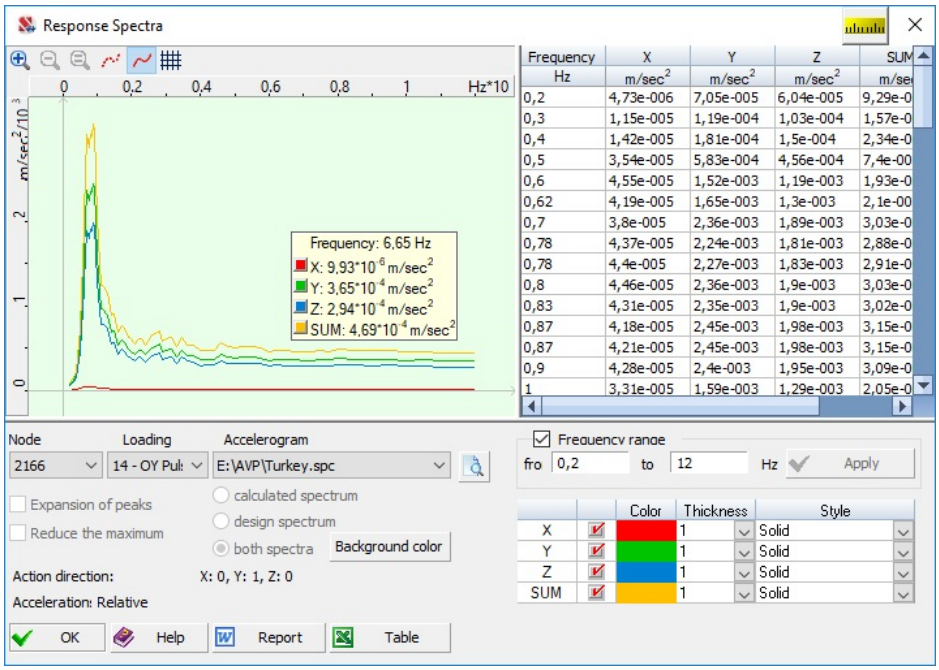


Figure 29.3-2. *The* **Response Spectra** *dialog box*

If only one plot is displayed on the screen, the software enables to perform the procedure of expanding the peaks and reducing the maximums (see. [1, Chapter 7]). The user can select which plots should be displayed on the screen (the calculated spectrum, design spectrum, or both spectra).

The **Report** and **Table** buttons enable to generate a report in a text format or obtain an Excel table.

If you place the cursor in the field of the graph, the information field with the values of the factors in the selected point will appear next to the cursor which enables to explore the graph dynamically.

You can change the color, thickness and style of lines for each type of data, and the **Background color** button enables to change the background for the graphs.

29.4. Preparation of the Accelerogram Files

Four standard sample accelerograms are supplied with **SCAD** . After the installation their files with the **.spc** extension can be found in the root directory of **SCAD Office**.

In order to use these files, they should be copied to the directory with the initial data. Other accelerograms should be saved in this directory as well.

Information on the specification of the accelerograms as the text files with the specified extension is given in Sec. 8.7. The **Accelerogram Editor**, which is described in Sec. 31.7, can be used for this purpose.

References to Chapter 29

1. A.N. Birbraer, Seismic analysis of structures. St. Petersburg. — Nauka. — 1998 — 255 p.

30. Influence Lines and Surfaces

Influence lines and surfaces are graphs the ordinates of which express the value of the displacement at a given point or the value of the internal force in an element depending on the position of the concentrated unit force of constant direction moving along the structure.

When plotting an influence line it is assumed that a unit force moves along a certain predefined load area. Possible positions of the unit force can be specified as the whole model, a group of nodes or a route.

30.1. Creating Routes

This section considers operations of creating and editing routes.

The route is given by a set of nodes arranged in a certain order. In order to make it more convenient to represent the routes in the graphics editor the nodes are connected by segments, which are not considered as individual objects and do not affect the characteristics of the design model. A route is an auxiliary element (type 1102), which can be generated when creating a design model, or attached to the model in the assembly mode.

Creating a Route and Its Snap to the Existing Model

The following procedure is recommended to create a route by snapping its nodes to the existing nodes of the design model:

- invoke the operation by clicking the **Route** button in the **Loadings** toolbar and select the input method in the drop-down menu — by selecting nodes in a certain order (**Create a new route**) or by selecting bar elements of the model coinciding with the route segments (**Create a new route from the selected bars**);
- if the first operation is selected, a dialog box will appear (see Fig. 30.1-1) where you have to specify the name of the route (the **Name** field) and close the dialog box by clicking the **OK** button;

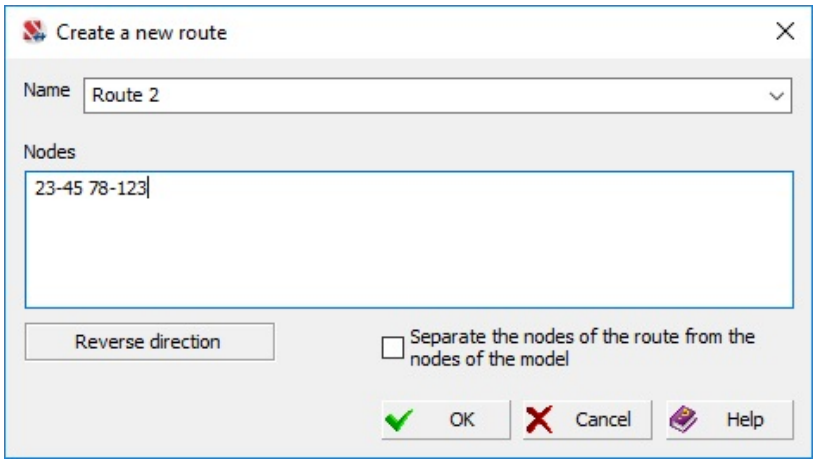


Figure 30.1-1. *The* **Create a New Route** *dialog box*

- select the nodes in a certain order with the help of a "rubber band" and click the **OK** button in the **Loadings** toolbar (rectangular or polygonal marquees can not be used, because they do not enable to specify the order of nodes);
- a similar procedure is used in the case of the **Create a new route from the selected bars** operation; specify the name in the **Create a New Route** dialog box, close it, select the bar elements and click the **OK** button in the **Model** toolbar (the bar selection order is not important, because it is assumed that the segments connecting the nodes of the route form a continuous broken line).

Peculiarities of the Implementation

If the selected bar elements do not form a continuous broken line, an error message will be generated. The user can decide to create several routes on the basis of one set of selected bars. They will be created automatically after a positive response to the respective request. Each route receives a name specified when invoking this operation supplemented by a number in the order of creation. The first created route receives the original name.

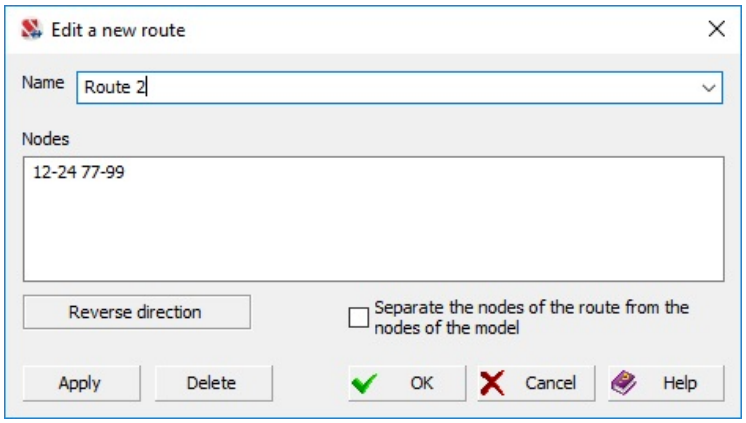


Figure 30.1-2. *The* **Edit a Route** *dialog box*

30.2. Preparing Data for the Calculation of Influence Lines and Surfaces

In order to calculate and display the influence lines and surfaces it is necessary to specify the initial data in the project tree — select routes, groups of nodes and/or specify the whole model as the load area (Fig. 30.1-1).

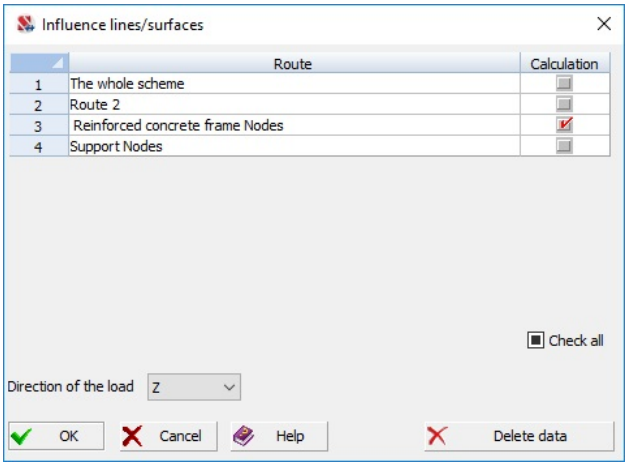


Figure 30.2-1. *The* **Influence Lines/Surfaces** *dialog box*

By default the calculation of influence lines and surfaces is performed assuming that unit forces are vertical (parallel to the global Z axis), but a different direction can be selected as well when specifying the initial data.

30.3. Calculation

In order to perform the calculation of influence lines and surfaces select the *Influence lines and surfaces* item in the project tree (in the *Analysis* section). A simple linear calculation has to be performed first. It should be noted that the calculation with the help of the iterative solver is inappropriate in this case, therefore **SCAD** blocks it if the iterative method is selected.

Forces/stresses in the centers of elements are used in the calculation.

30.4. Graphical Display of the Results of the Calculation of Influence Lines and Surfaces

The **Displacement influence surfaces** (Fig 30.4-1) and **Force/stress influence surfaces** (Fig. 30.4-2) modes of the **Postprocessors** toolbar is used for the graphical display of the results of the calculation of influence lines and surfaces.

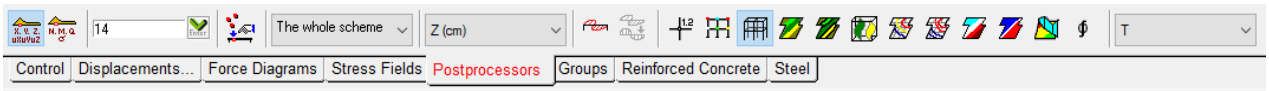


Figure 30.4-1. *The toolbar of the* **Displacement influence surfaces** *mode*

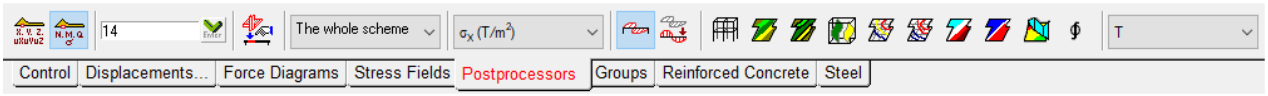
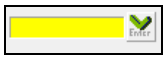


Figure 30.4-2. *The toolbar of the* **Force/stress influence surfaces** *mode*

The toolbars contain the following functions:

 — specify node/element number;

— select node/element with the mouse;

— display influence lines in bars;

— change the display axis of graphs;

— output values of displacements in nodes;

— color indication of the displacement value in nodes;

— display design model.

— display isofields;

— display isofields and isolines;

— display isosurfaces;

— display isolines;

— display isolines with a constant step;

— display isofields with a constant step;

— display isofields with the selected value;

— plot graphs along a secant line;

— calculate areas for influence lines and volumes for influence surfaces.

In order to perform the graphical analysis of the results it is necessary to:

- select a load area (route, group of nodes, or the *Whole model* item) in the respective drop-down list;
- select a node (in the **Displacement influence surfaces** postprocessor) or an element (in the **Force/stress influence surfaces** postprocessor), which will be analyzed; it can be done with

the mouse, by invoking the operation ( ), or by specifying the number in the text field , and pressing the **Enter** key in the keyboard (or clicking the **OK** button next to the text field);

- select the considered factor in the drop-down list (a displacement direction in the case of the displacement influence surfaces or a force factor in the case of the force/stress influence surfaces).

Then the options of displaying graphs (on bars or routes), isofields/isolines/isosurfaces on plate and solid elements will become available. If a route has been selected as a load area, only graphs can be displayed on the route. Most operations are similar to those used for the analysis of the stress-strain state.

The filters button enables to display the numerical values of the factor on isofields and isolines. The values can be displayed in the centers of elements and in nodes. In order to obtain the values in nodes it is necessary to activate the respective option in the **Environment Settings** dialog box (it can be invoked from the ***Settings*** menu). Since a unit force (equal to 1 N) leads to very small values of displacements/forces/stresses, which are inconvenient to analyze, you can specify the value of the "unit force" — you can select, for example, an item T in the respective drop-down list and obtain the digitization assuming that a force of 1 ton was applied in each node.

The operation enables to calculate positive and negative values of areas for influence lines and volumes for influence surfaces. The results are displayed in the dialog box (Fig. 30.4-3)

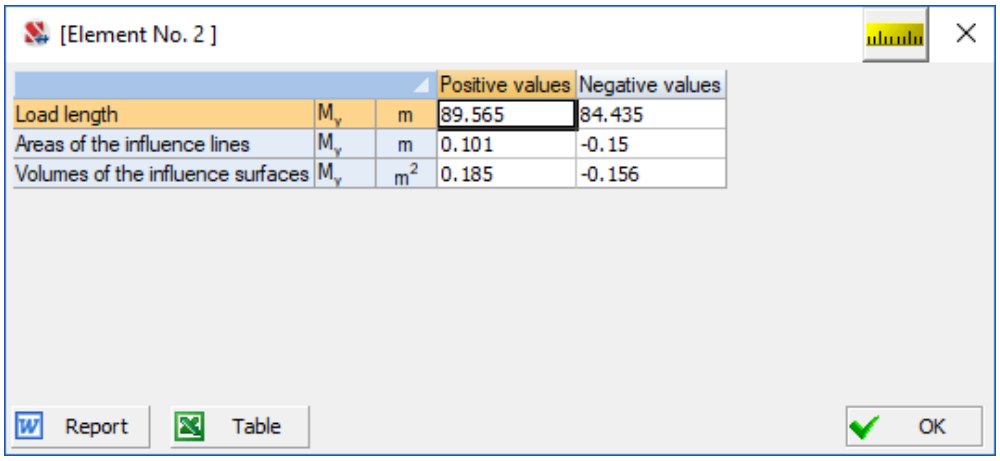


Figure 30.4-3. Calculated values of areas and volumes of influence lines and surfaces.

31. Centers of Rigidity

31.1. Determination of the Center of Rigidity

In some cases, it is necessary to determine the center of rigidity of a building. The concept of "center of rigidity" is used, for example, in seismic analysis. If the center of rigidity does not coincide with the center of mass, torsional oscillation modes appear. Some building frame design guides recommend arranging vertical diaphragms so that the center of rigidity is close to the common center of mass.

A center of rigidity of a storey can be considered in buildings where space-planning design varies in height. The center of rigidity of a storey is defined as a point that has the following property: if the resultant of all horizontal forces passes through it, then only the translational displacement of the floor disk occurs and there is no torsion.

In accordance with the recommendations of "Instructions for Determining the Design Seismic Load for Buildings and Structures" (Moscow, Gosstroyizdat, 1962) coordinates of the center of rigidity are determined by the formulas of the following type

$$x_0 = \frac{\sum_{i=1}^{n} G_{xi} x_i}{\sum_{i=1}^{n} G_{xi}}; \quad y_0 = \frac{\sum_{i=1}^{n} G_{yi} y_i}{\sum_{i=1}^{n} G_{yi}},$$

where G_{xi} and G_{yi} are rigidities of vertical elements of a storey (walls, columns, pylons etc.) in the direction of the X and Y axes, respectively, and x_i, y_i are coordinates of the elements' centers.

31.2. Input and Calculation

Information for analysis is specified in the **Special Initial Data** section of the **Project tree** with the help of the **Centers of Rigidity** operation. Moreover, this information can be specified in the preprocessor (**Special** tab). In this case, if the user has previously selected elements, then the *Selected* item will appear in the list of groups of elements (see below).

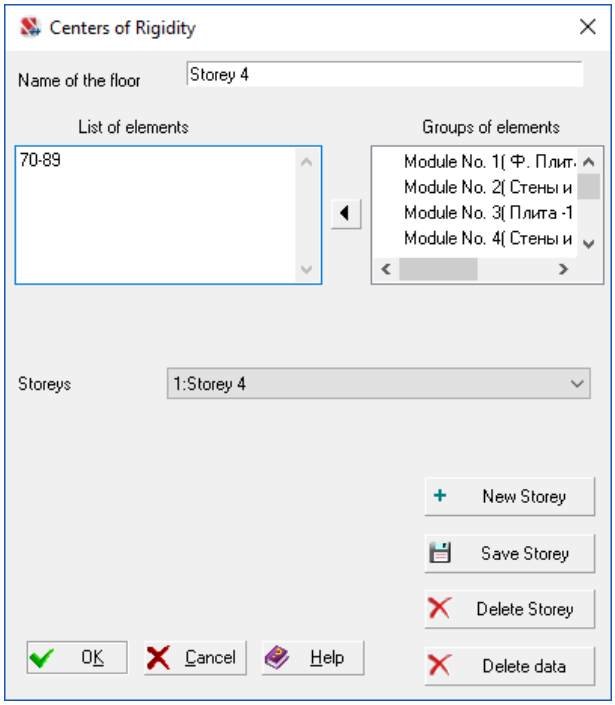


Fig. 31.2-1. **Center of Rigidity** dialog box

The initial data for calculating the center of rigidity are lists of elements that make up each conventional storey. The data is input in the **Center of Rigidity** dialog box (Fig. 31.2-1).
The following data input procedure is recommended:

- click the **New storey** button;
- specify the name of the storey in the respective field;
- specify the numbers of elements belonging to the current storey in the **List of elements** field. If the groups of elements including the elements of the storey were prepared in advance, select the names of these groups and transfer them (as a set of elements) from the respective field into the **List of elements** field;
- after specifying the list of elements of the current storey, save it by clicking the **Save storey** button;
- repeat the above actions for other storeys.

In order to delete the information on the storey, select its name in the **Storeys** list and click the **Delete storey** button.
The **Delete data** button is used to delete all the initial data prepared for calculating the center of rigidity.

If the FORUM preprocessor was used when preparing the initial data, groups created in the preprocessor can be used as the groups of elements of each storey.

31.3. Analysis of Results

This operation enables to determine the X, Y and Z coordinates of the center of rigidity for each storey. The **Determine the Center of Rigidity** dialog box appears on the screen after invoking this operation, which provides a table with the list of storeys and coordinates of the center of rigidity.

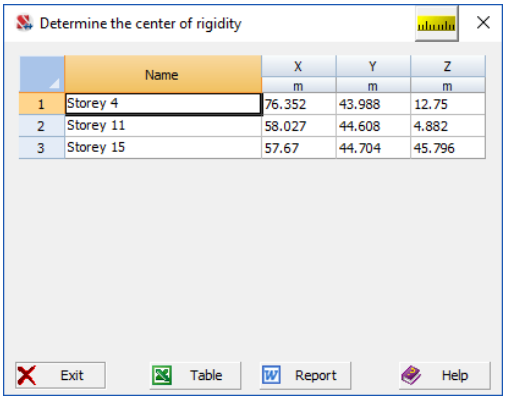


Figure 31.3-1. *The* **Determine the Center of Rigidity** *dialog box*

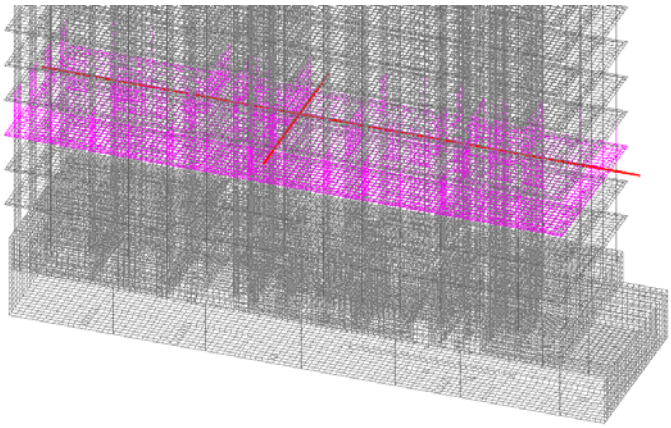


Figure 31.3-2. *Design model with the displayed center of rigidity*

If you select a line with the name of the storey, two lines with the center of rigidity of the given storey at their intersection will be displayed on the design model. Simultaneously elements of the selected storey will be “highlighted” in purple in the model.

32. Data Archiving Language

The possibilities of modern computer technology, advanced interactive computer graphics and fast numerical methods for solving systems of linear equations have allowed to create design models containing tens of thousands of nodes and elements, and, hence, to increase the dimensions of the solved problems by dozens of times in comparison with the earlier versions of **SCAD**. It obviously becomes problematic to use the language tools for describing the parameters of such design models. However, taking into account the fact that there are a large number of projects in the archives, the design models of which have been prepared long ago and described in the text format, all modern versions of the software enable to import the initial data in this format. The user can prepare some data on the structure as a text file, import it, and supplement the missing data with the help of the **SCAD** graphical preprocessor. It should also be noted that a text file enables to use the additional features of the software which are not yet implemented in its graphical preprocessor.

32.1. Export and Import Text Files with the Initial Data

The latest versions of **SCAD** have the options of not only importing but also exporting the descriptions of the design model in the text format, which enables to transfer the initial data into the ***archive*** in the form available for the control. The syntax of the data archiving language was changed starting with version 11.1. This is due to the fact that the previous version was created in the late 70s and now it can no longer describe all the peculiarities of the specification of the design model.

The file of the initial data must have a **.txt** extension in the text format, and, as before, it can be created in the text editor. For the continuity of the versions the input of the initial data in the text format remains in the previous edition of the syntax[1].

It should be noted that this format of the program supports the “bottom-up” compatibility with the previous versions, i.e. the initial data prepared for **SCAD** of the previous versions can be imported into the latest versions of the software, but the direct import of data (without correcting) from the latest versions into the previous ones is not guaranteed. In order to obtain a text format of the previous versions it is necessary to first save the project of version 21.1 as a project of version 11.7, and only then save it in the text format in the version 11.5 or 11.7.

This chapter provides the description of the architecture and syntax of the archiving language which enables to correctly and briefly describe the design model, its topology, constraints and loads, use the specific features of the finite element library, etc.

A design model is an idealized model of a structure (see Sec. 2). When the text input is used, it is necessary to divide the model into finite elements, and make the drawing; number the nodes; specify the support nodes and create the respective constraints in them (restrain the displacements for some degrees of freedom in the node or limit the displacements of the node by finite elements which model the constraint); specify the FE type and the rigidity type for the elements. The numbering of nodes and elements is necessary for the determination of the sequence of the specification of the initial information in the language of archiving and reading the counter results. Finite elements which have the same stiffness properties are combined into the rigidity types the numbers of which should be output for the element in the drawing.

A design model is described in the right-hand Cartesian coordinate system X, Y, Z, which is considered to be global. Not only Cartesian, but also cylindrical, spherical, toroidal, and other coordinate systems can be used for the specification of the coordinates of nodes. It should be noted that the values of the displacements of nodes are always output in the global coordinate system.

The position and values of the weights of the masses of the structure and other dynamic parameters are indicated on the design model at the calculation for the dynamic actions.

In the general case each node of the model has 6 degrees of freedom which are assigned the following numbers: 1, 2 and 3 — linear displacements along the X, Y and Z axes respectively; 4, 5 and 6 — rotations about the X, Y and Z axes. Some types of finite elements have up to 15 degrees of freedom.

[1] See Chapter 25 **Data Archiving Language** in the previous editions of “SCAD Office. SCAD system” 2004-2008.

Boundary conditions in the design model can be specified directly (restriction of displacements: linear — along X, Y, Z; angular — about U_X, U_Y, U_Z in any combinations in the Cartesian coordinate system). They can also be specified with the help of the constraints of finite rigidity or with the help of the special FE, which enable to impose the constraints in the directions of the axes of the local coordinate systems.

Static actions (see Sec. 8.1) are specified as concentrated forces and moments in the nodes of the design model (nodal loads) in the directions of the global coordinate system, and also on elements as concentrated and distributed forces and moments in the directions of the local or global coordinate systems (see Sec. 3).

Dynamic actions (see Sec. 8.7) are specified as nodal loads. Masses of the structure are specified as the self-weight of structures, equipment, etc.; both local and nodal loads can be used.

Different systems of measurement units can be used when specifying the initial data. The following units of measurement are the base ones:

- **L** — lengths (default value — meters);
- **F** — forces (default value — ton-force);
- **s** — cross-sectional dimensions of bars (default value — centimeters);
- **t°** — temperatures (default value — degrees Celsius).

Units of measurement of other values are derivatives of the base ones.

Time measurement units — seconds.

32.2. Architecture and Syntax of the Archiving Language

Initial information is divided into separate documents in accordance with Table 32.2-1.

Summary table of the documents of the initial data. Table 32.2-1

Name	Document number	Character of the information
Header	0	General information: code, index of the system type, division of elements for the output of forces, coordinate system, etc.
Elements	1	FE type, rigidity type and node numbers for each element
Boundary conditions	2	Number of the bar FE, number of its node (1 — beginning, 2 — end) and number of the degree of freedom in the direction of which the constraint is removed
Stiffness properties	3	Set of the stiffness properties for the determination of each type of rigidity
Coordinates	4	Three coordinates of each node (Cartesian, spherical, cylindrical, etc.)
Constraints	5	Numbers of nodes and numbers of the degrees of freedom in the direction of which the displacements are restrained by the external constraints or the displacements are merged (declared identical)
Characteristics of loadings	6	Description of the loading, its name, type and dynamic data
Loadings	7	Specification of the loads in the loadings
Characteristics of groups of loads	9	Specification of the names of the groups of loads and some of their additional characteristics
Groups of loads	10	Specification of the loads in the groups
Modules in **FORUM**	12	Lists of elements included in the **FORUM** modules
Variations of models	14	Lists of models included in the variation of models
Modeling the loading of the structure	16	Modeling the loading of the structure at the nonlinear analysis
Prestressing	17	Specified prestressing in finite elements
Local coordinate systems of nodes	19	For the specification of constraints in the local coordinate systems of nodes
Local coordinate systems of the load	20	For the specification of the load in the local coordinate systems of the nodes
Grid lines	21	Name and snap of grid lines
Characteristics of the nonlinear material	22	Diagrams of nonlinearity etc.
Contours	27	Name of the contour and the list of nodes belonging to it
Structural members and their groups for checking the steel structures	28	Parameters of the structural member and the list of its constituent finite elements, lists of finite elements forming a group

Name	Document number	Character of the information
Unification when checking and selecting the rolled steel sections	29	Names of the unified groups and the lists of finite elements included in them
Results of selecting the rolled steel sections	30	List of sections suggested on the basis of the results of the selection
Rigid inserts	31	Values of the rigid inserts
Position of the principal axes of inertia	32	Rotation angles of the principal or coordinate axes of inertia of the bar elements, position of the orthotropy axes for the plate and solid elements
Coordinate systems for calculating stresses and forces	33	Local coordinate systems for the calculation of forces and stresses in the plate and solid elements
Subsoil parameters	34	Subsoil parameters for all types of elements
Combinations of loadings	36	Coefficients with which the loadings are included in the combination
Stability analysis	37	Types of the analyzed characteristics, parameters, and a list of loadings (combinations of loadings) for which the calculation is performed
Principal and equivalent stresses	38	Number of the type of analysis and number of the failure theory in accordance with which the stresses are calculated
Loads from the structural fragment	39	List of elements from which the loads on the fragment are taken, orientation angle of the fragment, list of nodes to which the loads are transferred
Groups of elements	47	
Groups of nodes	48	
Reinforcement of sections of r/c elements	53	Data for the generation: parameters of groups, lists of elements
Response spectra	59	
AFC	60	
DCF	61	
List of DCF elements	62	
Groups in DCF	63	
Unification in DCF	64	
Data on the progressive collapse	70	
Loads at the direct integration	71	
Data on the erection	72	
Constraints at the erection stages	1005, 2005, …	Constraints at the erection stage the number of which is given by the first part of the figure
Boundary conditions at the erection stages	1002, 2002, …	
Subsoil parameters at the erection stages	1034, 2034, …	Subsoil parameters at the erection stage the number of which is given by the first part of the figure
Coefficients of variation of the rigidities at the erection stages	1073, 2073, …	Coefficients of variation of the rigidities at the erection stage the number of which is given by the first part of the figure
Specified reinforcement of plates	74	
Specified reinforcement of bars	75	
DCF for the nonlinear analysis	81	
List of DCF elements for the nonlinear analysis	82	
Groups in DCF for the nonlinear analysis	83	
Unification in DCF for the nonlinear analysis	84	

Information in the documents is mostly digital — real and integer numbers. Moreover, there are some alphabetic inclusions of type **R** (repetition statement) and identifiers — keywords of the data, for example, **AX** (rigid insert along the X_1 axis) etc.

The following rules are observed when specifying the initial information:

- each document has its number and is enclosed in parentheses;
- "(" — symbol of the beginning of the document;
- ")" — symbol of the end of the document;
- information between the symbols ") (" is ignored — it is considered as a comment;
- symbol "(" is followed by the number of the document separated from the subsequent information by the symbol "/" (slash);
- information in the documents is divided into lines. The number of the document is followed by the actual information. Line ends with the symbol "/" — slash;

- line starting with the symbol "*" is considered as a comment;
- each document can have its structure of lines the description of which is given in the respective section;
- numbers and the used text inserts placed in a line are separated from each other by one or several spaces;
- the integer and fractional parts of a number are separated by a period;
- text variables containing special characters have to be given in quotes;
- documents can be arranged in any order.

When specifying the initial information a number of approaches is allowed which considerably reduce its volume: indicators of repetitions when there are regularities, default specification principle which enables to imply the presence of certain information for a number of common cases without specifying it, etc.

Reduction of the Volume of the Initial Information When There are Regularities

Methods of the Reduced Record of the Lists

Input language allows the use of lists.

The volume of the information in the lists is reduced in the following way:

- if the numbers of the elements (or nodes) in the line of the document are arranged in order, the record of the type **1–N** is applicable (i. e. from the 1-st to the N-th), which does not exclude a simple enumeration;
- numbers of elements of the list 1, 5, 9, 13, 17 can be described with the help of the repetition statement **R** in the following way: 1 R 17 4; this record indicates a list which includes each fourth element (4) from the set of elements with the numbers from the first (1) to the seventeenth (17).

Repetition statement works in the lines of the document 0 in the following way: number before **R** is the first number of the list; number after **R** is the last number of the list (if it is a term of the progression); the third number is the step of the arithmetic progression.

Both methods of the description can be combined in the description of the list of objects, for example, 1–25 26 R 41 3 42 43. In the given example the list will include all elements with the numbers from 1 to 25 (1–25), as well as 26, 29, 32, 35, 38, 41 (26 R 41 3), 42, 43.

Repetition Statement of the Lines of Documents

A repetition statement placed in a single line is used:

R N1 N2 N3 : K1 K2 K3 K4.../

The line begins with a character **R**, which is followed by 1, 2, or 3 integers:

- N1 – number of duplicate lines;
- N2 – number of repetitions (number N2 can be absent, in this case the number of repetitions is equal to 1);
- N3 – number of the line from which the number of the duplicate lines is counted. Number N3 can be absent. In this case, the lines immediately preceding the repetition statement are repeated.

Steps of the repetitions for each component of a normal line of the document are indicated after the colon. Their number can not exceed the amount of numbers (positions) in the line of the document, in which the repetition statement is used, i.e.:

- K1 – step with which the numbers of the 1-st position of the line change;
- K2 – step with which the numbers of the 2-nd position of the line change, and so on.

Their values can have different magnitude and sign for each position.

If the number of specified values of Ki is less than the number of positions in a line, the line of repeaters is supplemented with zeros, for example:

5 10 20 40/		5 10 20 40/
R 1 4:0 0 1 1/		5 10 21 41/
	corresponds to the record:	5 10 22 42/
		5 10 23 43/
		5 10 24 44/

Repetition statements can be arranged in order. Number of the line and the number of lines counted by the repetition statements are determined for an expanded document (taking into account the previous repetitions).

Zero strings appear in the positions containing integer data when a repeated string is added to zero at any repetition step. The character of the integer data makes this rule convenient.

In the documents (for example, 4) with real numbers zeros will be added with a repetition step.

Examples of using the repetition statements are given in the description of the respective documents.

32.3. General Information on the Problem. Document 0 "Header"

The header document consists of separate lines each of which has its number and contains the calculation control parameters.

The unnumbered first line indicates the version of the language:

(0;Version=21/

Table 32.3-1

Line No.	Character of the information	Default value	Note
1	Code of the problem, information on the problem		Not obligatory
2	Index of the system type	Index of the system type 5	
3	Coordinate systems	Cartesian coordinate system	
4	Organization of the calculation of forces in the intermediate sections of bars and forces in the nodes of other FE	Forces are calculated in the beginning, middle, and end of the bar, for other FE — in the center of gravity	
8	Control of the calculation of the design combinations of forces	DCF are calculated for all elements of the model	When there is document 8
12	Unification of the sections of elements	No unification	Only when there is document 8
15	Specification of the DCF groups	One DCF group for all elements	Only when there is document 8
16	Allowing for crane and brake loadings in DCF	DCF includes not more than 2 crane and one brake loading	Only when there is document 8
17	Groups of elements		
18	Groups of nodes		
19	Characteristics of steel	Used in the checks and selection of rolled steel sections. Absent by default	
21	Data for stability		
22	Data for the calculation of principal and equivalent stresses		
23	Calculation parameters		
31	Names of groups of elements		
32	Names of groups of nodes		
33	Units of measurement	T, cm, m, degrees Celsius, sec.	

Some lines can contain only their number. Lines are filled as and when necessary, i.e. a document may consist of only one line, for example:

(0; Version=21/1; Frame/)

Most lines of a document have a list structure. Peculiarities of arranging the lists are given in the description of these lines. General rules of filling the lines are as follows:

- line starts with its number followed by the semicolon "**;**";

- the line number is followed by any number of lists separated from each other by a semicolon “**;**”;
- list can have a title separated from the rest of the list by a colon “**:**”;
- line ends with the symbol “/”;
- each line can be created as a separate zero document.

Code of the Problem (Line 1)

Code of the problem is a text expression. The first 128 characters of this expression are used to identify the problem. If the code of the problem consists of several words or contains special characters, it is given in quotes. In this case the space is allowed, for example,

1;"This is the name of the problem"/

Comments can be written in a line. If there are comments, the code of the problem and the comments are separated by a semicolon “**;**”.

Index of the System Type (Line 2)

The index of the system type is specified as a number and defines the set and number of the degrees of freedom in the nodes of the design model, and characterizes the peculiarities of its stress-strain state. In accordance with this index zero unknowns are removed from the system of equations.

The following numbers are used to specify the index of the system type (see Table 32.3-2): 1–5, 8, 9 and 11, 12. The specified index of the system type has to include all the necessary degrees of freedom for the finite elements used in the design model.

The main characteristics of the models depending on the value of their index. Table 32.3-2

Index of the system type	Position in space	Displacements in the node in the directions of the global coordinate system	Types of the degrees of freedom in the documents	Characteristics of structures
1	XOZ plane	Linear along X, Z	1, 3	Plane hinged bar system. Deep beam (plane stress problem) and a plane-strain problem of the theory of elasticity
2	XOZ plane	Linear along X, Z; rotation U_Y about the Y axis	1, 3, 5	Plane frame
3	XOY plane	Linear along Z; rotations U_X, U_Y about the X and Y axes	3, 4, 5	Grade beam, plate, plate on an elastic subgrade
4	Any	Linear along X, Y, Z	1, 2, 3	Spatial hinged bar system, massive body (spatial problem of the theory of elasticity)
5	Any	Linear along X, Y, Z; rotations U_X, U_Y and U_Z about the X, Y and Z axes	1–6	General type system. Enables to design the spatial structures of buildings, shells. Any structure with the indexes of system type 1–4 can be calculated with the index 5
8	Any	The following six analogues are added to the six degrees of freedom of the index 5: AX, AY, AZ and BX, BY, BZ	1–12	Structures from multilayer shallow shells and plates. Additional degrees of freedom take into account the effect of deformations of transverse shear and compression at the spatial behavior of the structure
9	Any	The following six analogues are added to the six degrees of freedom of the index 5: AX, AY, AZ and CX, CY, CZ	1–9, 12–15	Structures from multilayer shallow shells and plates. Additional degrees of freedom take into account the effect of the interlayer shear and curvature

Index of the system type	Position in space	Displacements in the node in the directions of the global coordinate system	Types of the degrees of freedom in the documents	Characteristics of structures
11	ROZ	Linear AX and Z along the R and Z axes respectively	10, 3	Axisymmetric problem of the theory of elasticity when the body and the load applied to it have the rotational symmetry
12	ROZ	Linear AX and Z along the R and Z axes respectively, and rotation angle CY	10, 3, 14	Axisymmetric problem of the theory of elasticity when the body and the load applied to it have the rotational symmetry

Coordinate Systems (Line 3)

Line 3 is introduced in those cases when it is necessary to transform the coordinates of nodes specified in the document 4. Cylindrical, spherical and other coordinate systems can be used.

Information in a line is specified as lists with the following structure:

<type of the transformation> [list of nodes][: numerical properties of the transformation]*;

The transformations of the coordinates specified in the line 3 can be also used when specifying the loads in these coordinate systems. If the list of nodes is not specified, the transformation is performed for all nodes.

The allowable types of transformations and the numerical properties necessary for them, as well as the specification of loads in these coordinate systems is described in detail in Sec. 32.13.

The transformations are performed consecutively for each list. You should remember that the information of this line is not saved in the project.

Calculation of Forces in the Intermediate Sections and Nodes (Line 4)

The number of sections in which the forces have to be calculated is specified in this line for the bar finite elements. For other finite elements it is indicated whether it is necessary to calculate the nodal forces and (or) calculate the stresses in the nodes of the elements.

A line consists of the lists of the type:

κ: <list of elements>;

For bars **κ** is the number of sections. If **κ** ≤ 2, the forces will be calculated only in the beginning and in the end of the bar.

For other finite elements **κ** can have one of the following values:

- **κ** = 0 — forces in the center of the finite element are calculated (by default);
- **κ** < 100 — forces in the center of the finite element and forces transferred by the element to the node (reactions) are calculated;
- **κ** = 100 — stresses (forces) in the points of junction of the FE with the nodes and in its center are calculated;
- **κ** = 101 — forces in the points of junction of the FE with the nodes, in its center, and forces transferred by the element to the node (reactions) are calculated.

If an element is included in several lists, the information from the last list where it is specified is valid for it.

If the line 4 is missing, it means that forces in the beginning, middle and end of the bars will be calculated, and only the forces in the center will be calculated for other FE.

Control of the Calculation of the Design Combinations of Forces (Line 8)

This line is filled when there is document 8 (see Sec. 28.11). The list of elements for which the design combinations of forces have to be calculated is specified in it.

* Hereinafter optional language elements are enclosed in square brackets.

Unification of the Sections of Elements (Line 12)

The numbers of the unified groups (see Chapter 18), sections for which the elements of the groups should be unified, and the type of unification are specified in this line.

A line consists of lists the titles of which indicate the type and group of unification, and the listing contains the numbers of elements included in the given group. The number of groups in each type is from one to nine.

Type and group of unification are specified by a two-digit number, where the first digit indicates the type of unification, and the second one — the unification group, for example, record

12; 21 75-86/

means: line 12, type of unification 1, unified group 2, elements included in this group have numbers from 75 to 86 inclusive. Any type of unification can be used for plate FE, i.e. possible 3×9 = 27 unified groups can be implemented.

If this line is missing, it means that there is no unification.

Organization of the Groups of Design Combinations of Forces (Line 15)

This line is filled when there is document 8 and in the case, when DCF coefficients specified in the document 8 are not identical for different groups of elements of the model (see Chapter 18).

The values of the coefficients with which the given loading will be involved in the selection of DCF are specified in the document 8 starting with the 10-th column. The numbers of the columns of the coefficients with which the group of elements is included in the DCF are specified in the title of the line 15. The numbers of these elements are listed after the title. The column 10 corresponds to the 1-st combination and when the line 15 is filled, it gets the number 1, column 11 gets the number 2, column 12 — 3 and so on.

The number of integers in the title indicates the number of the considered columns of the coefficients in the combinations. The first three numbers of the columns correspond to:

- the first main combination;
- the second main combination;
- the special combination.

One element can be included in not more than two groups of combinations.

The record: **15; 1 2 3: 181-184; 4 5 6: 508-511;/** means that the DCF the coefficients of which are in 10 (1), 11(2), 12(3) columns are determined for elements 181–184, and the DCF the coefficients of which are in 13(4), 14(5) and 15(6) columns are determined for elements 508–511.

If the number of the element is not indicated in any group of the combinations (or the line is missing), the DCF for this element will be performed for all the filled columns of the coefficients of the document 8.

Allowable Number of Crane and Brake Loadings Included in the Design Combinations of Forces (Line 16)

This line is filled only if there are crane loadings. Two numbers are specified in the line: the number of simultaneously considered crane loadings and the number of simultaneously considered brake loadings.

One brake and two crane loadings are considered simultaneously by default.

Data for Stability (Line 21)

A line is specified in the following form:

<parameters> [:<list of loadings,(combinations)>]

where parameters:

- type of analysis (0 – loadings, 1 – combinations);
- indicator of the calculation of coefficients;
- indicator of the calculation of modes;
- indicator of the calculation of unsupported lengths;
- upper search limit;
- lower search limit.

The list of loadings, (combinations) is separated from the parameters by a colon "**:**" and can be absent. In this case the calculations are performed for all static loadings.

Data for Calculating the Principal and Equivalent Stresses (Line 22)

The following parameters are specified:

- indicator of the calculation of principal stresses for loadings;
- indicator of the calculation of principal stresses for combinations of loadings;
- indicator of the calculation of equivalent stresses for loadings;
- indicator of the calculation of equivalent stresses for combinations of loadings;
- number of failure theory;
- constants for the failure theories (up to 4).

Parameters of the Calculation (Line 23)

This line contains the values of the parameters many of which are described in Chapter 11. Here is an example of filling of such a line:

23;LoadingCode=1;
SteelCode=1;
ConcreteCode=1;
SafetyCode=1;
TypeSolver=0;
NoForceInsert=1;
PrecessionGauss=12;
MaxQuantityPointTime=2;
MaxIterStabForm=20;
QuantityStepAddLoad=10;
CoeffIntervalGeom=5;

LoadingCode, **SteelCode**, **ConcreteCode**, **SafetyCode** parameters define design codes used for combining loads and actions, analysis of steel structural members, analysis of reinforced concrete structural members, and safety codes respectively. The following table shows the numbering used.

LoadingCode	SteelCode	ConcreteCode	SafetyCode
1 - SNiP 2.01.07-85 with modifications No. 2) 2 - SNiP 2.01.07-85 3 - DBN B.1.2-2-2006 6 - SP 20.13330.2011 9 - SP 20.13330.2016	1 - SNiP II-23-81 2 - SP 53-102-2004 3 - Eurocode 3 5 - SP 16.13330.2016 6 - DBN B.2.6-163:2010 7 - DBN B.2.6-198:2014	1 - SNiP 2.03.01-84 2 - SNiP 2.03.01-84 (with modifications for Ukraine) 5 - SNiP 52-01-2003 7 - SP 63.13330.2012 8 - SP 63.13330.2018	1 - GOST 27751 2 - GOST P 54257-2010 3 - DBN B.1.2-14-2009 4 - Eurocode 0 5 - GOST 27751-2014

Lists of the Deleted Nodes and Elements (Lines 31 and 32)

Contains the lists of the deleted nodes (31) and elements (32), which can be returned into the design model with the help of the respective restoration operations (see Chapter 6).

Units of Measurement (Line 33)

This line enables to specify the dimensions of the values in the initial data. The base units of the linear values are **meters**, of the force values – **tons**, of the temperature values – **degrees Celsius**. A special conversion factor is introduced for the description of the cross-sectional dimensions of bars.

The line contains the designations of the units and the conversion factors with respect to the base units. The order of the specification of the conversion factors is given below. The line has the following form:

$$\mathbf{33:\ L}\ \alpha_1\ \mathbf{s}\ \alpha_2\ \mathbf{F}\ \alpha_3\ \mathbf{t}\ \alpha_4/$$

Here **L**, **s**, **F**, **t** — designations of the units of measurement of the linear sizes, cross-sectional dimensions of bars, forces, temperature, respectively; α_1, α_2, α_3, α_4 — conversion factors.

32.4. Documents Describing the Design Model

Document 1. "Elements"

Each line of the document contains either the description of the finite element or that of the repetition statement.

The description of the element has the following form:

< type > < rigidity number> <list of the nodes of the elements> [Name=<name>]/

where

- element type is specified according to Table 3.2-1.
- rigidity number from document 3;
- list of nodes of the element in accordance with its description;
- name of the element in the case of its specification is saved under any transformations of the design model. It is given in quotes if there are special characters.

The repetition statement has the form described in Sec. 29.2.

Example:

```
(1/ 2 15 3 10/   r 1 5: 0 0 1 1/  51 21 12/  31 2 34 35 40 41 22 23 28 29/    21 1 11 14 18 19/ 12 1 12 15 27/ )
```

The example contains the information on the first 10 elements:

1-st – has type 2, rigidity number 15 from the document 3; 2-node element, numbers of the nodes in the element:
3 – number of the first node, 10 – number of the second node;

2–6-th – elements also have type 2 and rigidity number 15, and the numbers of nodes are 4, 11 for the 2-nd element and 8, 15 for the 6-th element respectively. They are obtained with the help of the repetition statement.

7-th – type 51, rigidity number 21; single-node element, node 12;

8-th – type 31, rigidity type 2, 8-node element, numbers of nodes: 34, 35, 40, 41, 22, 23, 28, 29;

9-th – type 21, rigidity type 1, 4-node element, numbers of nodes: 11, 14, 18, 19;

10-th – type 12, rigidity type 1, 3-node element; numbers of nodes: 12, 15, 27.

You should keep in mind that:

- each FE in the design model is assigned its number in the document 1, which is then used for the identification of the results of the calculation (forces, stresses, DCF, etc.) and it is used in all the postprocessors of the software for working with the element;
- the sequence of the description of nodes must correspond to their order described in Chapter 3 (see Table 3.2-1);
- the sequence of the description of nodes defines the direction of the local coordinate axes of the element X_1, Y_1, Z_1 for many types of FE.

Document 2. "Boundary Conditions"

This document is used to describe the boundary conditions in the nodes of the design model: releases of angular constraints (hinged connection) or releases of linear constraints (slider connection).

The line of the document has the following form:

<list of released constraints> Node = <node number> [Name=<name>]: <list of elements>

A combination of the released constraints at which an element can move with respect to the structure as a rigid body is not allowed.

Example:

```
(2/  1  Node=1 : 1-10/ 6  Node=2 : 1-10/
```

It means that the elements 1-10 have a slider connection in the X direction in the first node and the hinged connection along Z.

Document 4. "Coordinates"

The coordinates of the nodes of the design model are specified in the document. The type of the coordinate system is defined in the line 3 of the document 0 (Cartesian, cylindrical, spherical, toroidal). The Cartesian coordinate system is selected by default. Each line of the document 4 describes three spatial coordinates of the node. When filling the line 3 of the document 0 the coordinates are specified in accordance with the type of the used transformations in the selected coordinate system.

Since the lines of the document are not numbered, the coordinates are specified in the ascending order of the node numbers. If the line contains one or two numbers, the last missing coordinates are assumed to be equal to 0. Only two coordinates can be specified for the indexes of the system type 1 and 2 (line 2 of the document 0) which describe the XOZ plane. The name of the node can be specified using the **Name** identifier.

The repetition statement has the form described in Sec. 29.2.

Example:

(4/1.5 2.8 Name="Node at the level 1-1"/
2.3 3.7 8.8/ 5/r 1 5: 0 0 2/
	Coordinates of the first 8 nodes are described here:
	node 1 with the coordinates X=1.5m and Y=2.8m (Z=0) and the specified name;
	node 2 with the coordinates X=2.3m and Y=3.7m, Z=8.8m;
	node 3 with the coordinates X=5m (Y=Z=0);
	nodes 4-8 are obtained from the 3-rd one by successively rising it by 2 m along Z.

Document 5. "Constraints"[1]

This document is used to specify the boundary conditions. The line of the document has the following form:

<list of constraints > [Type = <type>] [Name=<name>]: <list of nodes> /

<list of constraints> is the list of the numbers of the degrees of freedom of the node on which the constraints are imposed.

The type defines the way in which the constraints are imposed:

- 0 – constraint of the respective degrees of freedom;
- 1 – combination of the degrees of freedom in the specified directions for all nodes of the list.

Constraints are specified in the global coordinate system and are designated in the following way:

1, 2, 3 — linear displacement along the X, Y and Z axes respectively;
4, 5, 6 — rotation angles about the X, Y, Z axes respectively;
7, 8, ..., 15 — additional degrees of freedom for the multilayer and other elements.

Example:

```
(5/ 1-3 : 1-10/  3 Type=1 : 11-15/)
```

Here the constraints are imposed in the directions 1, 2, 3 in the nodes 1-10; the displacements are merged in the direction Z in the nodes 11-15.

This document is not obligatory, for example, it can be missing in the calculation of the plates on an elastic subgrade.

32.5. Properties of Elements

Document 3. "Stiffness Properties"

The document is used to describe physical and mechanical characteristics and some geometric properties of finite elements. Since the stiffness properties are directly related to the type of the finite element, the detailed description of the input values is given in the respective sections of Chapter 3. The rules for preparing a document are given here.

[1] Documents with the numbers 1005, 2005, … describe the constraints at the erection stages (see Chapter 14).

The number of the rigidity type is indicated in the first position of each line of the document. This number is used in the column 2 of the document 1 when describing the FE. Then a certain set of characteristics is specified for each type of FE. Keywords can be used in the document for denoting the input characteristics.

The next positions contain the parameters specific to the elements of the given type. Both positional parameters (they always take a certain position in the line), and keyword ones, for the identification of which keywords are used, can be used in a document.

When specifying the characteristics in the document 3, a system of units described in the line 33 of the document 0 is used.

In the simplest cases the stiffness properties of the FE can be specified as the following sets of the positional parameters:

E, ν, δ — for plate elements;

E, ν, δ, C_1, C_2 — for plates on an elastic subgrade [1];

E, ν — for solid FE,

where E — elastic modulus; ν — Poisson's ratio; δ — plate thickness; C_1, C_2 — subsoil parameters.

The numerical description of the rigidities of the bar finite elements can be performed with the help of the positional parameters according to the rules given in Table 32.5-1.

Table 32.5-1

FE types	Stiffness properties	Designations
1, 4	EF	EF — axial stiffness;
2	EF EI_Y [GF_Y]	EI_Y, EI_Z — bending stiffness about Y_1 and Z_1 axes;
3, 4	EI_Y GI_{kr}	GI_{kr} — torsional stiffness;
5, 6	EF EI_Y EI_Z GI_{kr} [GF_Y GF_Z]	GF_Y, GF_Z — shear stiffness.

Moreover, the stiffness properties can be specified as a combination of keywords (identifiers) and their corresponding parameters. The identifiers, unless stated otherwise, can appear in any order. The order of the identifiers can be arbitrary for the types of rigidity of the bar elements. For other types of elements the identifier from the first line of the respective table is the first.

All the possible variants of the specification of the stiffness properties are given: in the Table 32.5-2 – for bar elements, in the Table 32.5-3 – for plates, in the Table 32.5-4 – for solid elements, in the Table 32.5-5 – for multilayer plates.

Table 32.5-2

Characteristic of bar elements

Contents	Identifier	Parameters	Notes
Numerical characteristics	no	EF EI_Y EI_Z GI_{kr} [GF_Y GF_Z]	Specified at the numerical and numerical and parametric specification of the stiffness properties. The amount of numbers depends on the type of the finite element according to Table 32.5-1. Follow immediately after the number of the rigidity type
Parametric sections	**S0**	E, b, h	E — elastic modulus Description and sizes of sections are given in the Table 3.3-3
	S1	E, b, h, b_1, h_1	
	S2	E, b, h, b_1, h_1	
	S3	E, b, h, b_1, h_1, b_2, h_2	
	S4	E, b, h, b_1, h_1	
	S5	E, b, h, b_1, h_1	
	S6	E, D, d	
	S7	E, b, h, b_1, h_1, h_2, h_3	
	S8	E, b_1, h_1, Y_1, Z_1, b_2, h_2, Y_2, Z_2, …	

[1] The specification of the rigid inserts, subsoil parameters, local coordinate systems in the 3-rd document is obsolete, but acceptable. It is better to use documents 31-34.

Contents	Identifier	Parameters	Notes
Variable cross-section	**VS**	<geometric properties of the end section>	Geometric properties at the beginning of the section are specified in S00–S08, and those at the end are specified in VS
Welded cross-sections	**S41**	E, b, t_f, h, t_w	E — elastic modulus. Description and sizes of sections are given in the Table 3.3-3
	S42	E, b, t, h, t_w, b_2, t_2	
	S43	E, b_{f1}, t_{f1}, h_w, t_w	
	S44	E, b_f, t_f, h_w, t_w	
	S45	E, b_f, t_f, h_w, t_w	
	S46	E, b, h, t_1, t_2	
Cold-formed cross-sections	**S51**	E, h, b, s, r	Description and sizes of sections are given in the Table 3.3-3. The elastic modulus and the calculated stiffness properties are specified using the parameter S198
	S52	E, h, b, s, r	
	S53	E, h, b, s, h_1, r	
	S54	E, h, b, s, c, r, α	
	S55	E, h, b, s, h_1, r, b_2	
	S56	E, h, b, s, r	
	S57	E, h, b, s, h_1, r	
	S58	E, h, b, s, h_1, r, α	
	S59	E, h, b, s, a, r	
	S60	E, h, b, s, h_1, r, a	
	S61	E, h, b, s, a, r	
	S62	E, h, b, s, h_1, r, a	
	S63	E, h, b, s, b_1, r, h_2, h_3, h_4	
	S64	E, h, b, s, h_1, h_2, h_3, h_4, b_1, r	
	S65	E, h, b, s, h_1, h_2, h_3, h_4, b_1, b_2, r	
Cold-formed bilt-up cross-sections	**S66**	E, h, b, s	Description and sizes of sections are given in the Table 3.3-3. The elastic modulus and the calculated stiffness properties are specified using the parameter S198
	S67	E, h, b, s, h_1, r	
	S68	E, h, b, s, h_1, r, b_2	
	S69	E, h, b, s, b_1, r, h_2, h_3, h_4	
	S70	E, h, b, s, h_1, h_2, h_3, h_4, b_1, r	
	S71	E, h, b, s, h_1, h_2, h_3, h_4, b_1, b_2, r	
	S72	E, h, b, s, h_1, r	
	S73	E, h, b, s, r	
Rolled steel profiles	**ST**	<section name> <line number>	Section from the *Full assortment of GOST profiles* (see Appendix)
	STZ	<catalogue name> <section name> <line number>	The **Appendix** contains the assortment of rolled profiles included in **SCAD**. The following records are equivalent: **STZ RUSSIAN <section name> <line number>** **ST < section name> <line number>**
Compound sections from rolled steel profiles	**STK**	<catalogue name> <section name> <line number> <layout code> <additional parameters>	The types of the compound sections are given in Fig. 3.3-2. The layout code and the additional parameters are specified in accordance with Table 3.3-4
Sections from **Section Builder**, **Consul**, **TONUS**	**S199**	E ν EF EI_Y EI_Z GI_{KR} GF_Y GF_Z Y_1 Y_2 Z_1 Z_2 Alfa	E — elastic modulus; ν — Poisson's ratio; EF, EI_Y, EI_Z, GI_{KR}, GF_Y, GF_Z — stiffness properties (see Table 32.5-1); Y_1, Y_2, Z_1, Z_2 — section core sizes; Alfa — rotation of the principal axes of inertia with respect to the grid lines
	FILE	<file name>	This file must contain the data from the builder

Contents	Identifier	Parameters	Notes
For cold-formed sections S51-S73	**S198**	E ν EF EI_Y EI_Z GI_{KR} GF_Y GF_Z Y_1 Y_2 Z_1 Z_2 Alfa	E — elastic modulus; ν — Poisson's ratio; EF, EIY, EIZ, GIKR, GFY, GFZ — stiffness properties (see Table 32.5-1); Y1, Y2, Z1, Z2 — section core sizes; Alfa — rotation of the principal axes of inertia with respect to the grid lines
Elastic modulus	**E**	E	
Poisson's ratio	**NU**	ν	The value 0.2 is taken for parametric sections by default and 0.3 – for the rolled steel sections
Element type	**KE**	<element type>	For specifying the element type. The set of parameters at the numerical and numerical and parametric description of the rigidities depends on its value (see Table 32.5-1)
Section core sizes	**Y**	$l_{Y,1}$ $l_{Y,2}$	See Fig. 3.3-10. Used for the selection of DCF at the numerical specification of the rigidities
	Z	$l_{Z,1}$ $l_{Z,2}$	
Shear	**GF**	no	Allowing for shear in the elements. It is necessary to specify GF_Y and GF_Z at the numerical description
Prestressing	**N**	N	In the nonlinear analysis, for example, the prestressing of the cable stay
Specific weight	**RO**	ρ	Specific weight
Name	**NAME**	<name of the rigidity type>	Name of the rigidity type. It cannot be longer than eight characters. If it contains special characters, it is given in quotes
Calculated additional characteristics	**SHIFT**	GI_{kr} GF_Y GF_Z	They are additionally calculated by the software when there is no rolled steel in the assortment
For cable-stayed elements	**VANTA**		Indicator of the cable-stayed element
Reduction factors	**CREI** **CREF**	C_{ei} C_{ef}	Reduction factor for bending stiffness Longitudinal stiffness reduction factor

Table 32.5-3

Characteristics of plates

Contents	Identifier	Parameters	Notes
Properties of the material	**GE, GEI**	E μ δ	For the isotropic, orthotropic, anisotropic materials respectively (see Sec. 3.4). *The number of the rigidity type of plates* ***must be followed*** by the description of the material
	GEO	E_1 E_2 ν_{12} ν_{21} G_{12} δ	
	GEA	E_1 E_2 ν_{12} ν_{21} G_{12} $\eta_{12,1}$ $\eta_{12,2}$ $\eta_{1,12}$ $\eta_{2,12}$ δ	
Additional properties of the material for plane-strain systems	**PD, PDI**	no	The analysis of the plane-strain systems (instead of the plane stress ones) for the isotropic (**PD** and **PDI**), transverse isotropic (**PDT**), orthotropic and anisotropic materials (**PDA**) respectively
	PDT	E_3 ν_{13} ν_{31}	
	PDO	E_3 ν_{13} ν_{31} ν_{23} ν_{32}	
	PDA	E_3 ν_{13} ν_{31} ν_{23} ν_{32} $\eta_{12,3}$ $\eta_{3,12}$	
	XNT	X_T Y_T Z_T	
Name	**NAME**	<name of the rigidity type>	Name of the rigidity type. It cannot be longer than eight characters. If it contains special characters, it is given in quotes
Reduction factors	**CRPL** **CRBL**	C_{pl} C_{bl}	Reduction factor for bending stiffness Reduction factor for membrane stiffness

Table 32.5-4

Characteristics of solid elements

Contents	Identifier	Parameters	Notes
Properties of the material	**OB**	E ν	Specification of the numerical characteristics of the isotropic (**OB**), transverse isotropic (**OBT**) and
	OBT	E E3 ν_{12} ν_{13} G_{13}	

Contents	Identifier	Parameters	Notes
	OBO	E1 E2 E3 ν_{12} ν_{21} ν_{13} ν_{31} ν_{23} ν_{32} G_{12} G_{13} G_{23}	orthotropic (**OBO**) materials (see Sec. 3.5). The number of the rigidity type of solid elements ***must be followed*** by the description of the material
Name	**NAME**	<name of the rigidity type>	Name of the rigidity type. It cannot be longer than eight characters. If it contains special characters, it is given in quotes

Table 32.5-5

Characteristics of multilayer plates

Contents	Identifier	Parameters	Notes
Indicator of the characteristics of multilayer plates	**GEM**	no	Follows immediately after the number of the rigidity type in the document 3. Its specification is ***obligatory***
Properties of the material	**L, LI**	$E^{(k)}$ $\nu^{(k)}$ $\delta^{(k)}$	For the isotropic, transverse isotropic, and orthotropic materials respectively (see Sec. 3.7). The characteristics of the materials of layers are specified right after the **GEM** identifier consecutively, starting with the lowest. There ***must*** be at least one identifier **L**, **LI**, **LT** or **LO**
	LT	$E^{(k)}$ $\nu_{13}^{(k)}$ $\nu_{31}^{(k)}$ $G_{13}^{(k)}$ $\delta^{(k)}$	
	LO	$E_1^{(k)}$ $E_2^{(k)}$ $E_3^{(k)}$ $\nu_{12}^{(k)}$ $\nu_{21}^{(k)}$ $\nu_{13}^{(k)}$ $\nu_{31}^{(k)}$ $\nu_{23}^{(k)}$ $\nu_{32}^{(k)}$ $G_{12}^{(k)}$ $G_{13}^{(k)}$ $G_{23}^{(k)}$ $\delta^{(k)}$	
Eccentricity	**DZ**	Δ	Specification of the eccentricity in the case when the coordinate surface does not coincide with the midplane of the element
Name	**NAME**	<name of the rigidity type>	Name of the rigidity type. It cannot be longer than eight characters. If it contains special characters, it is given in quotes

The examples of the description of the stiffness properties in the document 3 are given in the Table 32.5-7.

Table 32.5-6

Characteristics of unilateral constraints

Contents	Identifier	Parameters	Notes
Indicator of the characteristics of unilateral constraints	**OSD**	EF	Follows immediately after the number of the rigidity type in the document 3. Its specification is ***obligatory***
Direction of the constraint	**ORT**	ΔX, ΔY, ΔZ	Direction of the constraint if it is defined by two nodes of the element
Sign	**SGN**	±1	The constraint is disabled at compression/tension
Gap	**DL**	DL	The value of the gap after which the constraint is enabled/disabled
Prestressing	**N**	N	
Prestressing by displacing	**DZ**	DZ	

Table 32.5-7

Characteristics of the constraints of finite rigidity, elastic constraints, null elements and rigid bodies

Contents	Identifier	Parameters	Notes
Indicator of the characteristics	**SPRING**	EX, EY, EZ ...	Follows immediately after the number of the rigidity type in the document 3. Its specification is ***obligatory***. It is followed by the stiffness values. If there are no constraints, a zero value is specified
Prestressing by	**TYPE**	51, 55, 100 or 151	Type of the finite element.

Contents	Identifier	Parameters	Notes
displacing			151 – for null elements

Table 32.5-8

Characteristics of the peripheral elements of the plate

Contents	Identifier	Parameters	Notes
Indicator of the characteristics	**BED**	e1, e2	Follows immediately after the number of the rigidity type in the document 3. Its specification is ***obligatory***. It is followed by the stiffness values in accordance with the description of the element
Prestressing by displacing	**TYPE**	53 or 54	Type of the finite element

Table 32.5-9

Examples of the description of stiffness properties

Text of the document 3	Contents
1 **S0** 3.52e+06 30 40 **RO** 2.5 **NAME** "Column"/	Rectangular section 30x40cm, elastic modulus of the material 3.52e+06T/m^2, specific weight 2.5T/m^3
2 **STZ** RUS_OLD I2 10 **RO** 7.85/	Section from "*The old assortments (1926, 1932 years)*". I-beam 24a according to OST 16-1932 (10-th line of the assortment), specific weight — 7.85T/m^3
3 **GE** 3.06e+06 0.15 0.1 **RO** 2.5/	Rigidities of an isotropic plate with the thickness of 0.1m, lying on the elastic subgrade

Document 31. "Rigid Inserts"

The line of the document has the following form:

X_1 Y_1 Z_1 X_2 Y_2 Z_2 [Type=3]: <list of elements> /,

where

X_1 Y_1 Z_1 – vector defining a rigid insert at the beginning of the element;
X_2 Y_2 Z_2 – vector defining a rigid insert at the end of the element;
GroupElem=1 – type of the bar element;
Type – type of specification of the rigid inserts: 0 – Fig. 3.3-12, 3 - Fig. 3.3-13 when a rigid insert is specified in the global coordinate system.

Document 32. "Position of the Principal Axes of Inertia"

The line of the document has the following form:

<identifier> <data >: <list of elements> /

It enables to specify the directions of the principal axes of inertia of the bar elements for the specification of the correct position of the section of the element, and those of the plate and solid elements to take into account the orthotropy of the material.

The allowed identifiers and the data for them are given below.

Element type	Identifier	Data	Contents
Bars	**F**	F_K	Value of the angle in degrees or radians (see Fig. 3.3-6)
	FR	F_K	
	FK	X_T Y_T Z_T	Coordinates of the point (see Fig. 3.3-7 and 3.3-8) lying on the Y_1 or Z_1 axis
	ZK		
	FD	X_D Y_D Z_D	Vector defining the direction of the Y_1 or Z_1 axes

Element type	Identifier	Data	Contents
	ZD		
Plates	**EX**	X_D Y_D Z_D	Specification in the global coordinate system by the vector (**EX**) or by the coordinates of the point lying on this axis (**EXT**). The specified direction is projected on the plane of the element for the determination of the orthotropy X_0 axis. The Z_0 axis coincides with the Z_1 axis, the Y_0 axis is orthogonal to X_0 and Z_0, the system X_0, Y_0, Z_1 is right-hand
	EXT	X_T Y_T Z_T	
	EY	X_D Y_D Z_D	Specification in the global coordinate system by the vector (**EY**) or by the coordinates of the point lying on this axis (**EYT**). The specified direction is projected on the plane of the element for the determination of the orthotropy Y_0 axis. The Z_0 axis coincides with the Z_1 axis, the Y_0 axis is orthogonal to X_0 and Z_0, the system X_0, Y_0, Z_1 is right-hand
	EYT	X_T Y_T Z_T	
Solid elements	**VXY**	ΔX_1, ΔY_1, ΔZ_1, ΔX_2, ΔY_2, ΔZ_2	Specification by the vectors defining the directions of the X_0 and Y_0 axes
	VXZ		Specification by the vectors defining the directions of the X_0 and Z_0 axes
	VYZ		Specification by the vectors defining the directions of the Y_0 and Z_0 axes
	FR	α, β, γ	Three Euler angles
	VCPP	X_1, Y_1, Z_1, X_2, Y_2, Z_2	In the cylindrical system by two points
	VSPP	X_1, Y_1, Z_1, X_2, Y_2, Z_2	In the spherical system by two points

Document 33. "Coordinate Systems for Calculating Stresses and Forces"

This document is similar to document 32.

Document 34. "Subsoil Parameters"

The line of the document has the following form:

<identifier> <data > [<identifier> <data >…]: <list of elements> /

Specifies the parameters of the elastic subgrade for the bar and plate elements.

Element type	Identifier	Data	Contents
Bars	**CZ**	b C_{1Z} C_{2Z}	b — width of the bar section (m); C_{1Z} — Winkler coefficient in the direction of the Z_1 axis (T/m^3); C_{2Z} — Pasternak coefficient in the direction of the Z_1 axis (T/m)
	CY	h C_{1Y} C_{2Y}	h — height of the bar section (m); C_{1Y} — Winkler coefficient in the direction of the Y_1 axis (T/m^3); C_{2Y} — Pasternak coefficient in the direction of the Y_1 axis (T/m)
Plates	**C1**	C_1	Compression stiffness of the elastic subgrade
	C2	C_2	Shear stiffness of the elastic subgrade for the isotropic, orthotropic, and anisotropic elastic subgrade. Their coordinate system coincides with the coordinate system of the stiffness characteristics
		$C_{2,x}$ $C_{2,y}$	
		$C_{2,x}$ $C_{2,y}$ $C_{2,xy}$	

32.6. Specifying Static Loadings

Documents 6 and 7 are used to specify the static loadings. Static loads can be defined as concentrated and distributed forces and moments, thermal actions, and also by the value of the displacement of the constraint (specified displacement). The following information is specified for each load:

In the document 6:
- type of load (static, dynamic);
- data for dynamic loadings.

In the document 7:
- number of the loading in which the load is applied;
- type of load;
- direction of the load along the axes of the local or global coordinate systems;
- point of load application (lists of the numbers of nodes or elements);
- load values (with their signs) and, if necessary, snaps which define the position of the load within one finite element, linear expansion coefficients for the thermal load, etc.

Document 6. "Types of Loads"

Each line of the document has the following form:

<loading number> [Type = <type>] [Name=<name>]

[<Numerical characteristics of the dynamic loadings>]
[Aks=<file name>]
[AksX=<file name>] [AksY=<file name>] [AksZ=<file name>]
[Betta=<file name>]
[BettaX=<file name>] [BettaY=<file name>] [BettaZ=<file name>]
[: <mass mask for the dynamic loading>]

Type – reserved for the description of the characteristics of loadings. For static loadings – 0.

[Aks=<file name>] – name of the accelerogram file for the seismic loadings. The accelerogram acts in all three directions. The following identifiers are used for the specification of different accelerograms in different directions.

[AksX=<file name>] – name of the accelerogram file for the seismic loadings. The accelerogram acts in the X direction.

[AksY=<file name>] – name of the accelerogram file for the seismic loadings. The accelerogram acts in the Y direction.

[AksZ=<file name>] – name of the accelerogram file for the seismic loadings. The accelerogram acts in the Z direction.

[Betta=<file name>] – name of the dynamic amplification factor file for the seismic loadings. The dynamic amplification factor acts in all three directions. The following identifiers are used for the specification of different dynamic amplification factors in different directions.

[BettaX=<file name>] – name of the dynamic amplification factor file for the seismic loadings. The dynamic amplification factor acts in the X direction.

[BettaY=<file name>] – name of the dynamic amplification factor file for the seismic loadings. The dynamic amplification factor acts in the Y direction.

[BettaZ=<file name>] – name of the dynamic amplification factor file for the seismic loadings. The dynamic amplification factor acts in the Z direction.

<mass mask for the dynamic loading> – enables to take into account the masses only in the given directions in the given loading. If this identifier is missing, all the specified masses will be taken into account. For example, ":1 2" means that only the masses in the X and Y directions will be taken into account in the calculation.

Document 7. "Values of Loads"

The document is used to specify the loads on the structure. Each line of this document has the following form:

<type of load> <direction of the load> <list of the load values> [Load=<number>] [File=<name>]

: <list of nodes, elements or loadings depending on the type of load> ,

where

[Load=<number>] - indicates the loading to which the described load corresponds. If this identifier is missing, the load refers to the last given number.

<type of load> - specifies the description of the load.

The main types of loads are given in Table 32.6-1.

Table 32.6-1

Types of loads

Characteristics	Type	Contents
Load on the node	0	Load in the directions of the axes of the global coordinate system
	1	Specified displacement in the node (see Sec. 8.2)
	2	Load in the special coordinate system specified in the line 3 of the header document
	3	Load in the special coordinate system specified in the document 7
Masses at the description of the dynamic loadings	4	3-rd number 3. Masses are generated automatically by converting the static loading the number of which is indicated in the first position of the line (see Sec. 8.6)
	100	Node's loading history given as a graph
Load on the finite elements in the directions of the axes of its local coordinate system	5	Concentrated in a point
	45	Concentrated in a point of the bar element with the snap in the fractions of the length of the flexible part
	6	Uniformly distributed
	7	Trapezoidal
	47	Trapezoidal on the bar elements with the snap in the fractions of the length of the flexible part
	8, 88	Thermal
	9	Uniformly distributed along the line connecting any two nodes of the element
	10	Trapezoidal load along the line connecting any two nodes of the element
Analysis for the specified displacements with the help of the null elements	11	Load on the "null elements" modeling the specified displacement in the local coordinate system of the element (see Sec. 8.2)
Load on the finite elements in the directions of the axes of the global coordinate system	15, 16, 17, 19, 20	Similarly to the load in the local coordinate system
	55, 57	Analogues of the loads of type 45 and 47 in the global coordinate system
Self-weight	96	The self-weight factor is specified
Allowing for the rigid inserts	46, 56	Uniformly distributed taking into account the load applied to the rigid inserts in the local and global coordinate systems
Self-weight	116	The self-weight factor is specified taking into account the load applied to the rigid inserts
Masses	85, 86	Concentrated and uniformly distributed masses in all directions
Area loads	220	uniformly distributed loads along a line and a polyline
	219	trapezoidal loads along a line
	216	uniformly distributed loads on an arbitrary polygon given by a contour
	217	trapezoidal loads on a triangle and a quadrangle

<direction of the load> - numerical designation of the direction of the load.

The load directions are denoted by integers — 1, 2, 3 along the X, Y, Z axes of the global coordinate system, or X_1, Y_1, Z_1 of the local coordinate system depending of the type of load; and 4, 5, 6 — angular directions about the same axes. The peculiarities of the specification of loads for the dynamic actions are described in Sec. 32.7. The order of the specification of the static and dynamic loadings is arbitrary.

<list of the load values>

The nodal force or moment, as well as the uniformly distributed load are described by one number — their value. For example, the local load for bars in the form of a concentrated force or moment is described by two numbers: the value and snap to the first node, and that for shell elements — by three numbers: the value and snaps to the first node in the directions of the local X_1 and Y_1 axes.

The values of loads specified for the finite elements and their order are indicated in the description of elements in Chapter 3 and, as a rule, correspond to the sequence of their input in the graphical preprocessor. The peculiarities of the specification of thermal loads on plates and solid elements are given in the Tables 32.6-2 and 32.6-3.

Table 32.6-2

Thermal load on flat elements

FE type	Document 6		Load values in the document 7	Contents
	type	direction		
11-20	8	0 4 5	Δt, α Δt, α_x Δt, α_y	Δt — temperature difference between the upper and lower surfaces, α, α_x, α_y — linear expansion coefficients in all the directions and only in the X or Y direction, respectively, t — increase (decrease) of the temperature of the middle layer of the plate with respect to the normal operating conditions (restraint temperature)
21-30	8	0 1 3	t, α t, α_x t, α_z	
41-50	8	0 1 2 4 5	t, Δt, α t, α_x t, α_y Δt, α_x Δt, α_y	
11-20 21-30 41-50	88	0 1 2 4 5	t, Δt, α t, Δt, α_x t, Δt α_y t, Δt, α_x t, Δt, α_y	Used for the unification of loads on the plate elements. For the directions 1, 2 Δt is not taken into account, for the directions 4, 5 t is not taken into account

Table 32.6-3

Thermal load on solid elements

Document 6		Load values in the doc.7	Contents
type	Direction		
8	0 1 2 3	t, α t, α_x t, α_y t, α_z	t(degrees) — increase (decrease) of the temperature with respect to the normal operating conditions (restraint temperature), α — thermal linear expansion coefficient for an isotropic body, α_x, α_Y, α_z — thermal linear expansion coefficients for the axes of the orthotropic body.

The specification of loads for the dynamic loadings is given in Sec. 32.7.

The following sign convention is used in **SCAD** when specifying the loads:

- positive concentrated and distributed forces are opposite to the respective axes of the global or local coordinate system, and the moments act clockwise when viewed from the end of the axes;
- the specified displacement is considered to be positive if it is directed along the axis or counter-clockwise.

It means that the vectors of displacements and rotations determined according to the right-hand screw rule coincide with the respective axes, and the vectors of forces and moments are opposite to them.

[File=<name>] – used when specifying some types of dynamic loads. For example, when describing the pulse or at the direct integration of the equations of motion (see Chapter 15).

Documents 9 and 10. "Groups of Loads"

The description of the groups of loads is completely identical to the description of loadings. Loads included in the groups are recorded in the document 9 (analogue of the document 6) and document 10 (analogue of the document 7).

32.7. Description of Dynamic Loadings

When describing the dynamic loadings it is necessary to specify the following data in the documents 6 and 7:

- specify the characteristics of dynamic loadings in the document 6;
- describe masses in the document 7, and specify the additional data necessary for the analysis for the impact and pulse actions, harmonic oscillations, etc.

Description of Masses in Documents 6 and 7

It is necessary to describe the masses for all types of dynamic actions in the document 7, which are specified as the weights in tones, i.e. mass multiplied by the gravitational acceleration. The software enables to generate the masses automatically on the basis of the local loads of one or several static loadings. Masses can be specified as a special local load intended only for the description of masses, or in the form of the concentrated nodal forces.

Masses can be described in the following three ways (including the case when they are in one loading):

- as concentrated nodal forces;
- as a local load of the type 4 on the elements of the design model indicating index 0 — for uniformly distributed and 1 — for concentrated loads (loads of the type 4 0 and 4 1) in the position used for the indicator of the direction of the load action;
- with an automatic transformation of the static loads into masses (loads of type 4 3).

Masses (their weights are specified) form the inertial forces corresponding to the translational displacements of a node X, Y, Z. Inertial moments correspond to the angular displacements of the nodes. Moments of inertia of the masses concentrated in these nodes are specified to take them into account.

Concentrated nodal masses are described as ordinary nodal loads. When they are specified, the degrees corresponding to the index of the model type have to be assigned.

Automatic Generation of Masses

SCAD enables to generate the masses automatically for the dynamic loadings on the basis of the static loadings (loads of type 4, direction 3). In this case a line of the document 7 will have the following form:

4 3 Load=k r : <list of loadings>/,

where 4 3 — code of the type and direction of the load;

k — number of the dynamic loading for which the masses are specified;

r — coefficient by which the weights in the static loadings are multiplied when the masses are generated;

<list of loadings> — list of static loadings from which the masses are formed.

Characteristics of Dynamic Loadings

Chapter 8 contains the list of dynamic modules currently implemented in **SCAD**. The characteristics of dynamic actions depend on the dynamic module and are recorded in the document 6, which is composed in such a way so that the content of the first 3 positions for all types of dynamic actions is the same and means:

- *position 1* — number of the module (type) of the dynamic action, which defines the nature of the information following it;
- *position 2* — maximum number of the natural oscillation modes which have to be taken into account in the calculation;

- *position 3* — for the modules 21 and 31 (wind pulsation) — number of the wind static loading, for all other types of loadings — zero.

The information depending on the dynamic module follows from the 4-th position. Since it is much easier to specify the characteristics of the dynamic actions in the graphical preprocessor, we will not describe their filling in detail. We will only note that both the coefficients depending on the selected codes, and the names of the accelerogram files or the dynamic amplification spectra are indicated (see Sec. 8.7).

Specifying Pulse Loads

When the dynamic modules 22 and 42 are used, pulse actions are described in the document 7 as ordinary nodal concentrated forces with the specified directions, but instead of one number the following numbers are recorded:

Q P|A *f* τ|0 T_0 n F/ ,

where **Q** — weight of the mass, in the node (it is set to 0, if the mass is already described elsewhere, or the pulse comes to the point where the mass is not specified);

P or **A** — the average pulse value in T·sec, if the number of the pulse mode is specified by a positive number from 1 to 6, or the pulse amplitude at the values of the number from 11 to 16;

f — number of the pulse mode (see Fig. 2.4-1). If the number is equal to 100, the pulse mode is in the file the name of which is given after the **FILE** identifier;

τ|0 — duration of the pulse or impact action, sec. When it has a zero value, the pulse is considered to be instantaneous;

T_0 — repetition period of the action, sec;

n — number of repetitions. It does not have to be specified if there is one pulse;

F — start time of the pulse action, sec. By default — 0.

It is not necessary to specify the last four numbers.

If the pulse mode is specified with the help of a graph, it must be in the text file with the **.spi** extension. Its structure is given below.

A file with the **.p30** extension containing the specified and calculated pulse characteristics is generated in the calculation process. For example:

```
Loading - 3
node  direct. pulse  time  amplitude  period  phase  multiplicity  file name
2      3      2.000  104.72  2.000    52.360  0.000  1            harmonics
Loading - 4
node  direct. pulse  time  amplitude  period  phase  multiplicity  file name
2      3      2.602  100.00  2.000    0.000  0.000  1
```

Specifying Impact Loads

The impact is described in the document 7 as ordinary nodal concentrated forces with the specified directions of the action, the following numbers are recorded:

Q M V T R A/,

where **Q** — weight of the mass coming into the node. It can be set to 0, if this mass is already described elsewhere, or the impact comes into the point where the mass is not specified;

M — weight of the mass of the impact body. If after the impact the mass of the impact body is attached to the structure, it has to be added to the first number;

V — velocity of impact m/sec;

T — impact duration (sec). When it has a zero value, the pulse is considered to be instantaneous;

R — coefficient of restitution;

A — angle (in radians) between the direction of the impact and the normal to the area it is acting on (only for the module 33).

Files with Pulse Parameters

A text file with the description of the pulse has the **.spi** extension and contains the following information:

- comment (obligatory) which ends with the character "#" ;
- scaling factor;

- type of information, which can take the following values:
  0 — P(A) is a scaling factor to the values specified by the graph, which is scaled by the specified value τ along the time axis;
  1 — P(A) is a pulse value, then the area of the graph will be equal to the specified value P, and duration — τ;
  2 — P (A) is the amplitude of the pulse graph, and duration — τ;
  3 — it is considered that the known accelerations of masses due to the external action are specified with the help of the graph. The graph is normalized similarly to the type 2, but the specified values are also multiplied by the mass in the node in the direction of the pulse action;
- sequence of pairs of numbers, the first one of which is the time value, and the second one is the corresponding pulse value. The starting point is obviously zero.
  Example of a file with a pulse mode:

```
Test
#
1 0
0.000000 0.000000      0.050000 0.006000    0.100000 0.012000    0.150000 0.017999
0.200000 0.023998
0.250000 0.029996      0.300000 0.035992    0.350000 0.041988    0.400000 0.047982 …
```

Document 71. "Loading History"

It includes the data for the module of the direct integration of the equations of motion (see Chapter 15).

32.8. Auxiliary Documents

Document 21. "Grid Lines"

The parameters of grid lines are specified in this document. The line of the document has the following form:

<direction> <name> <coordinate>

where the indicator of the direction has the following values:

1 — for the transverse grid lines (these are usually numbered grid lines);
2 — for the longitudinal grid lines (these are usually lettered grid lines);
3 — for the elevations.

The name of the grid line is given in quotes (for the elevations — it coincides with the value of the coordinate), and the value of the coordinate corresponds to:

- for the elevations — the Z coordinate;
- for the numbered grid lines — the X coordinate;
- for the lettered grid lines — the Y coordinate.

Example of filling the document 21.

```
(21 /
  1 "1"    0./
  1 "2"   7.8/
  1 "3"   15./
  2 "A"    0./
  2 "B"   2.4/
  2 "C"   7.2/
  3 "0"    0./
  3 "3.3" 3.3/)
```

Document 27. "Meshing Contour"

Groups of nodes which make up the meshing contours are specified in this document. Each line of this document contains the description of one contour saved when performing the meshing, and has the following form:

Name <contour name>: <list of nodes>

where **Name** — keyword followed by the meshing **contour name** given in quotes;

list of nodes — numbers of nodes which form the vertices of the contour (the numbers of the first and the last nodes coincide).

Example:

(27/ Name="Contour No. 1": 1 149 164 16 1/
Name="Contour No. 2": 10 5 15 11 28 10/...)

Document 47. "Groups of Elements"

Each group of elements has its corresponding record of the following form in the document

<purpose of the group><type> [Name=<name>]:<list of elements>

The first position of the record contains the index indicating the purpose of the group (1 — group for the reinforcement selection postprocessor; 2 — general purpose group). The second position contains the types of elements in the group (0 — arbitrarily; 1 — bars; 2 — plates). The second position is used only in the groups for the reinforcement selection postprocessor.

Document 48. "Groups of Nodes"

Each group of nodes has its corresponding record of the following form in the document

2 0: [Name=<name>]:<list of elements>

It includes an index of the group type (2) and the type of operation with the nodes (0).

32.9. Erection Documents

The main data for the **Erection** mode (Chapter 14) are in the document 72 **Erection Mode**. Each line of this document has the following form:

Name <name of the stage> LoadBase <main loading>
Elem <list of elements>; Load <list of loadings>;
{LoadGroup <list of groups of loadings>;]
[Type<accounting for the deformation of the model at the previous stage>] /

Moreover, the following groups of elements can appear for the erection stages:

- 1005, 2005, … – constraints;
- 1002, 2002, … – boundary conditions;
- 1034, 2034, … – subsoil parameters;
- 1073, 2073, … – coefficients of variation of the rigidities.

The first part of the number describes the number of the erection stage. The **Constraints** and **Subsoil Parameters** documents are completely similar to those described above. The lines of the **Coefficients of Variation of the Rigidities** documents have the following structure:

<coefficient> : <list of elements> / .

32.10. Documents for Checking and Selecting Rolled Steel Sections

The check and selection of the rolled steel sections is performed on the basis of the following documents:

28 – "Check and selection of rolled steel sections";
29 – "Unification at the selection of rolled steel sections".

They have a complex structure, and therefore they should be specified with the help of the graphical postprocessor.

32.11. Documents for Reinforcing Sections of R/C Elements and Its Check

The check and selection of reinforcement is performed on the basis of the following documents:

53 – "Reinforcement of the sections of the r/c elements";
74 – "Specified reinforcement of plates";
75 – "Specified reinforcement of bars".

They have a complex structure, and therefore they should be specified with the help of the graphical postprocessor.

32.12. Initial Data of Postprocessors

Documents of Design Combinations

The information is specified in the documents 61, 62, 63, and 64. The following information is specified in the document 61 (see Fig. 18-12):

<characteristics of the loading> :
<list of simultaneous loadings>;
<list of mutually exclusive loadings>;
<list of related loadings> /

The characteristics of the loading include:

- type of loading;
- subtype of loading;
- crane number;
- crane mode;
- indicator of the activity (consideration) of the loading.

If there is no corresponding list, 0 is output.

List of the elements for the selection of DCF, groups of DCF, and unification groups are output in the documents 62, 63, and 64 respectively.

Documents of Design Combinations for Nonlinear Analysis

The information specified in the documents 81, 82, 83, and 84 is completely identical to the ordinary DCF.

Document 36. "Combinations of Loadings"

The document describes combinations of loadings. The values of the coefficients with which the respective loadings are included in the combination are given in the sequential order of the loadings in each line of the document. Starting from the second combination the number of the coefficients is increased by one, since the preceding combinations can be included as loadings in the subsequent ones with their coefficients.

Example of filling the document 36.

```
(36/
1.  1.  0.  0.  0.  0.  1./
1.  0.  1.  0.  0.  1.  0.  0./
1.  1.  1.  1.  1.  1.  1.  0.  0./)
```

Document 37. "Stability"

The document contains one line with the following data:

- position 1 — contains the index the zero value of which indicates that the stability analysis is performed for loadings, and one — for combinations of loadings;
- position 2 — one in this position means that in the analysis process it is necessary to determine the stability factor of safety;
- position 3 — one in this position means that in the analysis process it is necessary to determine the buckling modes;
- position 4 — one in this position means that in the analysis process it is necessary to calculate the unsupported lengths of the elements;
- position 5 — value defining the upper search limit of the stability factor of safety;
- position 6 — value defining the accuracy of the calculations;
- position 7 — number of loadings or combinations of loadings in the problem;
- position 8 — numbers of loadings or combinations for which the stability analysis is performed.

Example of filling the document 37.

```
(37/ 0  1  1  1  2.  .01  4  1  2  3  4/)
```

Document 38. "Principal and Equivalent Stresses"

This document contains the data which are used for the control of the mode of the calculation of the principal and equivalent stresses. The positions in the line of this document are described below:

- position 1 — not used (always 0);
- position 2 — one in this position means that the calculation of the principal stresses for loadings will be performed;
- position 3 — one in this position means that the calculation of the principal stresses for the combinations of loadings will be performed;
- position 4 — one in this position means that the calculation of stresses for loadings will be performed;
- position 5 — one in this position means that the calculation of the equivalent stresses for the combinations of loadings will be performed;
- position 6 — index of the failure theory according to which the calculations will be performed is specified in this position. Indices correspond to:
  1 — the maximum normal stress theory;
  2 — the maximum linear strain theory;
  3 — the maximum shear stress theory;
  4 — the Huber–Hencky–von Mises theory;
- positions 7–12 — reserve positions (filled with zeros).

Example of filling the document 38.

(38/ 0 1 1 1 1 4 0 0. 0. 0. 0. 0/)

Document 39. "Loads from the Structural Fragment"

The document contains the data for performing the calculation of loads from the structural fragment. The information includes a line with the settings of the mode, a line with the numbers of the elements which make up the fragment, and additionally one line with the data for each structural fragment, with the lists of nodes in which the loads are calculated. The first position of each line with the description of the segments contains the value of the rotation angle of the segment about the X axis of the global coordinate system.

Example of filling the document 39.

```
(39/
1 1 1/
1-456 461-684/
30.0 253/
0.0 13 21 73/
60.0 1 2 53 79/)
```

Document 53. "Reinforcement of Reinforced Concrete Elements"

The document has a complex structure and enables to save the data for the reinforcement at the export/import of the text file.

Document 70. Progressive Collapse

It contains the following information:

- evaluation level: 0 – cautious, 1 – no;
- number of the combination;
- number of the groups of loads;
- indicator for saving the progressive collapse problem with an addition to a **.dst** file;
- dynamic amplification factor (by default 2);
- coefficient for the collapsing structures (groups);
- uncertainty interval, where the strength of the element is not yet exhausted;
- type of the analysis: 0 – linear, 1 – nonlinear;
- method of the nonlinear analysis: 0 – simple incremental, 1 – incremental-iterative, 2 – iterative;
- number of steps;
- maximum number of iterations.

32.13. Special Coordinate Systems

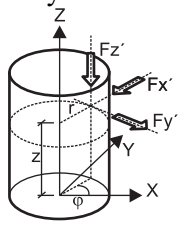
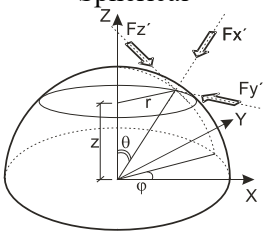
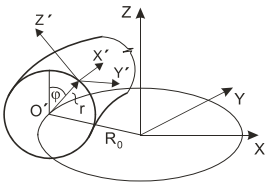
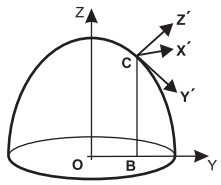
Line 3 of the document 0 is introduced in those cases when it is necessary to transform the coordinates of the nodes specified in the document 4. Cylindrical, spherical and other coordinate systems can be used.

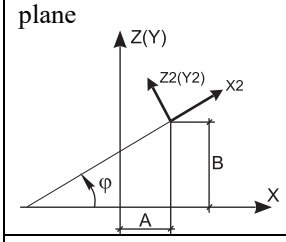
Information in a line is specified as lists with the following structure:

<index of the transformation type> [: <list of nodes>]
[: <numerical properties of the transformation>];

The transformations of the coordinates specified in the line 3 can be also used when specifying the loads in these coordinate systems. If the list of nodes is not specified, the transformation is performed for all nodes. The allowable types of transformations and the numerical properties necessary for them are given in the Table 32.13-1. If two numbers are indicated in the **Type** column, the angles are specified in degrees for the first type, and in radians — for the second one.

Table 32.13-1

Name	Type	Coordinates	Directions of the load axes	Numerical properties
Cylindrical 	1(11)	r — polar radius; φ — polar radius in radians (degrees); z — applicate	X′ — along the radius from the center; Y′ — along the tangent counterclockwise when viewed from the end of the Z axis; Z′ — coincides with the Z axis	No
Spherical  and toroidal 	2(12)	r — polar radius; φ — polar radius in radians (degrees); ψ — angle between the radius and the vertical axis in radians (degrees)	X′ — along the radius from the center; Y′ — along the parallel counterclockwise when viewed from the end of the Z axis; for a torus — in a circle parallel to the XOY plane; Z′ — along the meridian, making a right-hand system together with X′ and Y′	R_0 — for the toroidal one
Tension (compression) along the coordinate axes	3			**Q_1 Q_2 Q_3** — coefficients of tension (compression) along the X, Y, Z axes, respectively
Quadratic transfer surface 	4	$X_2 = X$; $Y_2 = Y$; $Z_2 = AX^2+BY^2+C$	Z′ — perpendicular to the surface and makes an angle less than 90° with the Z axis; Y′ — perpendicular to Z_2 and lies in the same plane with the Z axis making an angle greater than 90° with the Z axis; X′ — perpendicular to Y_2 and Z_2; System X_2, Y_2, Z_2 is right-hand	

Name	Type	Coordinates	Directions of the load axes	Numerical properties
Transfer of the coordinates in space	5	$X_2 = X + A$; $Y_2 = Y + B$; $Z_2 = Z + C$		A B C
Affine transformation	6	$X_2 = L_{11} X + L_{12} X + L_{13} X$; $Y_2 = L_{21} Y + L_{22} Y + L_{23} Y$; $Z_2 = L_{31} Z + L_{32} Z + L_{33} Z$		L_{11} L_{12} L_{13} L_{21} L_{22} L_{23} L_{31} L_{32} L_{33}
Transfer with the rotation in the XOY plane 	7(17)	$X_2=X \cos\varphi - Y \sin\varphi + A$; $Y_2=X \sin\varphi - Y \cos\varphi + B$; $Z_2= Z$		φ A B
Transfer with the rotation in the XOZ plane	8(18)	$X_2 = X \cos\varphi - Z \sin\varphi + A$; $Y_2 = Y$; $Z_2 = X \sin\varphi + Z \cos\varphi + B$		φ A B
	9	$X_2 = X$; $Y_2 = Y$; $Z = \sqrt{A \cdot x^2 + B \cdot y^2 + C}$		A B C
Transfer surface	10	$X_2 = X$; $Y_2 = Y$; $Z = \sqrt{A \cdot x + B \cdot y + C}$		A B C

32.14. Specifying a Load in the Special Coordinate Systems

Documents 6 and 7 are used to specify the loads in the special coordinate systems. Loads of type 2 are specified for the nodes the coordinates of which are given in the special coordinate systems, and those of type 3 — for all nodes.

A nodal load in the special coordinate system can be specified for each node the coordinates of which are given in this system. In order to do it index 2 (load type) is input into the second position of the document 6. The number of the direction of load is specified in the position 3 of the document 6: 1, 2, 3 — along the X_2, Y_2, Z_2 axes, respectively; 4, 5, 6 — about the same axes.

The values of the nodal loads are specified in the document 7. The positive sign corresponds to the action against the local coordinate axes and the clockwise rotation when viewed from the end of the respective local axis.

Load type 3 is used when specifying the loads in the special coordinate systems for the nodes specified in the global coordinate system. Document 7 contains the information necessary for the transformation. The direction of the load in the document 6 is specified in the similar way as the direction for the load of type 2.

Two coordinate systems are implemented for the load of type 3:

- *cylindrical*, for which the following information is specified in the document 7:

***f* 1 [A B]**

where ***f*** — value of the load the direction of which in the local coordinate system is described in the document 6;

1 — indicator of the cylindrical transformation;

A, **B** — coordinates of the center of the cylinder in the XOY plane. If they are not specified, it is assumed that the axis of the cylinder lies on the Z axis.

- *spherical*, for which the following information is specified in the document 7:

***f* 2 [A B C],**

where ***f*** — value of the load the direction of which in the local coordinate system is described in the document 6;

2 — indicator of the spherical transformation;

A, **B**, **C** — coordinates of the center of the sphere in space. If they are not specified, it is assumed that the sphere is located in the origin.

Since the amount of information in one line in the document 7 is limited by four numbers, more than one line has to be used to specify five characteristics necessary for the specification of loads in the spherical coordinate system.

33. Service Functions

Service functions include not only the special functions of the ***Service*** menu, but also a number of other functions which enable to save the screen image, save the description of the design model in the text format etc.

33.1. Save the Fragment of the Model

The operation of saving the fragment can be invoked from the Control toolbar and enables to save the fragment of the design model (selected with the help of the fragmentation functions described in Sec. 10.1) as a separate model. The new model will include all the attributes of the fragment (rigidities, constraints, loads etc.). Before saving, the program asks whether it is necessary to pack the data and prompts for the path to the considered file. In the case of a positive answer, a dialog box with packing options appears.
The operation is available only from the preprocessor.

33.2. Determine the area, perimeter, and angles of polygons

This operation enables to determine the characteristics of a flat polygon bounded by straight line segments passing through the nodes of the design model. In order to perform this operation it is necessary to:

- click the respective button in the **Control** toolbar;
- select the vertices of the polygon by stretching the rubber band and holding the left mouse button;
- finish the selection by double clicking the left mouse button.

The polygon and its characteristics will be shown in the dialog box (Fig. 33.2-1). The color of the polygon corresponds to the color specified for the meshing, and the colors of the background and the node numbers correspond to those specified for the design model. The angles are shown in the same color as the load values. The polygon characteristics are tabulated and can be exported to MS WORD® or MS EXCEL®.

If you double click on a node when specifying a polygon, it will be released from the rubber band.

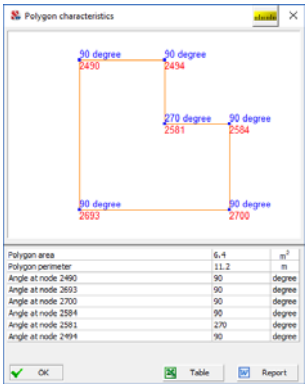


Figure 33.2-1. *The* **Polygon Characteristics** *dialog box*

33.3. Special Functions of the Service Menu

The ***Service*** menu enables to invoke special functions, such as the standard Windows calculator, the formula calculator, and the measurement units converter. Moreover, reference tables with the properties of concrete and the characteristics of reinforcement can be also invoked from this menu.

Formula Calculator

The formula calculator can be invoked from the **SCAD Office** program by clicking on the icon . The ***Service*** menu enables to invoke the standard Windows calculator (if it has been installed together with the system), and a special formula calculator (Fig. 33.3-1).

This calculator is used to perform calculations by formulae that can be specified in a text field.

The following rules should be observed when entering a formula:

- names of functions must be entered in lowercase Roman letters;
- the fractional and the integer parts of a number must be separated by a period;
- arithmetic operations must be specified with the symbols +, –, *, /, ^ (raising to a power), for example, 2.5*2.5*2.5 can also be written as 2.5^3.

The following mathematical functions can be used in the formulae:

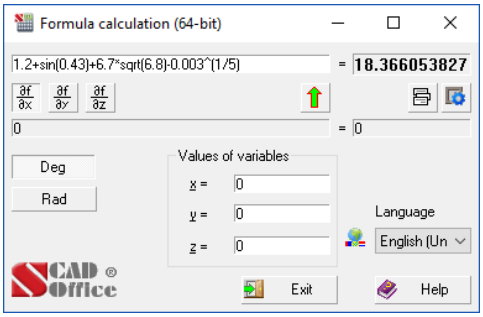


Figure 33.3-1. *The dialog box of the calculator*

floor — the greatest integer not greater than the argument;
tan — tangent;
sin — sine;
cos — cosine;
asin — arc sine;
acos — arc cosine;
atan — arc tangent;
exp — exponent;
ceil — the least integer greater than the argument;
tanh — hyperbolic tangent;
sinh — hyperbolic sine;
cosh — hyperbolic cosine;
ln — natural logarithm;
log — decimal logarithm;
abs — absolute value;
sqrt — square root.

Depending on the state of the **Degrees/Radians** switch buttons, arguments of the trigonometric functions (**sin, cos, tan**) and results of inverse trigonometric functions (**asin, acos, atan**) can be presented in degrees or radians, respectively.

Only parentheses are allowed for grouping arguments together; these can be nested as deeply as desired.

Example.

Formula

$$1,2+\sin(0,43)+6,7\sqrt{6,8}-\sqrt[5]{0,003}$$

must be written as follows:

1.2+sin(0.43)+6.7*sqrt(6.8)–0.003^(1/5).

There is an additional option of using three independent variables **x, y, z** in formulas. Values for the variables will be specified in respective text fields. This makes it possible to perform a series of similar calculations with different parameters. For example, the following formula

$$1,2+\sin(x)+6,7\sqrt{6,8}-\sqrt[5]{y}$$

must be written in this mode as follows

1.2+sin(x)+6.7*sqrt(6.8)–y^(1/5).

Furthermore, if you have entered a formula depending on the variables **x**, **y**, **z**, the field at the bottom will display a symbolic expression of its partial derivative with respect to one of the variables (**x**, **y**, or **z**) for which the respective marker , , is enabled at the moment.

button allows to save the current state of the text field with a formula and the respective values of variables and to open data saved earlier. After pressing the button the drop-down menu appears.

Selecting its first position (**Save**) will save the entered data. The other lines display the formulas entered earlier, which can be selected by the mouse-pointer.

The button enables to invoke the **Settings** dialog box where you can specify the number of significant digits after the decimal point at the output of the results of the calculation.

Converting Units of Measurement

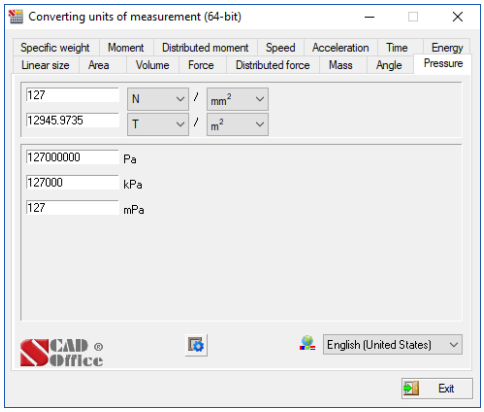


Figure 33.3-2. *The* **Converting Units of Measurement** *dialog box*

The calculator can be invoked from the **SCAD Office** program group — icon , or from the ***Service*** menu.

The application enables to convert the data specified in different units of measurement (Fig. 33.3-2). In order to perform the operation, select a tab of respective measures (**Length**, **Area** etc.).

The order of converting operations depends on whether the units of measurement are simple (i.e. length, area or time) or compound (i.e. pressure, speed or mass).

If you have to convert simple units of measurement, you just have to enter a number into one of the text fields. As a result you will receive values in all other units of measurement. If the units are compound, you have to select the original units of measurement in the drop-down lists of one line and the units you wish to convert to in the other one. Enter the number in the text field of the first line and you will receive the result in the second one.

The respective drop-down list enables to change the language (Russian/English).

Properties of Concrete

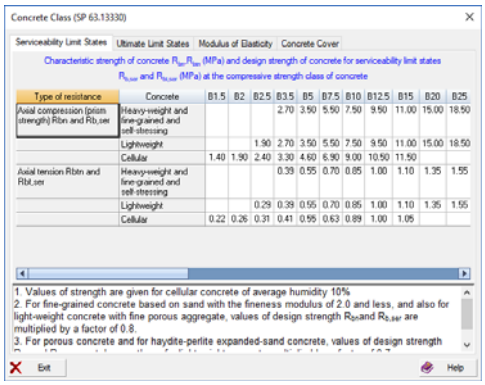


Figure 33.3-3. *The* **Concrete Class** *dialog box*

A multi-tab dialog box of this mode (Fig. 33.3-3) contains data given in Section 2 (Tables 12, 13 and 18) of SNiP 2.03.01-84* (Tables 5.1, 5.2 and 5.4 of SP 52-101-2003, Tables 6.7, 6.8 and 6.11 of SP 63.13330), as well as information on the minimum allowable concrete cover.

Characteristics of Reinforcement

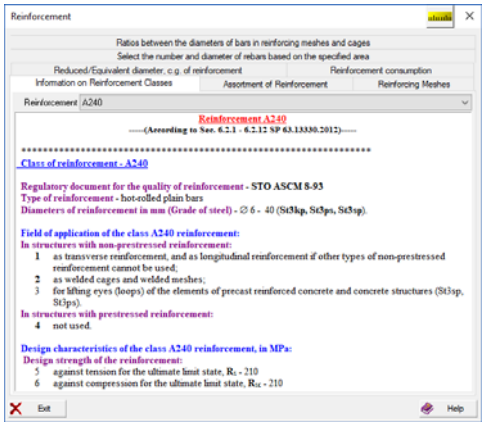


Figure 33.3-4. *The* **Reinforcement** *mode*

This mode (Fig. 33.3-4) provides data on the classes of reinforcement and the assortment of reinforcement products, including an effective area of the cross-section for a particular number of rebars, a theoretical weight of one linear meter and diameters of reinforcement of various classes. Moreover, a separate tab provides information about reinforcing meshes according to GOST 23279-85.

A separate tab enables to convert the cross-sectional area of reinforcement into bars, select the number of bars, and arrange the rebars in the section of the element.

The program enables to select both the minimum required amount of reinforcement of one diameter, and the combinations of bars. In this case, it is possible to obtain the dimensions of the bars and their arrangement in the section of the element.

The result can be obtained with a certain deviation specified by the user (i.e., the total area of the selected bars can be smaller/larger than the specified value by the specified deviation). If the deviation is 0%, the nearest larger value of the total area of the bars of a certain diameter is determined (the 0% deviation limitation is applicable only for single bars, a certain possible deviation has to be specified for a combination of bars).

The algorithm for selecting rebars is based on the following limitations and assumptions:

- the maximum possible number of rows of reinforcement along the height – 3 rows;
- number of rebars in the third row – 2;
- up to five different diameters can be used in a section of the element (it is recommended to use not more than two different diameters);
- bars of larger diameter are placed in the first row at the corners of the section. The bars of larger diameter in each row are always placed at the edges;
- the cross-sectional area of the reinforcement in the section of the smaller bar should not be less than 25% of the cross-sectional area of the larger bar;
- the bars of the 1st row are aligned along the border of the concrete cover (the centers of gravity of the bars are not on the same line, so that the stirrup could be installed);
- the centers of the bars of the second and third rows lie on the same straight line.

The program enables to obtain a report document. Only the variants of the calculation results selected by the user will be output in the report.

The **Reduced/Equivalent Diameter, C.o.G. of Reinforcement** tab enables to perform the following auxiliary operations:

Reduced diameter — enables to determine the reduced diameter of a rebar under the constrained placement of two rebars (pairwise without a gap between them, or with the distance between the rebars of a pair less than the distance required for individual rebars). When specifying the distances between the rebars and determining the anchorage length, this pair of rebars should be considered as a reduced rebar with a diameter of $d_{red} = \sqrt{d_1^2 + d_2^2 - c_1^2}$, where d_1 and d_2 – diameters of the considered rebars, d_1 – clear distance between these rebars taken as not greater than the diameter of a smaller rebar (see Sec. 5.40 and Fig. 102 Guide on designing of concrete and reinforced concrete structures made of heavy-weight or lightweight concrete (no prestressing) (to SNiP 2.03.01.-84)).

Equivalent diameter — enables to determine the equivalent diameter of a rebar when determining the crack opening width if rebars of different diameters are used in the section.

For a section with rebars n_1 of diameter d_1 and n_2 of diameter d_2, the equivalent diameter d_{eq} is determined according to the following formula:

$$d_{eq} = \frac{n_1 d_1^2 + n_2 d_2^2}{n_1 d_1 + n_2 d_2}$$

Center of gravity of the multi-row reinforcement — enables to determine the distance from the edge of concrete to the center of gravity of the multi-row reinforcement.

For a section with rebars of a certain diameter with the total area of $A_{s1}(A_{sn})$ located at a distance of $a_{s1}(a_{sn})$ from the edge of the element, the center of gravity of the multi-row reinforcement is determined according to the following formula:

$$a = \frac{A_{s1}a_{s1} + A_{s2}a_{s2} + \ldots + A_{sn}a_{sn}}{A_{s1} + A_{s2} + \ldots + A_{sn}}$$

Reinforcement Consumption

The tab **Reinforcement Consumption** enables the operation of calculation of the weight of the reinforcement, total cross-sectional area, as well as the number of rebars for a rectangular or arbitrarily-shaped area. The calculation is performed for a set of bars having both the same and different diameters.

The program enables to perform four types of analysis:

- reinforcement consumption (determination of the total area and the total weight of the bundle of N rebars);
- reinforcement consumption for a strip, a×L;
- reinforcement consumption for a rectangular area, a×b;
- reinforcement consumption for an arbitrarily-shaped area, A.

The control of the operation is performed in the multi-tab **Reinforcement** dialog box, which is invoked from the respective section of the **Service** menu. The type of analysis can be selected with the help of the respective radio buttons in the **Reinforcement Consumption** tab, then you have to specify the parameters of the reinforcement and, if necessary, the size of the area.

Reinforcement Consumption for a Bundle from N Rebars

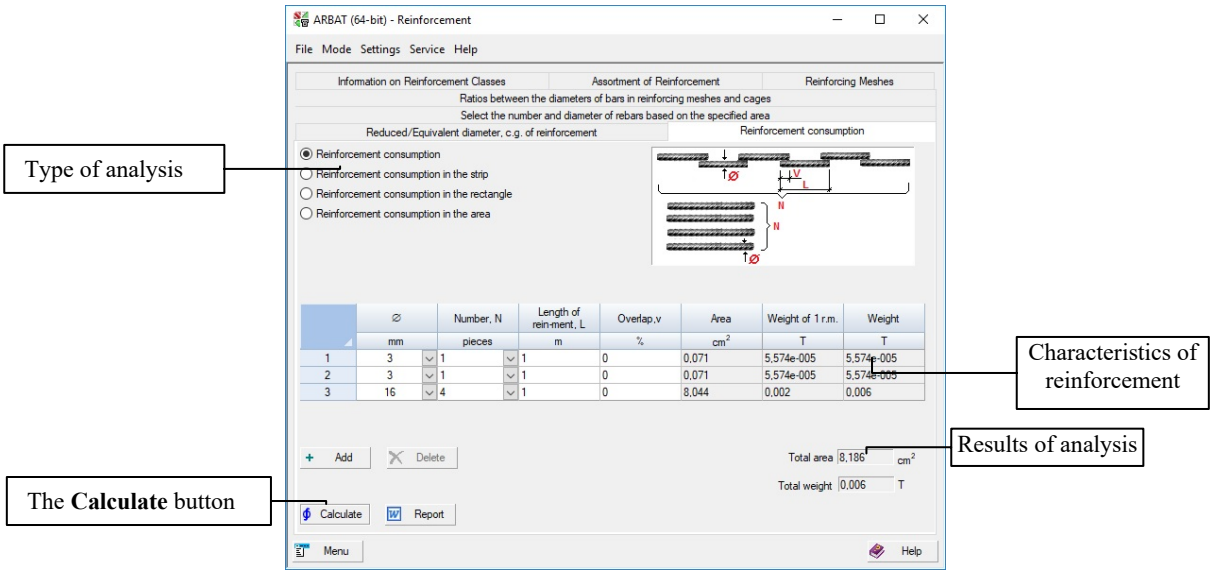


Figure 33.3-5. The **Reinforcement Consumption** tab for the calculation of a bundle from N rebars

The total area and the total weight of the given number of rebars of different diameter is calculated, and the ovelap can be taken into account (specified in % of the total length of the reinforcement). Each diameter of reinforcement has a corresponding line in the table (Fig. 33.3-5). New lines can be added and

the existing ones can be deleted with the help of the **Add** and **Delete** buttons respectively. The analysis is performed by clicking the **Calculate** button. A report with the results of the analysis can be generated by clicking the respective button.

Reinforcement Consumption for a Strip

The total length of rebars, their total weight and the number of rebars is calculated for a certain strip with the length — **a**, and the ovelap can be taken into account (specified in % of the length of the reinforcement). In the transverse direction of the strip, the reinforcement can be represented by either a single bar or a bundle of bars with the overall length — **L**. The analysis is performed by clicking the **Calculate** button. A report with the results of the analysis can be generated by clicking the respective button (Fig. 33.3-6).

When the reinforcement at the edges of the strip is taken into account, one bar is added to the total number of bars.

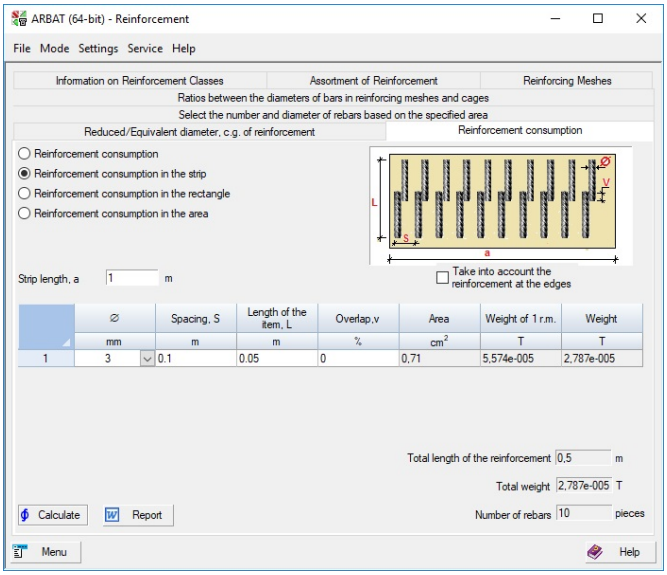


Figure 33.3-6. The **Reinforcement Consumption** tab for the calculation of the reinforcement consumption for a strip

Reinforcement Consumption for a Rectangle

The total length of rebars, their total weight and the number of rebars is calculated for a certain rectangular area — **a**×**b**, and the ovelap can be taken into account. The reinforcement can be placed along one face, for example, **a**, and along both faces. The overlap is given as multiple of the diameter of reinforcement, for example, the number 10 in the **Overlap** column corresponds to the value 10D, where D is the diameter of reinforcement. When the reinforcement at the edges of the rectangle is taken into account, bars of the given length arranged along the edge of the rectangle are added to the total number of bars.

The analysis is performed by clicking the **Calculate** button. A report with the results of the analysis can be generated by clicking the respective button (Fig. 33.3-7).

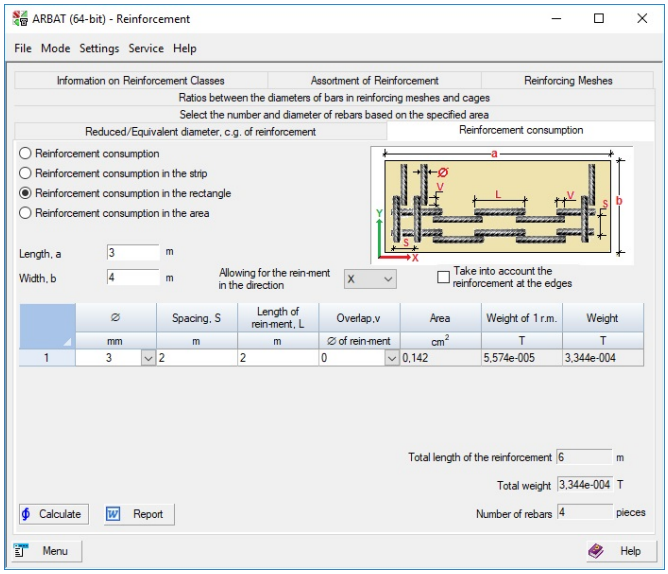


Figure 33.3-7. The **Reinforcement Consumption** tab for the calculation of the reinforcement consumption for a rectangle

Reinforcement Consumption for an Area

The total length of rebars, their total weight and the number of rebars is calculated for a certain given area, and the ovelap can be taken into account. The reinforcement can be placed along one of the axes X(Y), and along both axes. When the ovelap is taken into account, its value is specified in % of the total length of rebars.

The analysis is performed by clicking the **Calculate** button. A report with the results of the analysis can be generated by clicking the respective button.

You can save the results in a file with the initial data with the**.sav** extension, and generate a report with the results of the analysis.

33.4. Import and Export the Initial Data Prepared in the Form of the Text Description

SCAD enables to import the files of the initial data prepared in the text format (see Chapter 30), in the internal formats of the current version. It allows to use the prepared initial data, and for those who prefer a text description – to prepare some data or all the data in the form of the text description finishing them with the help of the tools of the graphical preprocessor.

The text file enables to transfer the initial data to the archive in the form accessible for the control.

The initial data do not necessarily have to include all sections of the description of the model to be imported. The geometric properties are enough (in terms of the archiving language of the first and fourth documents). And even this information may be incomplete. It can be corrected and supplemented with the help of the tools of the graphical preprocessor.

The **Read the project from the text format** operation from the ***File*** menu enables to import the data. And the **Read the project from the text format (previous versions)** operation is used to import the data prepared in **SCAD Office** 11.5.

After invoking the necessary operation and selecting (the name of the text file in the standard dialog box) import is performed, the information on which is provided in the **Result of the Performed Operation** dialog box (Fig. 33.4-1). It contains information on the errors and inaccuracies in the data, and error messages.

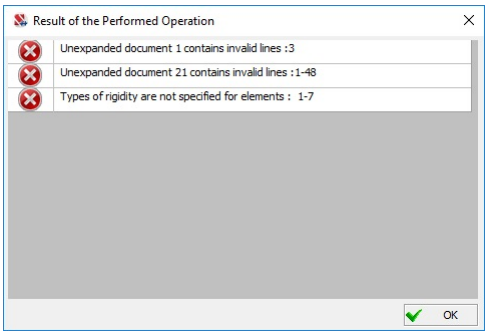


Figure 33.4-1. *The* **Result of the Performed Operation** *dialog box*

If the errors are not critical for the problem, then after clicking the **OK** button it is necessary to select the path to the new spr-file (in the standard dialog box) and the task will become available for working. Moreover, you will be asked whether it is necessary to show the log of the import operation. If the answer is positive, the respective log file will be opened.

The software provides the reverse process as well — export of data, i. e. transformation from the format of the project to the text description in the Windows encoding. This operation is called **Save data as text** and it is available in the ***File*** menu.

When importing the data from the text format the system of units of measurement specified in the data is used (see Sec. 29.3 of this book). When exporting the data to the text format you can specify the necessary system of units of measurement.

33.5. Comments to the Project

The ***File*** menu can be used to write the comments when creating a design model, and when necessary — to save the current status of the project together with the comments, i.e. archive the “intermediate” projects. The main peculiarity of the archive is that all copies of the project are in the same file as the main project and remain there until they are deliberately deleted from the archive or the file of the main project is deleted. The comments can be entered and saved in the **Comments to the Project** dialog box (Fig. 33.5-1), which contains a table with the texts of the comments and the buttons controlling their archiving process.

The table has three columns, data and time of entering the comments are specified in the first one, text is entered in the second column, and the third one contains checkboxes which enable to indicate whether the archive contains a copy of the project corresponding to the time of entering the comment.

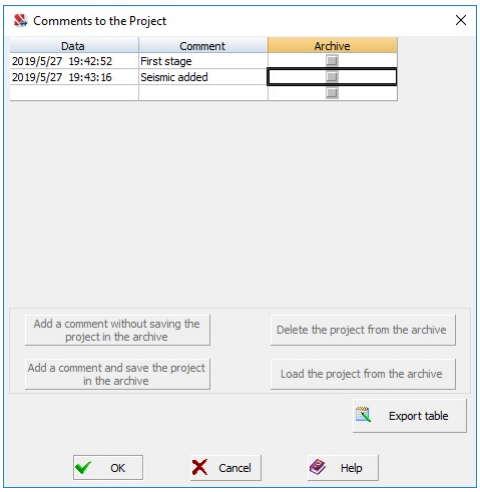


Figure 33.5-1. *The* **Comments to the Project** *dialog box*

Four buttons are used for working with the archive of the projects:

- **Add a comment without saving the project in the archive** — the comment entered in the second column of the table is saved in the archive after clicking this button;
- **Add a comment and save the project in the archive** — the current status of the project is saved in the archive together with the comment;
- **Delete the project from the archive** — the project corresponding to the line selected in the table is removed from the archive, and the comment remains;
- **Load the project from the archive** — the project corresponding to the line selected in the table is loaded as a separate design model (when loading it is necessary to specify the name of the file of the new project different from the name of the file of the original project).

All comments to the project can be saved as an RTF-file (the **Export table** button) and automatically loaded into the text editor specified in the settings.

33.6. Section Viewer

This application can be invoked from the **SCAD Office** directory on the desktop and is used to browse through steel profile assortments. The program window (Fig. 33.6-1) includes a list of assortments and a table of profile properties.

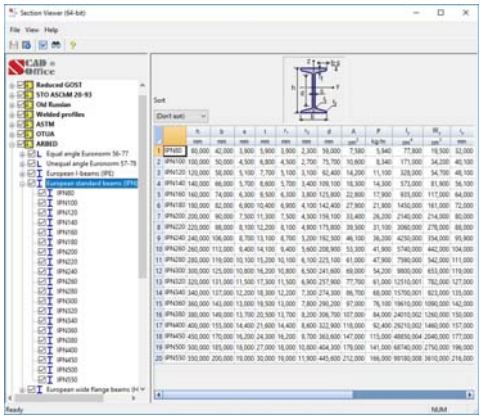


Figure 33.6-1. *The main window of the* **Section Viewer**

To activate a table that contains a list of profiles of a particular type, open the list of profiles of the desired assortment and choose a group of the respective profile type in the list.

To view the section with its dimensions, open the profile group list and choose the respective profile (Fig. 33.6-2).

There are checkboxes on the left from the names of sections in the list, which are checked by default. If you uncheck the checkboxes the respective profiles will be excluded from the list which is used at the selection in the mode of the check of the steel structural elements, although this profile can be assigned when specifying the rigidities of elements.

The sections tables can be sorted in the ascending order of values of different properties. The properties are selected from the **Sort** drop-down list. The order of the sections in the list by default corresponds to the accepted one in the standard (or catalogue).

If needed, the selected profile table can be exported as an **RTF** format fil[illegible] an editor associated with this format (the **Print** button —[illegible]). If not the group of profiles but the name of the assortment is selected when the printing is invoked, all the tables of the profiles of this assortment [illegible] printed.

The settings (the **Settings** button— [illegible]) of the mode of viewing and printing are performed according to the same rules as in **SCAD**.

The **Find** dialog box enables to find the profile based on

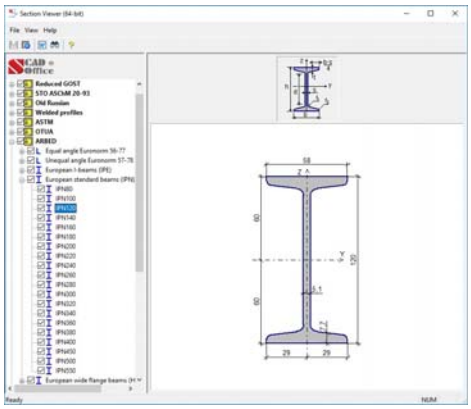


Figure 33.6-2. *The profile image*

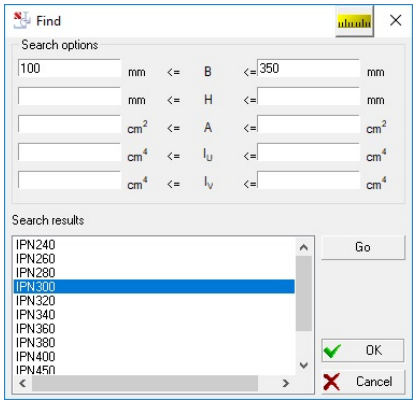


Figure 33.6-3. *The* **Find** *dialog box*

values of the main geometric properties (the **Find** button —) (Fig. 33.6-3). The values (minimum and maximum) of the characteristics of the profile based on which the search is performed are entered in the text fields of the **Search options** group. Clicking the **OK** button initiates the search. If one or more profiles with the specified characteristics are found, their names are output in the **Search results** list.

After indicating one of the selected profiles in the list and clicking the **Go** button, this profile will be highlighted in the table with the characteristics of the sections.

33.7. Graphics Editor

Graphics editor, which is invoked by clicking the button , enables to specify a design model of a bar structure or of its fragment using a set of graphics operations similar to those in the 2D AutoCAD (AutoCAD Light) environment.

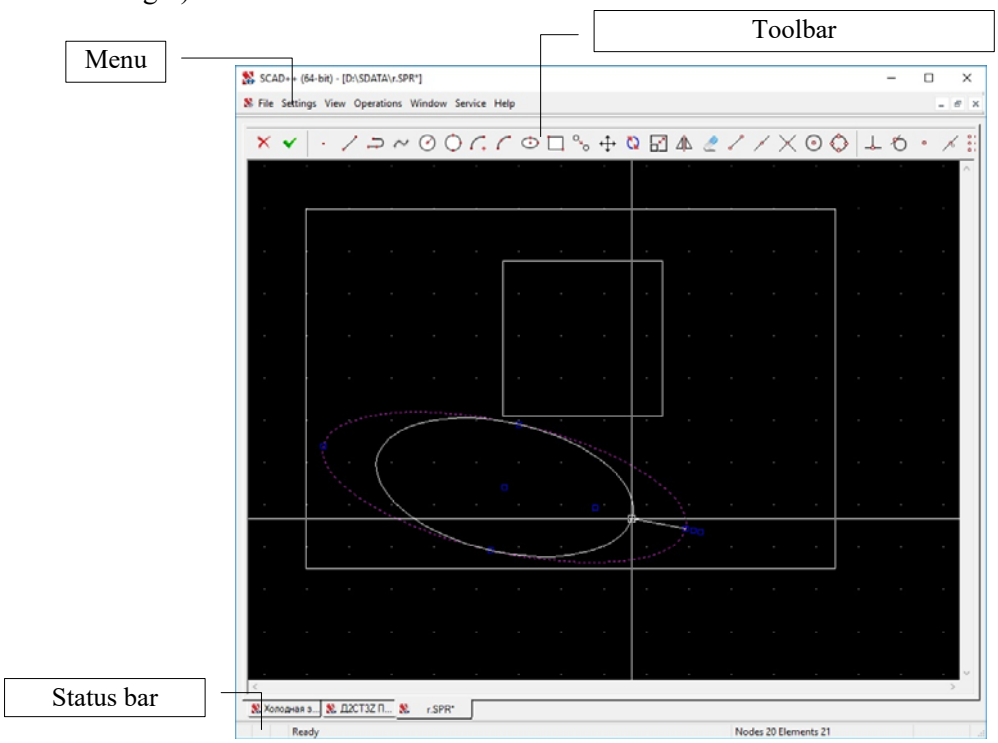


Figure 33.7-1. *General view of the window of the graphics editor*

It should be noted that the **SCAD** graphics editor does not use operations related to the preparation of drawings, dimensioning, etc. Therefore, some settings and operations might be unavailable. The authors of the system were not trying to replace such recognized graphical tools as AutoCAD, Micro Station, and the like. The goal was more modest – to enable the experts and amateurs of these graphic systems to work in a familiar environment. Since the rules of "drawing" implemented in the graphics editor are quite standard, only a brief list of objects which can be input in the software is given below:

Point — draw a point ;

Line — draw a line ;

Polyline — draw a polyline ;

Spline — draw a spline ;

Circle — draw a circle (with the specified center and radius , through three points);

Arc — draw an arc (center, start and end, start, center and end);

Ellipse — draw an ellipse ;

Rectangle — draw a rectangle .

Selection

The selection can be performed by clicking on the object (*Point*), by selecting the necessary objects by a rectangular *marquee*, an arbitrary *polygon* or *polyline*.

Object Snap

The objects can be snapped to each other in the following ways:

Point — snap to the point ;

Endpoint — snap to the start or end point of the line ;

Midpoint — snap to the middle of the line ;

Center — snap to the center of the circle ;

Nearest — snap to the nearest position of the line ;

Grips — snap to grips ;

Intersection — snap to the intersection point of the lines ;

Perpendicular — snap to the perpendicular ;

Tangent — snap to the tangent .

Cursor snaps — provide the accurate positioning.

If the snaps are not specified, it is almost impossible to place the cursor on a certain point of the model (the space of the model in the graphics editor has a continuous character of the material coordinates).

Snap Angular

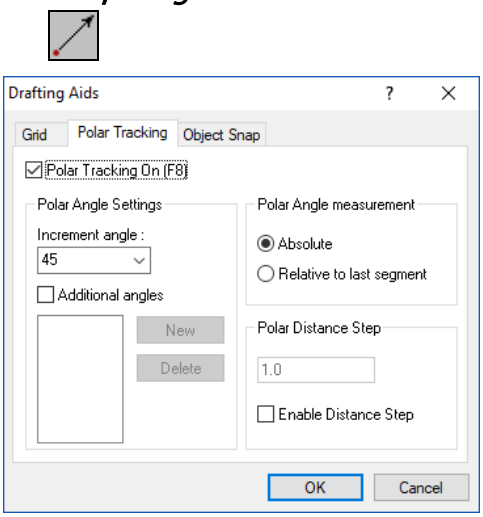


Figure 33.7-2. *The* **Polar Tracking** *tab of the* **Drafting Aids** *dialog box*

Using the polar (Fig. 33.7-2) snap enables to draw the lines at a certain angle and of the given length.

A unit angle in degrees is specified in the **Increment angle** field. It means that a line can be drawn at any angle multiple of the specified step.

Create Graphics Objects

The following objects can be used when generating a design model in the graphics editor: point, line, polyline, spline, circle, arc, ellipse, rectangle.

All objects can be created with the help of a cross cursor. If the input operation is not active, the cross of the cursor is surrounded by a “small” square, and it is used for selecting objects in this mode. If the snaps are enabled, the cross of the cursor is surrounded by a “large” square. You can stop the input of the object by right-clicking.

Point

In order to create a series of points you have to invoke the operation, indicate the position of the point in the model with the cursor, and left-click. If the **Snap to Grid** option is enabled (the button is pressed), the point will be created in the grid node nearest to the cursor.

Line

The editor enables to create a polygonal chain by sequentially indicating the vertices with the cursor and fixing each point by left-clicking. You can stop the input by right-clicking.

Polyline

The following procedure is used for creating the polylines:

- click the **Polyline** button;
- create a polygonal chain the vertices of which will belong to the resulting polyline;
- finish the input by right-clicking.

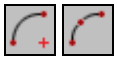
Spline

This operation is similar to creating a polyline with an automatically set *Fit spline* type. If you select the obtained spline and specify other properties for it (similarly to the polyline), you will obtain the same results as for the polyline.

Circle

A circle can be created by specifying its center and radius, or three points lying on the circle.

Arc



An arc can be created by specifying its center, start and end points, or its three points. The parameters of the arc can be changed according to the same rules as for the circle (see the **Circle** section).

Ellipse

In order to create an ellipse you have to indicate its center with the cursor and fix it by left-clicking, drag the rubber band to the point defining the end of one of the axes, fix it, and drag the second rubber band to the point defining the half of the length of the other axis. If you select a created ellipse, an “arm” attached to its center will appear in the drawing. You can capture the free end of the arm with the cursor and rotate the ellipse by the necessary angle with respect to the center.

The ellipse can be adjusted according to the same rules as the circle.

It should be noted that an ellipse or its part will be correctly transferred to the **SCAD** preprocessor only after applying the **Explode** operation to them. Curves forming the ellipse will be replaced by a set of straight line segments in the result of performing this operation.

Rectangle

A rectangle can be created by specifying the start and end points of its diagonal.

Editing Objects

Basic Points

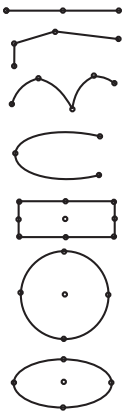


Figure 33.7-3. *Basic points of the geometric objects*

A system of basic points is provided for all geometric objects (Fig. 33.7-3), which enable to perform various transformations of the shape and position of the objects. Depending on the type of the object the basic points are located as follows:

- *line* — ends and midpoint;
- *polyline* — ends of the segments;
- *spline* — vertices;
- *arc* — ends and midpoint;
- *rectangle* — center, corner points, and midpoints on the sides;
- *circle* and *ellipse* — center and four points on the contour line.

It is necessary to perform the following actions to adjust the object with the help of the basic points:

- select the object;
- move the cursor to the basic point and left-click;
- drag the basic point to the new place while holding the mouse button.

If the basic point located in the center of the line, circle, rectangle or ellipse is used to move the object, then it is moved as a whole. Besides the basic points "arms" are provided for ellipses and rectangles, which enable to rotate the object. Similar arms are used to change the shape of the curve in the case of the transformation of a polyline into a controlled Bezier curve.

The list of editing functions (Fig. 33.7-4) is invoked by right-clicking (if the object is selected).

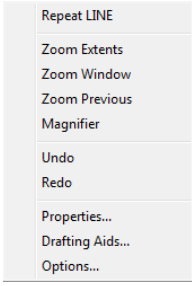


Figure 33.7-4. *Drop-down menu with a list of editing functions*

Copy

It is necessary to perform the following actions to copy the objects:

- select objects for copying (right-click to finish the selection);
- indicate the basic point of copying, i.e. a point which will be used to snap the object to the copying point;
- move the objects into the copying point using the "rubber band";
- left-click to confirm copying (right-click to cancel it).

The same object can be copied successively several times, confirming the copying every time. Right-click to finish the copying.

Move

The position of the objects can be changed by moving them without changing their orientation and size. In order to perform this operation it is necessary to:

- select the objects you want to move (right-click to finish the selection);
- indicate the basic point of moving;
- move the objects using the "rubber band";
- left-click to confirm the operation (right-click to cancel it).

Mirror

Mirroring is performed with respect to the given axis defined by two points. The original objects are not saved after performing the operation. In order to perform this operation it is necessary to:

- select the objects you want to mirror (right-click to finish the selection);
- specify the first point of the mirror axis;
- specify the second point;
- left-click to confirm mirroring (right-click to cancel it).

Rotate

Rotation of the objects can be performed by specifying the center of the rotation and the starting point of the rotation. In order to perform this operation it is necessary to:

- select the objects you want to rotate (right-click to finish the selection);
- specify the point corresponding to the center of the rotation;
- specify the starting point of the rotation;
- rotate the objects;
- left-click to confirm the rotation (right-click to cancel it).

Erase

In order to perform this operation select the objects you want to delete and click the **Erase** button. This operation can be also performed by pressing the **Delete** key on the keyboard.

Scale

When scaling the objects the equality of the scaling factors for the X and Y axes is observed. Thus, the proportions of the object do not change when the sizes are increased or decreased. Scaling is performed by specifying the basic segment. The scaling factor is determined from the ratio of the lengths of the basic and current segments.

Scaling:

- select the objects you want to scale;
- specify the start point of the basic segment;
- specify the end point of the basic segment;
- perform scaling by changing the length of the current segment (decrease or increase);
- left-click to confirm the scaling (right-click to cancel it).

33.8. Project Tree Service Section

Service section of the project tree contains the following items:

- **Delete Calculation Results;**
- **Delete Results for Oscillation Modes.**

The **Delete Calculation Results** operation deletes all files created during the calculation of the model.

The **Delete Results for Oscillation Modes** enables to delete all the information on displacements/forces/stresses/… calculated for individual oscillation modes. In this case, the results related to the total dynamic loads will not be deleted. The user has to specify which results he/she wants to delete in the dialog box (Fig. 33.8-1).

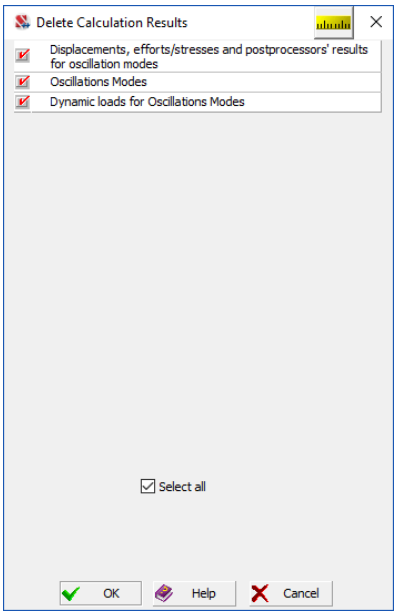


Figure 33.8-1. *The* **Delete Calculation Results** *dialog box*

This operation can be useful when it is necessary to free up space on your hard drive after solving large problems.

33.9. Accelerogram Editor

SCAD has a capability of analyzing the behavior of structures under seismic actions described by given accelerograms. The accelerograms must be prepared in advance as files of a certain format which will be described later.

The accelerogram file format is simple and straightforward, so such files can be created and processed by any text editor or word processor. To further facilitate the accelerogram preparation process, **SCAD Office** has been supplemented with auxiliary software named **Accelerogram Editor**.

Main Window

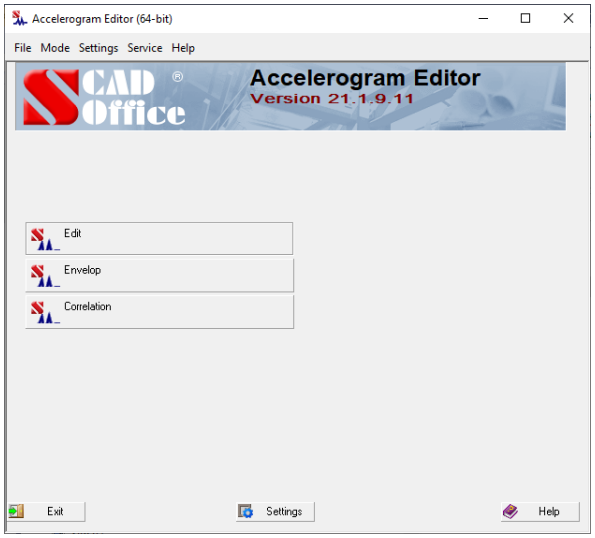


Fig. 33.9-1. **Accelerogram Editor**

Along with the common data editing functionality, this program enables you to perform an analysis of an accelerogram. Particularly, the following information can be obtained in both graphical and tabular form:

- an accelerogram (an acceleration vs. time relationship);
- a velocigram (a velocity vs. time relationship);
- a seismogram (a displacement vs. time relationship);
- an amplitude Fourier spectrum (absolute value of the accelerogram's Fourier transform[1] vs. frequency);
- a power spectrum (absolute value of the accelerogram's Fourier transform squared vs. frequency);
- response spectra (and pseudo-spectra) for given values of logarithmic decrements of oscillations[2];
- spectral density;
- general properties (maximum and root-mean-square accelerations, velocities and displacements, ...).

Accelerograms usually contain errors, no matter whether they are instrumental (i.e. recorded with seismograph) or analytical (calculated on the basis of a measured seismogram). In analytical accelerograms the errors commonly arise because of the numerical differentiation. Most often, these errors will be great in magnitude for the high-frequency component of the action. In instrumental accelerograms, the errors will arise during the recording and digitizing. Most of the recorded accelerograms contain great errors in a long-period range.

A typical error commonly made in accelerograms is introducing an offset and a slope to the baseline (see [2] for examples). Let's give an example borrowed from the book [4] that shows how even the smallest errors of this kind can enormously distort the depiction of displacements experienced by a designed structure. Suppose there is a given harmonic action $\ddot{x}(t) = w\sin(\omega t)$ subjected to a small perturbation in the form of a slope of the baseline of $0{,}001wt$ and an offset of the same line by $-0{,}03w$. The said perturbations are hardly noticeable on the accelerogram of $\ddot{x}(t)$, but they make the seismogram look utterly different from what it should be like; the equation of the latter becomes $x(t) = w(0{,}001\frac{t^2}{2} - 0{,}03t - \frac{1}{\omega^2}\sin\omega t)$. The program suggests a capability of accelerogram balancing in order to be able to amend errors of this type.

You can also adjust the baseline and perform frequency filtering in **Accelerogram Editor**.

Accelerogram files usually have the .**spc** extension and are plain text files. There is an arbitrary text (comment) at the beginning of each file, which is followed by a separator character # and three numbers (a factor for converting the given values to m/sec^2, a number of points in the accelerogram, and a time step in seconds). It is assumed that the acceleration values were recorded at a constant time step. The listed data is followed by the acceleration values. The number of specified values must conform to the number of points. The accelerations, the time step, and the conversion factor are floating-point numbers with a period serving as a separator between their integer and fractional parts. All numbers in the file must be separated by spaces or start from a new line. An example of such a file can be found in Chapter 8.

Settings

The application contains setting operations common for all **SCAD Office** programs. Therefore, the description of operations specific only to the **Accelerogram Editor** is given below.

The **Calculation** tab (Fig. 33.9-2) allows you to choose a maximum value for the frequency up to which the Fourier spectra and the maximum period (or the maximum frequency) for the response spectra will be calculated, time step for calculating the response spectra depending on the period, and to choose the solution method (Duhamel integrals or Newmark method). If the specified value of the maximum frequency of the Fourier spectrum is very large, then the program will automatically limit it to a Nyquist frequency equal to 1/(2Δ), where Δ is the accelerogram time step. By default, the Fourier amplitude

[1] **Accelerogram Editor** uses the following Fourier transform: $\int f(t)e^{2\pi i t\omega}\mathrm{d}t$.

[2] See books [1], [4] for definitions and discussions.

spectrum and power spectrum are calculated and visualized at frequencies k/Δ (k=1,2...), but the *Fourier spectrum with a constant step* option enables to obtain graphs with a user-defined frequency step.

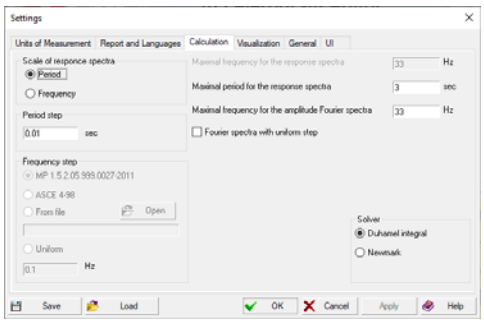


Figure 33.9-2. *The* **Calculation** *tab*

The user can choose the type of scale of response spectra (period-acceleration or frequency-acceleration). If the frequency vs. acceleration relationship is generated, the user can select a set of frequency increments (according to the recommendations of MP 1.5.2.05.999.0027-2011 or ASCE 4-98). Moreover, the user can prepare a text file (with the **.txt** extension) with a list of frequencies, for which the values of the response spectra should be output. In order to load a file select the **From file** option and click the **Open** button. The file should contain the values of the respective frequencies (in Hz).

If when calculating the response spectra it turns out that the calculated frequency exceeds the maximum one specified in the file, the list of frequencies is supplemented based on the last increment specified by the user (the difference between the last two numbers). The frequencies in the file must be specified as floating-point numbers with a period serving as a separator between their integer and fractional parts. All numbers in the file must be separated by spaces or start from a new line. A uniform scale with a step specified in this tab can be also used.

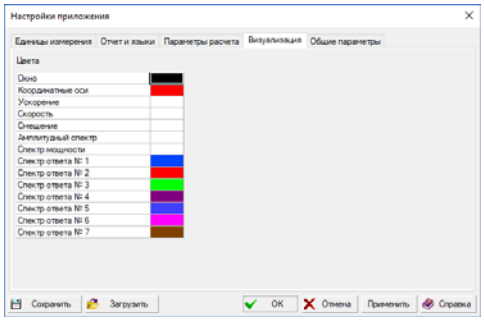


Figure 33.9-3. *The* **Visualization** *tab*

The **Visualization** tab (Fig. 33.9-3) contains controls for selecting the background color for the plot window (the **Window** row), colors of the coordinate axes, and of plots. To alter a color, place the pointer onto the row with the respective title and double-click to open a standard Windows color dialog.

Settings can be saved to an external file using the **Save** button, which can be subsequently loaded (the **Load** button).

Editing an Accelerogram

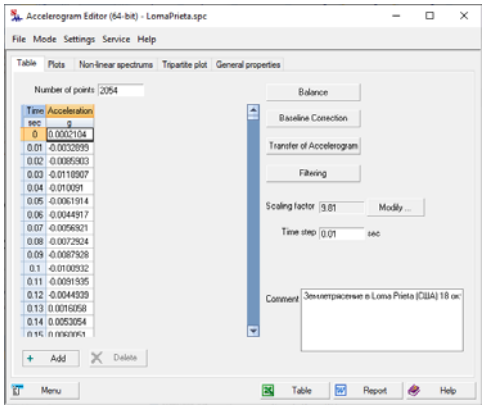


Figure 33.9-4. *The* **Table** *tab*

An accelerogram can be loaded using the **Open** item of the ***File*** menu. Choose a file to be loaded in the standard Windows **Open** dialog. The list of files in the dialog will contain only those with the **.spc** extension. After being loaded, the data of the accelerogram (time and acceleration) will appear in a table (Fig. 33.9-4). Data listed in the header of the accelerogram file will be displayed in the fields **Number of Points**, **Scaling Factor**, **Time Step** and **Comment**.

You can also import files of various formats with accelerograms in the Accelerogram **Correction** mode (menu ***File|Import***). The imported file is selected in the standard **Open dialog** box of the Windows environment. Then a dialog box appears, where you can select options specific to a particular format (several accelerations in one line, the number of the column with time, the number of the column with acceleration, the number of the first line with information on the accelerations , ...).

The editing of the accelerogram consists of changing the acceleration values, entering new lines, deleting lines, balancing, removing a trend, and filtering. The first three operations are done directly in the table. To add or delete a line, use respective buttons.

A new line is added before the one that has been selected. If none has been selected, the new line will be added to the end of the table. When deleting, all selected lines are removed from the table. The value in the **Number of Points** field will be adjusted automatically.

The **Modify** button invokes the dialog box where you can specify the accelerogram scaling factor. For example, if we have an accelerogram measured for the earthquake of magnitude 7 and we specify a factor of 2.0, we will obtain an accelerogram which corresponds to the earthquake of magnitude 8.

Clicking the **Balance** button will perform the balancing of the accelerogram.

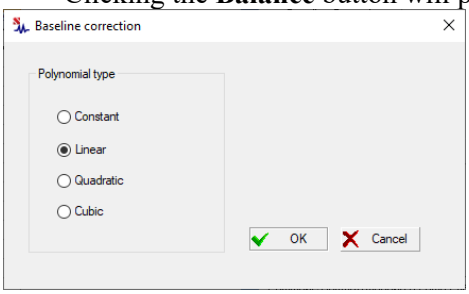


Figure 33.9-5. *The* **Remove Trend** *dialog box*

The **Remove trend** button (see Fig. 33.9-5) invokes the dialog box where you can select the algorithm that will be used to remove the trend (balance the baseline) of the accelerogram. You can use constants or polynomials up to the third order.

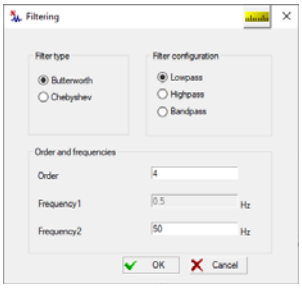


Figure 33.9-6. *The* **Filtering** *dialog box*

The **Filtering** button invokes the dialog box (see Fig. 33.9-6) where you can select the type of filter (Butterworth or Chebyshev), and its configuration (low-pass, high-pass or band-pass filter). You also have to specify the order of the filter and the cutoff frequency (two frequencies in the case of a bandpass filter).

The frequency filtering of the signal will be performed after clicking the **OK** button.

The application will recalculate a standard accelerogram for a certain level shifted from the measurement surface when you click the **Transfer of Accelerogram** button. This action will open a dialog box (see Fig. 33.9-7), where the user needs to specify data about the horizontally layered soil deposit, and a depth for which the new accelerations need to be calculated. The bottom of the lowermost soil layer is assumed to be the top of the bedrock.

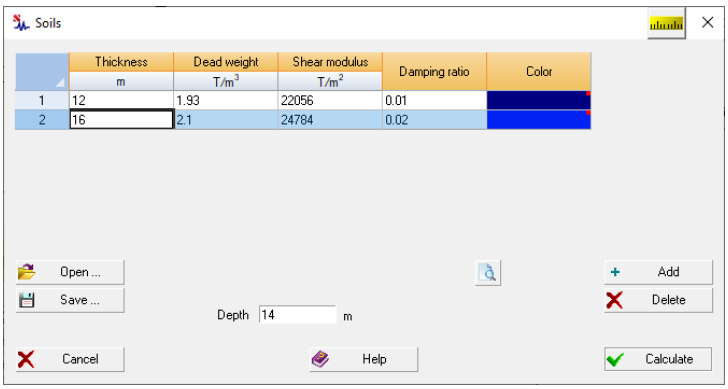


Figure 33.9-7. *The* **Soils** *dialog box*

The analysis is based on the following assumptions[1]:

- the soil profile is idealized as a system of homogeneous layers;

[1] The calculation method used in the program is described in the paper P.Schnabel, H.Bolton Seed, J.Lysmer, *Modification of seismograph records for effects of local soil conditions*, Bulletin of the Seismological Society of America, 1972, **62**, No. 6, (6): 1649–1664.

- each layer of soil is completely defined by its specific weight, shear modulus and damping ratio (these values are assumed to be independent of frequency);
- the responses in the soil profile are caused by the upward propagation of shear waves from the underlying rock formation;
- the relationship of the stresses vs. the shear modulus and the damping ratio is linear.

Creating Plots

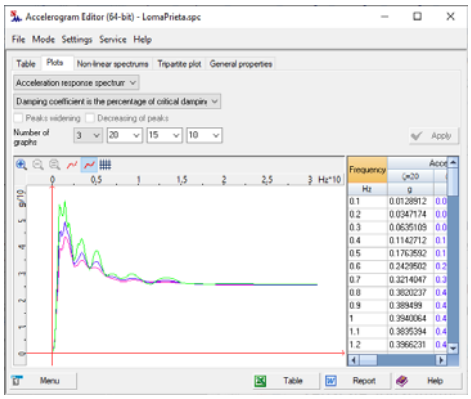


Figure 33.9-8. *The* **Plots** *tab*

Plots of accelerograms, velocigrams, and other functions are displayed in the **Plots** tab (Fig. 33.9-8). The plot type can be selected from a drop-down list located at the top of the tab. If the accelerogram is unbalanced, a warning will be displayed. There is a table of values of the factor being analyzed on the right from the plot.

When plotting the response spectra, you can choose the number of spectra to be calculated (up to 7) and specify a separate logarithmic decrement of oscillations for each. The decrements for each plot can be either entered explicitly in the text field of the respective drop-down list or selected from the set of standard values recommended by State Committee for Nuclear Regulation of Russian Federation [3] or IAEA [5]. If the values entered by the user are different from the standard ones presented in the lists, click the **Apply** button to see the change in the plot.

The plots will be displayed in different colors. The same colors are used to show the respective values in the table. If only one plot is displayed on the screen, the software enables to perform the procedure of expanding the peaks and reducing the maximums (see [1, Chapter 7]). If there is one plot, the **Save** button enables to save the response spectrum in the file with the **.spd** extension and to use this file as a plot of the dynamic amplification factor at the seismic analysis (when saving the **spd** file, the response spectrum is normalized to the maximum acceleration value of the accelerogram).

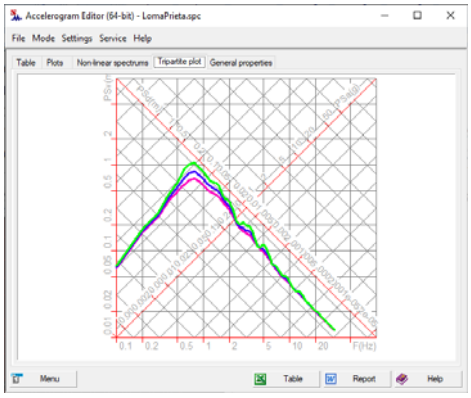


Figure 33.9-9. *The* **Tripartite Plot** *tab*

The application also enables to specify the information on the damping of oscillations specifying one of the following parameters:

- energy absorption factor;
- damping parameter (in fractions or percentages of the critical value);
- logarithmic decrement of oscillations;
- loss factor;
- coefficient of internal friction (inelastic resistance).

The respective item in the drop-down list enables to select the method for specifying the data on the internal losses.

If you place the cursor in the field of the graph, the information field with the values of the factors in the selected point will appear next to the cursor which enables to explore the graph dynamically.

The **Tripartite Plot** tab (see Fig. 33.9-9) also provides the three-axis pseudo-spectra plot.

Nonlinear Spectra

You can plot **nonlinear response spectra** (accelerations, velocities, and displacements) in a separate tab. This tab is similar to the **Plots** tab. The differences are that here you need to specify the damping parameter, the number of analyzed spectra (up to 7) and a special plasticity parameter μ for each of them. The non-linear spectra are calculated assuming ideal plasticity of the material. A plot of the linear response spectrum (which corresponds to μ=1) is added automatically.

General Properties

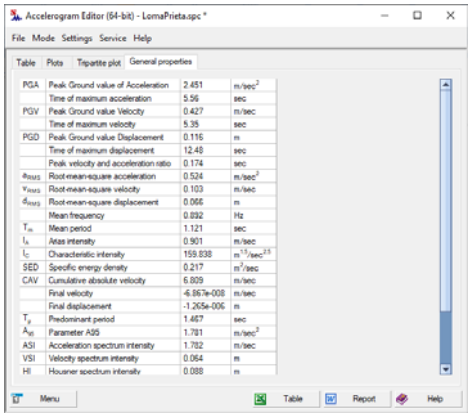


Figure 33.9-10. *The* **General Properties** *tab*

This tab (Fig. 33.9-10) displays general properties of the accelerogram.

Since different books and papers use different definitions of certain parameters, the following is a list of calculated characteristics with their definitions. We use the following designations:

g is the acceleration of gravity;

$a(t)$ is the acceleration at time t;

T is the total duration of the accelerogram;

$v(t)$ is the velocity at time t;

$S_a(\xi,t)$ is the acceleration response spectrum at time t with the damping parameter (in fractions of the critical value) equal to ξ;

$S_v(\xi,t)$ is the velocity response spectrum at time t with the damping parameter (in fractions of the critical value) equal to ξ;

$PS_a(\xi,t)$ is the pseudo-acceleration response spectrum at time t with the damping parameter (in fractions of the critical value) equal to ξ;

$PS_v(\xi,t)$ is the pseudo-velocity response spectrum at time t with the damping parameter (in fractions of the critical value) equal to ξ;

Designation	Name	Notes
PGA	Peak ground acceleration	
PGV	Peak ground velocity	
PGD	Peak ground displacement	
	Peak velocity and acceleration ratio	
a_{RMS}	Root-mean-square acceleration	
v_{RMS}	Root-mean-square velocity	
d_{RMS}	Root-mean-square displacement	
	Mean frequency	
T_m	Mean period	
T_p	Predominant period	Period corresponding to the maximum value of the Fourier amplitude spectrum
I_A	Arias intensity	$\frac{\pi}{2g}\int_0^T a(t)^2 dt$
I_c	Characteristic intensity	$(a_{RMS})^{\frac{3}{2}}\sqrt{T}$
SED	Specific energy density	$\int_0^T v(t)^2 dt$

CAV	Cumulative absolute velocity	$\int_0^T \lvert a(t) \rvert dt$
A_{95}	A95 parameter	Level of acceleration which contains up to 95 per cent of the Arias intensity.
ASI	Acceleration spectrum intensity	$\int_{0.1\ сек}^{0.5\ сек} S_a(\xi = 0.05, t) dt$
VSI	Velocity spectrum intensity	$\int_{0.1\ сек}^{0.5\ сек} S_v(\xi = 0.05, t) dt$
HI	Hausner intensity	$\int_{0.1\ сек}^{0.5\ сек} PS_v(\xi = 0.05, t) dt$
EPA	Effective peak acceleration	$\frac{1}{2.5} \underset{[0.1\ сек; 0.5\ сек]}{mean} S_a(\xi = 0.05)$

Envelopes

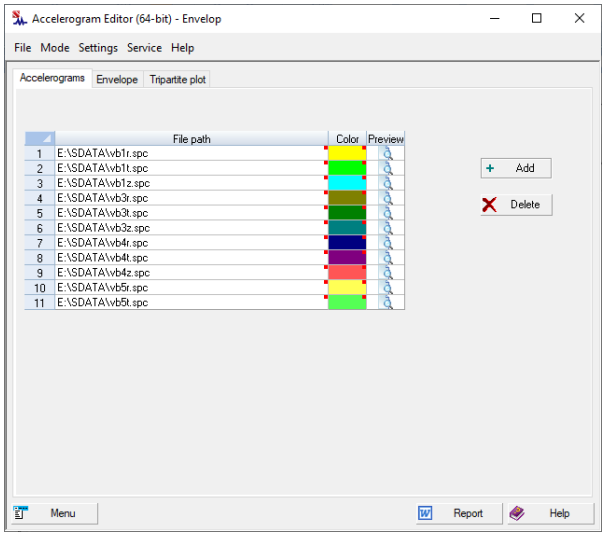


Figure 33.9-11. *The* **Accelerograms** *tab*

This mode enables to determine a response spectrum from a given set of accelerograms. You can create a list of accelerograms to be analyzed in the **Accelerograms** tab (see Fig. 33.9-11) using the **Add** button. Clicking this button invokes the standard **Open** dialog box, where you can select one or more accelerogram files. If a file has been added by mistake, you can select it in the table and click the **Delete** button to remove it from the list. The set of selected files is displayed in the table. The first column of the table contains the path to the file, and the second one contains the color for displaying the respective acceleration response spectrum. Double-clicking on a cell in the second column invokes a color adjustment dialog box.

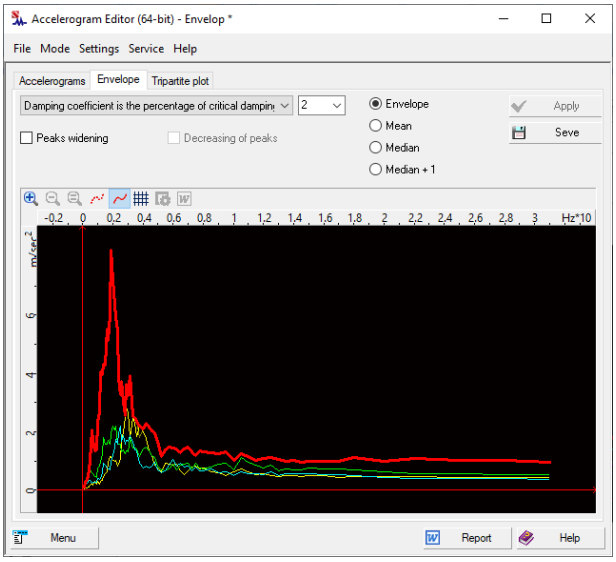


Figure 33.9-12. *The* **Envelope** *tab*

Once the required set of accelerograms is formed, you can specify damping and obtain the envelope of the spectra in the **Envelope** tab (see Fig. 33.9-12). In addition to the envelope you can also obtain mean and median response spectra, and the so-called *median+1* (median + standard deviation), which can be selected using the respective radio buttons in this tab. Regardless of the type selected for the resulting spectrum, you can expand the peaks and reduce the maximums by checking the respective checkboxes. The **Save** button performs the same function as in the **Edit** mode — generating the plot of the dynamic amplification factor.

It should be noted that Eurocode recommends using the envelope spectrum when the number of accelerograms in the set is not less than three, and the mean spectrum when the number of accelerograms in the set is not less than seven. Recommendations for using the median spectrum are provided in the American standards for seismic analysis of safety-related nuclear structures (see ASCE/SEI 4-16).

Correlation

When using three-component accelerograms, as well as in other cases when several accelerograms are used in the analysis, the design standards (for example, RB-006-98) require ensuring the statistical independence of the accelerograms. In particular, according to RB-006-98, the absolute value of the correlation coefficients should not exceed 0.3. ASCE4-16 additionally requires that the average correlation coefficient over the entire set must not exceed 0.16.

This mode enables to analyze the cross-correlations of the accelerogram set. A set of accelerograms is formed in the **Accelerograms** tab, which is similar to the respective tab in the **Envelope** mode.

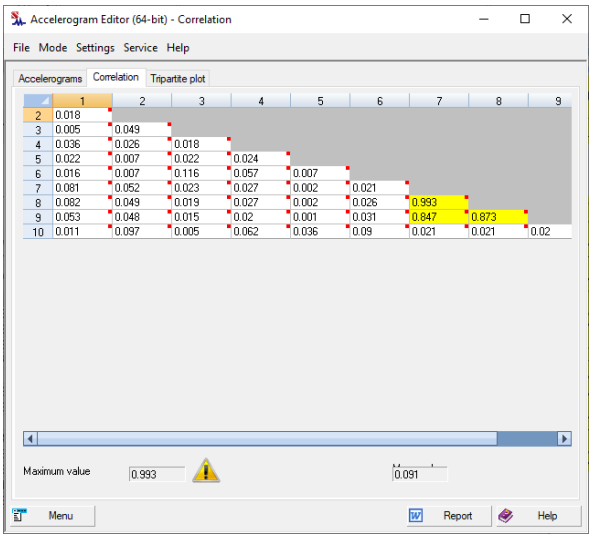


Figure 33.9-13. *The* **Correlation** *tab*

Once the required set of accelerograms is formed, a matrix with the coefficients of cross-correlation between all pairs of accelerograms included in the created set will be output (only the lower triangle of the matrix is shown since it is symmetric) in the **Correlation** tab (see Fig. 33.9-13). In addition, the maximum and average correlation coefficients are calculated for the entire set.

33.10. Editor of Dynamic Amplification Factor

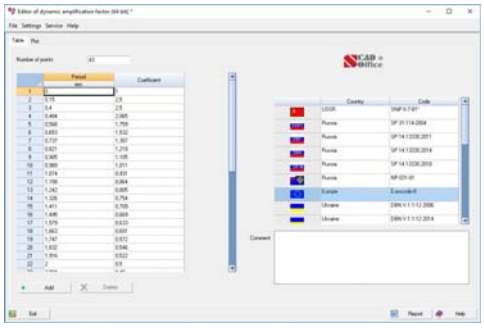


Figure 33.10-1. *The* **Table** *tab*

The structural analysis of seismic actions using a linear spectral theory that is stated in most design codes (such as SNiP II-07-81, PN AE G-5-006-87, NP-031-01, etc.) involves a plot of the dynamic amplification factor (DAF) β specified by the codes.

Recently there have appeared new regulatory codes such as those of the Republic of Kazakhstan or Georgia that make it necessary to use specific relationships between DAF and the natural oscillation period $\beta(t)$. A similar problem arises when local seismic peculiarities need to be taken into account. In this case the accelerograms of peculiar earthquakes of the region can be used to build the plots of the $\beta(t)$ function (such as long-period Romanian earthquakes).

Therefore, we supplemented **SCAD** with a DAF plot editor that enables the user to specify any plot of $\beta(t)$.

The DAF plot files have the **.spd** extension and are plain text files. There is an arbitrary text (comment) at the beginning of each file, which is followed by a separator character # and couples of numbers (a period and a factor). The factor and period values are floating-point numbers with a period

serving as a separator between their integer and fractional parts. All numbers in the file must be separated by spaces or start from a new line. An example of such a file can be found in Chapter 8.

The DAF plot can be created automatically in compliance with recommendations given in SNiP II-7-81* "Construction in seismic regions", PN-031-01 "Design regulations for earthquake-resistant nuclear power plants", Eurocode 8, etc. A code can be selected using one of the buttons located in the **Table** tab (Fig. 33.10-1) of the **Editor of DAF** window. Data needed for the plot can be specified in the **Parameters** (Fig. 33.10-2) dialog, and it depends on the selected code.

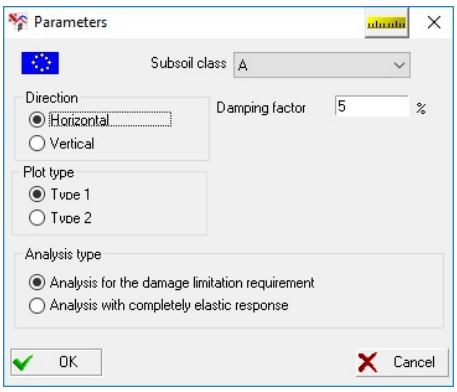


Figure 33.10-2. *The* **Parameters** *tab* (Eurocode 8)

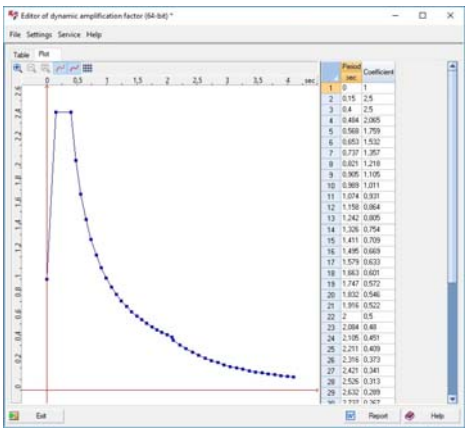


Figure 33.10-3. *The* **Plots** *tab*

When needed, the plot can be modified by editing data directly in the table; the allowed actions also include adding new lines and deleting the selected existing ones (the **Add** and **Delete** buttons). The obtained plot will be shown in the respective tab (Fig. 33.10-3). A comment can be saved together with the plot itself. The comment should be entered in the respective field in the **Table** tab.

If the DAF plot is built according to the rules different from those stated in the selected codes, then it can be specified by entering data into the table. To add lines in this case, use the **Add** button.

The data in the table must be arranged in the ascending order of the period values. There should not be points with equal period values.

If the plot has been created by a different tool such as a plain text editor, then a point must be used as a separator between the integer and fractional parts of the numbers. The period must be specified in seconds.

33.11. CoCon – Reference Manual

This application is an electronic reference manual for determining the stress concentration factors.

The following sources of stress concentration are considered:

- notches and grooves;
- shoulder fillets;
- circular holes;
- non-circular holes.

Moreover, the application also enables to calculate the stress intensity factors for different cases of the stress-strain state in the presence of cracks.

33.12. KUST – Design-Theoretical Reference Manual

The **KUST** reference program is designed for solving a certain class of mechanics problems for which there exist analytical or sufficiently accurate approximate solutions that can be found in the relevant literature. Though the majority of these problems can be solved with the help of **SCAD**, using the **KUST** program makes it possible to obtain a solution without creating design models. Moreover, some of the obtained results can be used to specify the initial data when building a finite-element model (e.g., effective length factors, estimation of natural frequencies, etc.).

All problems that can be solved by this program can be classified into the following categories:

- stability of equilibrium;

- natural vibration frequencies;
- other oscillation problems;
- static analysis;
- auxiliary calculations.

References to Chapter 33

1. A.N. Birbraer, Seismic analysis of structures. St. Petersburg. — Nauka. — 1998 — 255 p.
2. D.I. Mishin, Correcting accelerograms at the modeling of seismic actions. Earthquake-resistant construction. Safety of structures. — 2001— No. 2. — p. 17-22.
3. NP-031-01 Design codes for earthquake-resistant nuclear plants. — Moscow. — Gosatomnadzor of Russia. — 2001 — 31 p.
4. A.M. Uzdin, T.A. Sandovich, Al-Nasser Mohammad Samih Amin, Fundamentals of the theory of earthquake resistance and earthquake resistant construction of buildings and structures. St. Petersburg — B.E.Vedeneev VNIIG Publishing House. — 1993 — 176 p.
5. Safety Series No 50-SG-S1 (Rev. 1). *Earthquake and associated topics in relation to nuclear power plant siting. A safety guide*. International Atomic Energy Agency. —Vienna. — 1991.

34. Settings

Many parameters used by the software can be changed by the user. Some of these settings have already been described above, for example, color schemes (see Sec. 10.5), environment settings (see Sec. 10.4), calculation parameters (see Chapter 11), setting the list of reinforcement spacing (see Sec. 19.11). The setting of all these parameters can be invoked from the ***Settings*** menu.

Most calculation parameters are saved in the project file (for example, spr-file). If the dialog box for setting the parameters is invoked from the *Settings* menu when there are no open models, the saved values will be used for all newly created projects.

34.1. Units of Measurement

The program operates with numerous physical quantities having different dimensions. The units of measurement can be set for each of them which will be used for input and/or output. The multi-tab **Parameters** dialog box, which can be invoked by clicking the button in the toolbar or from the ***Settings*** menu, serves this purpose. Units of measurement for the initial data and the results are set in two different tabs – **Units of Measurement** and **Units of Measurement of Results**, which have a similar structure (Fig. 34.1-1).

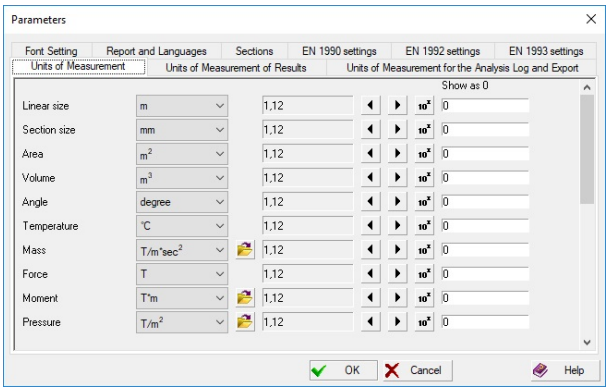


Figure. 34.1-1. *The* **Units of Measurement** *tab of the* **Parameters** *dialog box*

The **Units of Measurement** tab enables you to define units of measurement used in the analysis. It contains two groups of data. The first group is used to specify measurement units of linear sizes, forces, moments, etc.

For compound units (such as those of moment, stress, etc.), there is a possibility to define their component units (such as those for force and for moment arm) separately using the button .

The second group helps to choose a representation and precision of numerical data. Special controls are used here to select data representation formats. Make sure to specify the number of significant digits in either the fixed-point decimal representation or the floating-point scientific notation.

The precision of the data representation (the number of significant digits after the decimal point) can be assigned using the (decrease) and (increase) buttons, while the scientific notation is turned on by the button . You can also specify in respective text fields which values should be treated as negligibly small, so that all absolute values less than the given ones will be displayed as 0 in all visualizations.

Moreover, the dialog box has a tab where you can set the units of measurement which will be used when outputting the messages in the analysis log (for example, for the total loads) and when exporting the data.

34.2. Express Setting of the Units of Measurement

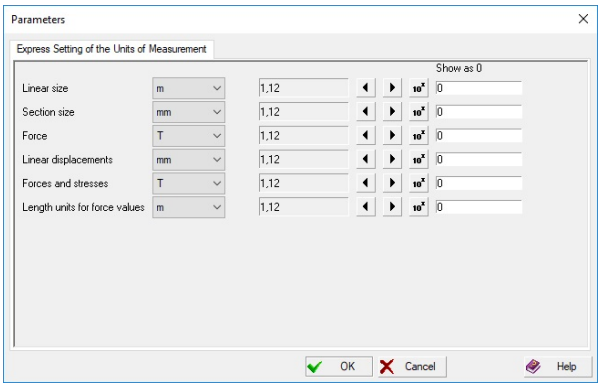


Figure 34.2-1. *The* **Express Setting of the Units of Measurement** *tab of the* **Parameters** *dialog box*

The express setting of the units of measurement which is invoked from the ***Settings*** menu can be used in those cases when it is necessary to change the units of measurement "quickly", setting them, for example, for all values related to the force factors (forces, moments, stresses,…). The dialog box used for these purposes (Fig. 34.2-1) is similar to the dialog box for setting the units of measurement (Fig. 34.1-1). The difference is that if we select, for example, N for the forces, N×m will be automatically selected for moments, and N/m^2 – for pressure etc.

34.3. Font Setting

Type, size, style and color of the font of the digital information on the design model are specified in the **Font Setting** tab (Fig. 34.3-1) of the **Parameters** dialog box, which can be invoked from the ***Settings*** menu (or by clicking the button in the toolbar).

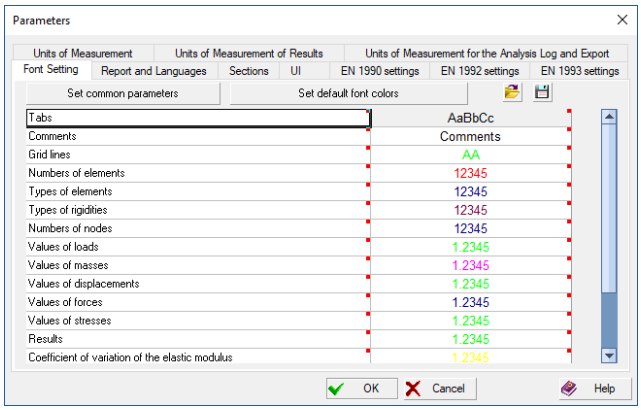


Figure 34.3-1. *The* **Font Setting** *tab of the* **Parameters** *dialog box*

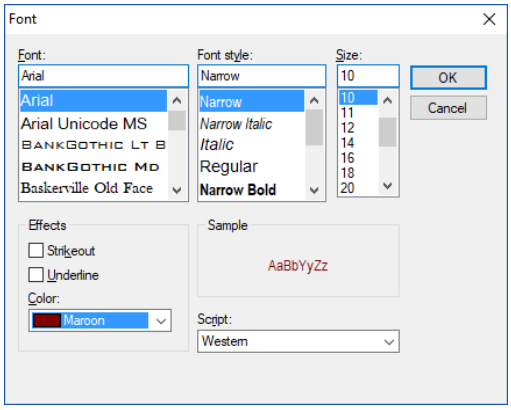


Figure 34.3-2. *The* **Font** *dialog box*

The buttons in the left column enable to select the type of the information for which the font settings are adjusted. Clicking these buttons invokes the standard **Font** dialog box of the Windows environment. You can specify the parameters of the font in this dialog box (Fig. 34.3-2). The font is displayed in the information field of the **Font Setting** dialog box. If you click the **Apply** button, the information will be output in the selected style in the model.

The **Report Print** font is used when displaying digital information in figures in reports in RTF, DOC, ... formats, and when displaying data on a color scale and saving a view in vector format (button — **Save the screen image).**

The **Set common parameters** button drops down a menu where you can select one of the following operations – **Set a single color for all fonts** and **Set a single font**.

The **Set default font colors** button enables to set the standard values of the font colors.

Font settings can be saved to an external file using the button , which can be subsequently loaded (the button).

34.4. Sections

If a limited set of the assortments of rolled profiles is used when working with the software, they can be included in the list of the available ones in the **Sections** tab (Fig. 34.4-1) of the multi-tab **Parameters** dialog box, which can be invoked from the **Settings** menu.

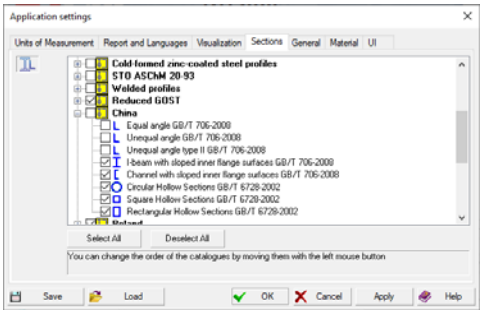


Figure 34.4-1. *The* **Sections** *tab*

There is a checkbox next to each group of catalogues. If the checkbox is not checked, the respective group of catalogues will not be available in the application. Catalogues can be arranged in any convenient order (the same order will be used in the lists or the dialog boxes for the profile selection). To move an item, drag it while holding down the right mouse button.

The full list of the assortments of rolled profiles supplied with the software package is given in the Appendix.

34.5. Report and Languages

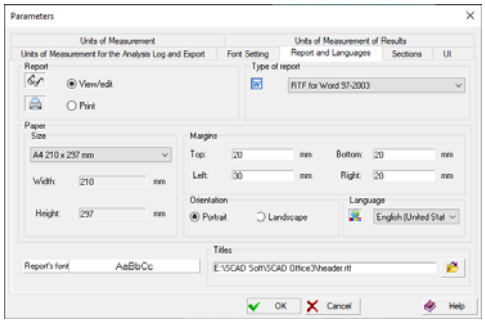


Figure 34.5-1. *The* **Report and Languages** *tab*

The **Report and Languages** tab (Fig. 34.5-1) enables you to choose a language for the user interface and for the report.

There are two modes for working with a report document: **View/Edit** or **Print**.

In the **View/Edit** mode, clicking the **Report** button in any active dialog will open the report and allow you to view/edit it. An application associated with **RTF** (Rich Text Format) files (such as MS Word Pad or MS Word) will be invoked to serve this purpose.

Obviously, it is the user who is fully responsible for any changes made to the text of the report (note that even results of the calculation can be edited).

There are differences in RTF formats used by MS Word v.7 and MS Word 97 (2000). Therefore, the program allows you to choose one of the formats in the **Type of Report** mode.

Clicking the **Print** button in the **Report** group will print the report in the form it has been generated by the program.

Use the **Titles** text field to specify an RTF file containing headers and footers for pages of the report document. The file can be selected from a standard list by clicking the button .

The **Paper Size** setting enables you to choose the paper format for printing the report (the size is selected from a drop-down list).

Moreover, the margins and the page orientation can be selected before generating the report.

Report Font is used to choose a font type for the report. A double left click invokes a standard Windows dialog for selecting the font. Only the selected font type will be used in the generated reports (style and size are assigned by the software).

34.6. UI

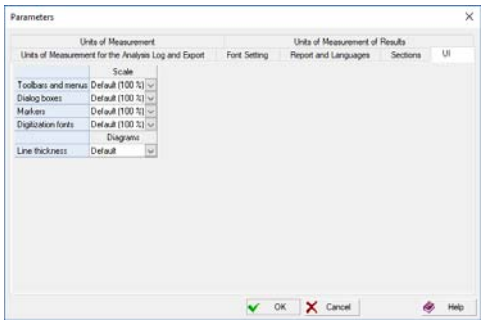


Figure 34.6-1. *The* **UI** *tab*

The **UI** tab (Fig. 34.6-1) enables to set scales for various user interface elements:

- toolbars and menus;
- dialog boxes;
- markers;
- digitization fonts.

The last two items are available only for the applications which can provide graphical display of models, for example, **SCAD** design models.

You can also set the default line thickness for the applications which can display graphs and diagrams.

The settings provided in this tab are intended primarily for users with HiDPI displays.

34.7. EN 1993 Settings

The **EN 1993 Settings** tab (Fig. 34.7-1) enables to select the partial safety factors in compliance with EN 1993-1-1 or in compliance with the National Annexes to Eurocode (NAD). Moreover, this tab enables to select the method for checking the general stability of the elements under bending and compression (select the respective Annex A or B of EN 1993-1-1), limit the execution of standard checks assuming that the deformations of steel are elastic, specify the parameter ρ_{lim} allowing for the effect of buckling of the section elements on the rigidity of the structure as a whole, select the method for allowing for shear lag for the ultimate limit state, specify the parameters η for checking the shear stability of webs, and to define the set of the used classes of bolts (see Fig. 34.7-1).

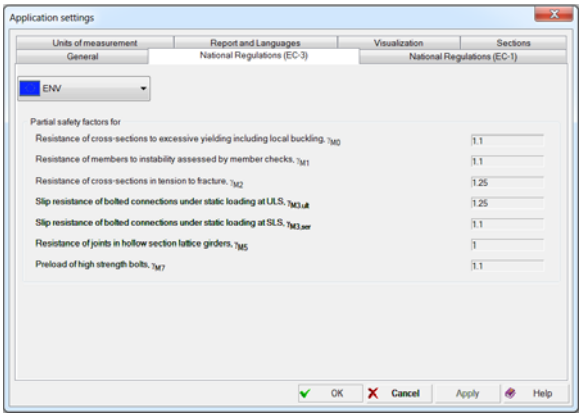


Figure 34.7-1. *The* **EN 1993 Settings** *tab of the* **Application Settings** *dialog box*

If the values of the partial safety factors and other standard parameters of the calculation are specified in compliance with the recommendations of EN 1993-1-1, then the row with the EU flag (Eurocode), which is selected by default, should be selected from the drop-down list. If you want to use the National Annexes, select the row with a flag of the respective state. The numerical values of the respective partial safety factors and the parameters of the standard calculation will be displayed in the table in the **EN 1993 Settings** tab.

If the values of the partial safety factors or the parameters of the standard calculation are specified by the user, then you have to select the **Other** item in the drop-down list. Thus you can use the application for custom, nonstandard situations, and also when any modifications are made in NAD.

34.8. Visualization Settings of the Result Tables

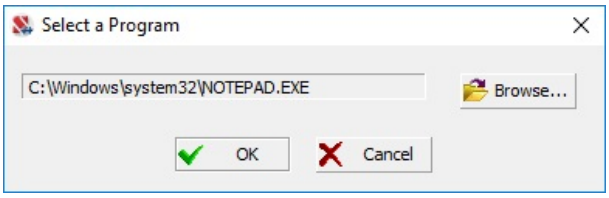


Figure 34.8-1. *The* **Select a Program** *dialog box*

SCAD invokes the Notepad text editor by default, which is supplied together with the Windows operating system, for the visualization of the text files, for example, created when documenting. But the user can specify another type of the text editor. In order to do it, use the ***Settings*** menu (**Visualization Settings of the Result Tables**) and select the necessary program after clicking the **Browse** button in the dialog box (Fig. 34.8-1).

34.9. Format Pages for Printing

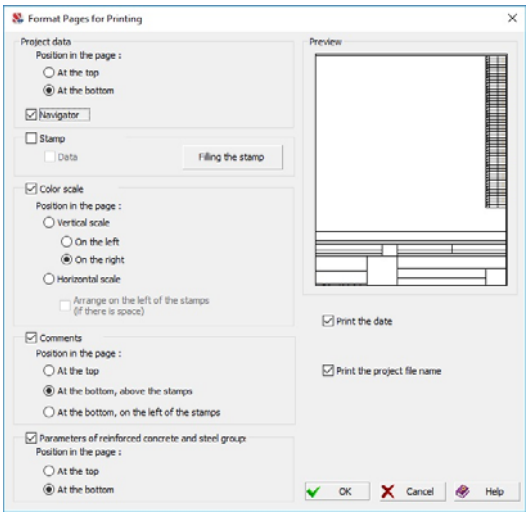


Figure 34.9-1. *The* **Format Pages for Printing** *dialog box*

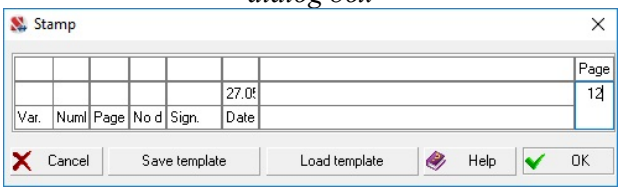


Figure 34.9-2. *The* **Stamp** *dialog box*

When printing the design model with the help of the command the software performs a certain formatting of the pages (outputs the frame and the stamp, displays the color scale, outputs the data on the project, etc.). The rules for formatting pages for printing can be customized by the user. In order to do it, use the **Format Pages for Printing** item in the ***Settings*** menu.

The dialog box will appear (Fig. 34.9-1), where you can:

- set the position of the data on the project in the page (at the top or at the bottom);
- invoke the navigator;
- check the checkbox indicating whether it is necessary to print the stamp and the data;
- set the rules for outputting the color scale (vertical/horizontal, on the left/on the right);
- select the position of the comments;
- specify the position of the parameters with the data on the structural group of steel or reinforced concrete structures.

Clicking the **Filling the stamp** button invokes the **Stamp** dialog box, where you can fill the stamp of the page (Fig. 34.9-2). The **Save as template** button enables to save the entered information as a template. The saved template can be used in the current and in other sessions of the program after clicking the **Load template** button.

34.10. Setting the Design Codes

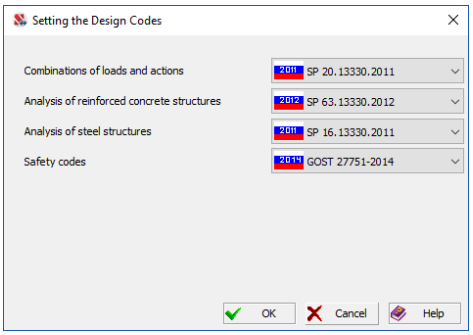


Figure 34.10-1. *The* **Setting the Design Codes** *dialog box*

This operation enables to change the design codes for:

- the combinations of loads and actions;
- analysis of reinforced concrete structures;
- analysis of steel structures;
- safety codes.

Select the codes from the drop-down lists in the dialog box (Fig. 34.10-1) and click the **OK** button.

34.11. Specifying Work Directories

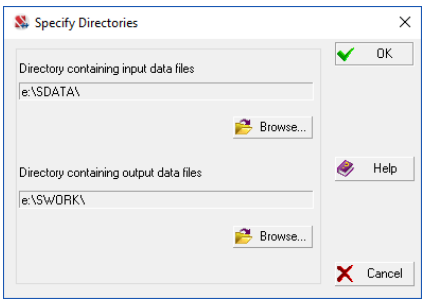


Figure 34.11-1. *The* **Specify Directories** *dialog box*

The **Specify Directories** dialog box invoked by this operation (Fig. 34.11-1) enables to specify the names of catalogues from which the project file should be taken and where the working files and the results should be put. You can change the purpose of catalogues only if the projects are not active, i.e. you have to close the active project before invoking the operation (the **Close Project** operation in the ***File*** menu).

34.12. Table Settings

If the software is running and there are no open models, the **View** menu contains the **Tables** submenu (Fig. 34.12-1), which enables to show or hide the tables of the display of filters, initial data and the results.

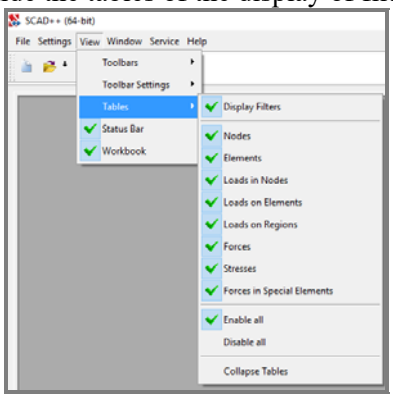


Figure 34.12-1. *The* **Tables** *menu*

Menu items **Select all** and **Deselect all** allow to show/hide all tables.

The **Collapse tables** menu item enables to switch to a mode whereby the filter tree and the tables will be hidden by default when you load any of the models. They can be opened by dragging the separator to the right.

34.13. Toolbar Settings

The **View** menu contains the **Toolbar Settings** submenu (Fig. 34.13-1), which enables to invoke the dialog boxes for setting the *Visualization* toolbar or the *Filters* toolbar.

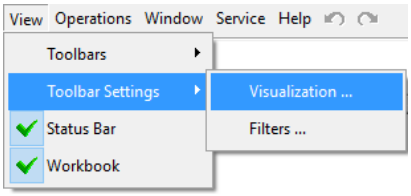


Figure 34.13-1. *The* **Toolbar Settings** *menu*

The **Toolbar Settings** dialog box appears (Fig. 34.13-2). It lists the buttons of the respective toolbar and contains the checkboxes which enable to specify the buttons that should be present in the toolbar.

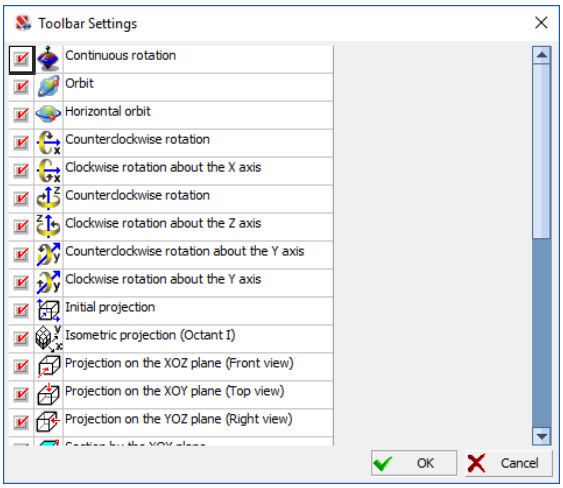


Figure 34.13-2. *The* **Toolbar Settings** *dialog box*

35. Import and Export SCAD Data

35.1. Import Design Models

A design model is usually not the only one, and definitely not the original model of the structure. It is preceded by the architectural models created with specialized software, called architectural design systems, which include ArchiCAD[1], Allplan[2], Revit[3], and a number of other specialized systems for 3D and 2D design of a certain class of structures, for example, AdvanceSteel. All of these systems create a more or less detailed model of the designed structure or of its part, which usually contains information on the geometric properties (shape, size) and sometimes on the materials the elements of the structure are made from. A mode of importing data from the above systems and a number of other systems used for the geometric modeling of structures, for example, from the graphics editors such as AutoCAD, is implemented in **SCAD**.

The ***Import*** item of the ***File*** menu enables to select the format of the imported data.

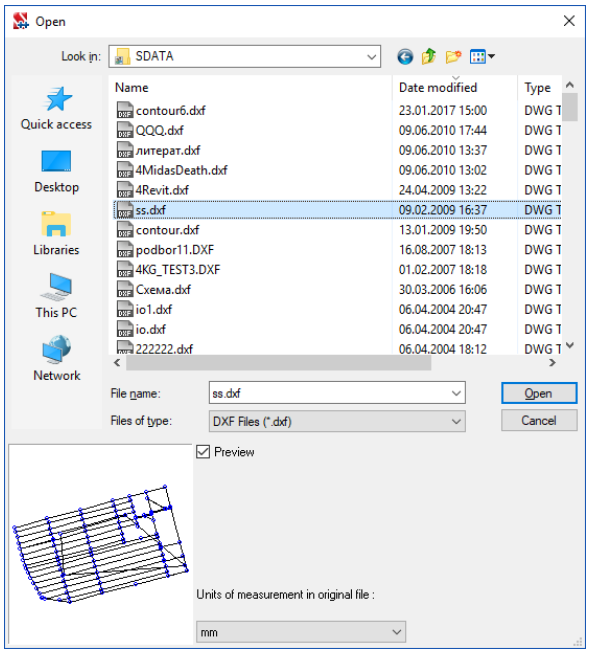


Figure 35.1-1. *Data format selection*

When importing files of a certain format a standard Windows dialog box for selecting the imported file appears. This dialog box provides the preview, and for some types of files which do not contain information on the linear units of measurement, a list appears where you can select the necessary measurement unit.

The types of finite elements in the design model are specified as follows for all the imported files:

- bar elements — type 5 (general type bar);
- quadrangular plate elements — four-node shell elements of type 44;
- triangular plate elements — three-node shell elements of type 42.

In most cases **SCAD** provides the “direct” reading of the files (for example, the DXF, IFC, SDNF files, etc.). However, in order to provide the connection with some software (Revit, ArchiCAD, Tekla, AllPlan) special extensions (plugin) have been developed, which work in the environment of the respective program and create an intermediate file with the extension **.r2s** in the result, which can be imported by **SCAD**.

[1] Registered trademark of the Graphisoft company.
[2] Registered trademark of the Nemetschek company.
[3] Registered trademark of the Autodesk company.

Once the user has selected a file for importing, the dialog box for creating a new model appears.

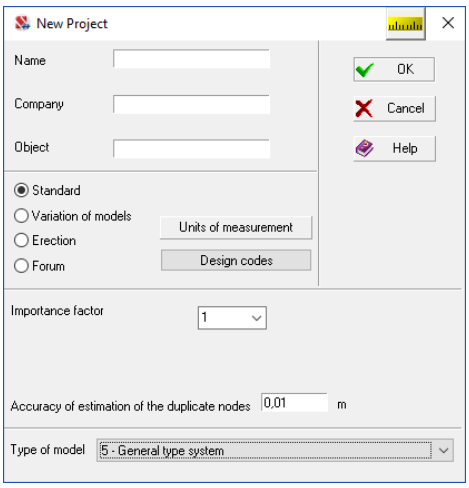


Figure 35.1-2. *Dialog box for creating a new model*

This dialog box (depending on the contents of the imported file) enables to select the type of the file obtained in the result of importing — **SPR** (finite element model) or **OPR** (enlarged model).

File of the project with the **.opr** extension contains the description of the enlarged structural model of the building. The transition from the architectural solution to the design model can be given as the following sequence of operations:

1) creating an architectural system of the enlarged structural model of the building consisting from such objects as columns, beams, walls, floors (slabs) and roofs on the basis of the internal data representation;
2) removing the objects which are not included in the design model from the structural model, for example, partitions, enclosing elements, architectural details etc.;
3) clarifying, if necessary, the position of objects in the structural model and supplementing it with new elements not taken into account in the architectural solution;
4) automatic or user-controlled generation of a finite element mesh with the simultaneous specification of the stiffness properties for the finite elements;
5) specification of the boundary conditions of elements;
6) specification of the loads and special initial data.

Problems dealing with the transfer of the data on sections of bar elements from rolled steel often arise when importing a model from the modeling systems. In those cases when the program of importing the data can not unambiguously determine the characteristics of the section and the assortment from which it should be selected, the user is prompted to select the respective profiles. These operations are performed in the **Matching Profiles** dialog box (Fig. 35.1-3).

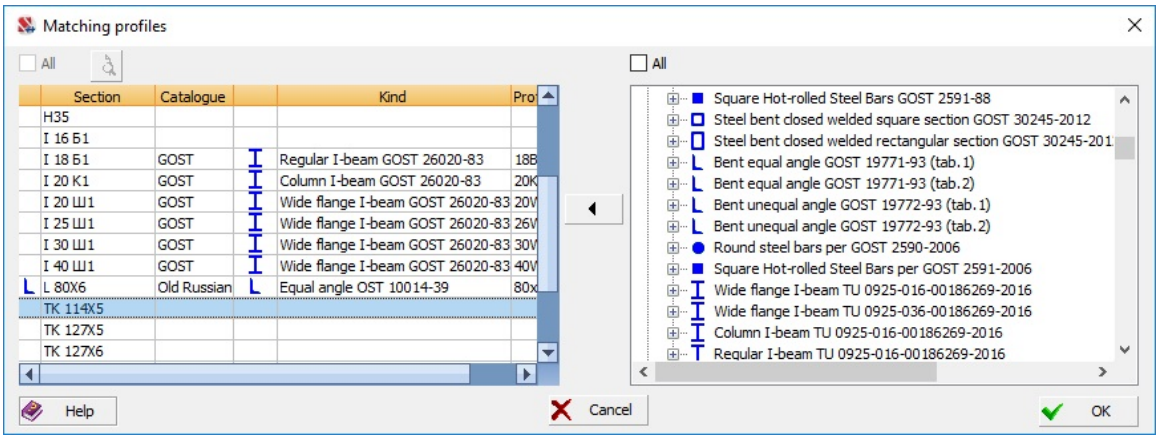


Figure 35.1-3. *The* **Matching Profiles** *dialog box*

The dialog box contains a table with the list of sections and a tree of assortments of rolled profiles. If the shape of the section and its sizes are recognized by the reader, an icon with the image of the section will be displayed in the first column of the table. In this case the tree of assortments will by default include only those assortments which include the sections of the similar type and size (if you want to get access to all the active assortments, check the **All** checkbox above the tree of assortments).

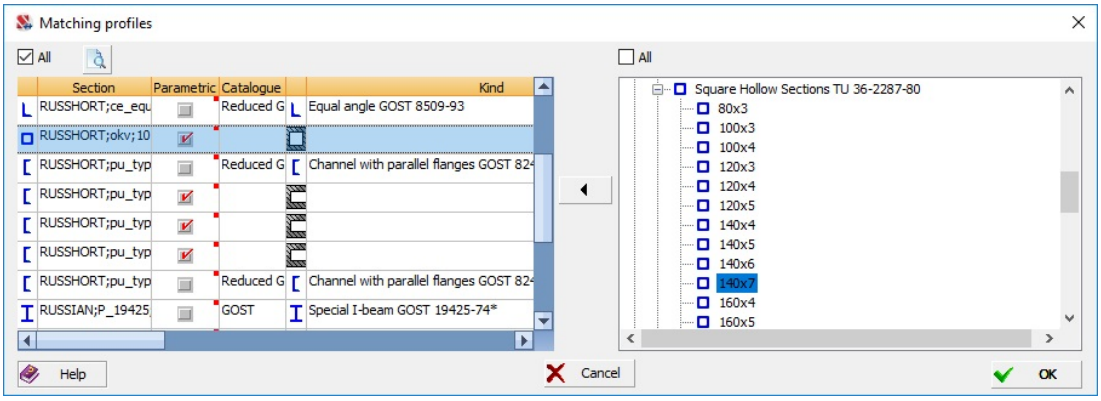


Figure 35.1-4. *The* **Matching Profiles** *dialog box* (representation of **parametric** sections)

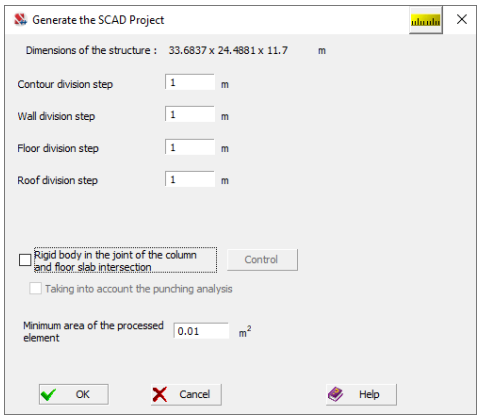
In those cases when the modeling system suggests an option of a parametric section, for example, I-beam with the respective sizes, the table in the **Matching Profiles** dialog box (Fig. 35.2-4) will contain a column with checkboxes. Checked checkboxes indicate sections which will be included in the design model as parametric. Otherwise, the section must be selected from the assortment.

The second column contains the name of the section from the imported model. It will be used in the design model as the name of the type of rigidity. It is necessary to perform the following actions to assign certain profiles from the assortment to the sections:

- select the line of the table with the name of the profile;
- select the necessary assortment of rolled profiles in the tree;
- open the list of the selected group of profiles;
- select the line with the necessary size of the profile;
- transfer the description of the selected profiles into the selected line of the table of sections by clicking the button ◀.

If a line with the specified or recognized profile is selected, you can use the **Preview** button — , clicking which invokes the dialog box with the image of the section.

If when importing **.r2s** files, the user has selected an option of creating a finite element model (creating an spr file), then a dialog box may appear during the import process where you have to specify the parameters for meshing plate elements.



This dialog box is similar to that used in the **Forum** preprocessor when generating a SCAD project (see Sec. 9.10).

Import Graphics Files

Import DXF, DWG Files

The description of the geometry of the design model can be imported from AutoCAD or other systems which support the formats of DWG- or DXF-files. The following types of graphics primitives are rendered:

3DFACE
SOLID
TRACE
LINE
POLYLINE
LWPOLYLINE
ELLIPSE
CIRCLE
ARC.

When importing DXF and DWG files it is possible to import "regular" files, and also to import files with nested modules.

Import Files in the 3D Studio Max Format

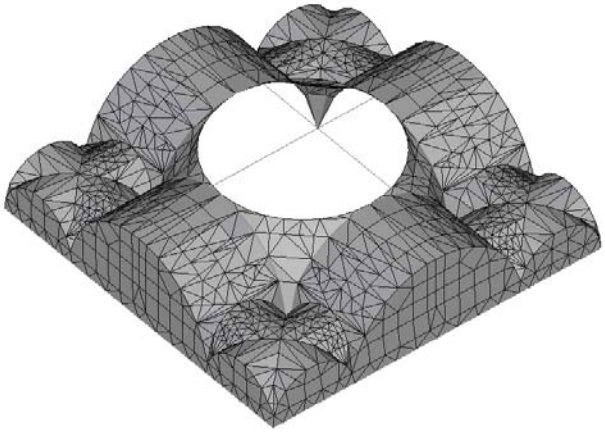


Figure 35.1-5. *Fragment imported from* Autodesk 3D Studio Max

The import of files in the format of Autodesk 3D Studio Max (extension **.3ds**) is implemented in the software, which enables to obtain the geometry of the design models of shell structures or their fragments. This system can be successfully used to generate geometric models of complex intersecting surfaces which are imported into **SCAD** as a three-dimensional mesh of three-node shell elements (Fig. 35.1-5).

Import Files in the SGI Invertor Format

Files in the SGI Invertor format (extension **.iv**) enable to transfer the information on the geometry of three-dimensional objects consisting from bars and shell elements into the software.

Import Files in the WaveFront Advanced Visualizer Format

Files of this type are used by the Advanced Visualizer application for storing the information on three-dimensional geometric objects. Data on bars and shell elements are transferred into the software.

Import Files in Other Formats

In addition to the formats described above, the geometric information can be imported in the following formats: Movie BYU Surface Geometry (extension **.byu**), ASCII Stereo Lithography (extension **.stl**) and Greg Hood ASCII Triangle Format (extension **.tri**).

Plugins

Revit

The plugin for Revit performs the export of the data on the model into the R2S format, which can be imported into **SCAD**.

The following data of the ***analytical*** model are transferred during the export.

Coordinates of nodes and constraints.

The information on geometry, orientation of the local axes, material, rigid inserts and hinges is transferred for *bar* elements. Curved bars are approximated by a set of ordinary bars.

The information on geometry (including the data on holes), thickness and the data on the material are exported for *plate* elements.

Only "Hosted" loads, i.e. those which are directly connected to the element or node, are taken into account when exporting *loadings*.

All loadings (with their names) are exported.

The following types of loads in the global coordinate system can be exported in each loading (Revit does not support the ability to specify loads in the local coordinate system of the element):

- nodal loads;
- concentrated loads on bars and plates;
- uniformly distributed and trapezoidal loads on bars;
- uniform and trapezoidal loads distributed along the line on plates;
- distributed and trapezoidal loads on plates.

ArchiCAD

The plugin for ArchiCAD performs the export of the data on the model into the R2S format, which can be imported into **SCAD**.

The following data are transferred during the export.

The information on geometry, orientation of the local axes is transferred for *bar* elements. The respective stiffness properties will be created for bars with a circular or rectangular cross-section during the export. For cross-sections with a shape other than rectangular a bar element will be created, but the cross-sectional shape will not be specified for it.

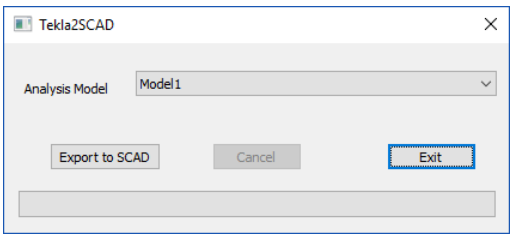
The information on geometry (including the data on holes) and thickness equal to the sum of thicknesses of bearing layers is exported for *plate* elements. Finishing layers of laminates are not taken into account neither in the calculation of thickness, nor in the calculation of the vertices of the plates. Curved plates are approximated by a set of ordinary plates. The average thickness is used for the plates with uneven thickness during the export.

Tekla

User Interface

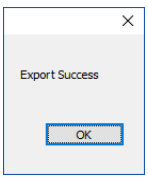
In order to use the plugin you have to create an analytical model with the help of the menu item Analysis-Analysis & Design Models-New.

In order to run the plugin go to the menu Detailing-Components-Component Catalog and activate the TeklaToSCAD component.



A dialog box will appear where you have to select a model created earlier, click the **Export to SCAD** button, and select the file name.

Once the data are exported, the following message will appear on the screen.



Export options

The plugin for Tekla performs the export of the data on the model into the R2S format, which can be imported into **SCAD**.

The following data of the model are transferred during the export.

Coordinates of nodes and imposed constraints.

The information on geometry (coordinates of the beginning and end of each element), orientation of the local axes, rigid inserts and hinges, physical properties of the material is transferred for *bar* elements. Curved bars are approximated by a set of ordinary bars. The following types of the cross-sections of bars can be exported — parametric, rolled steel sections and custom ones. A representation in the form of the **Consul** cross-section is created for the custom sections during the export. A list of correspondences between the profiles from the assortment of **SCAD** and Tekla is used during the export of rolled steel sections.

The information on geometry (including the data on holes), thickness and the data on the material are exported for *plate* elements.

The data on the rigid bodies can be exported as well.

All loadings (with their names) are exported.

The following types of loads can be exported in each loading:

- nodal loads (forces and moments);
- concentrated and distributed (uniform and trapezoidal) loads on bars in the global and local coordinate systems;
- concentrated loads on plates in the local coordinate system;
- loads distributed along the line on the side of the plate;
- uniformly distributed and trapezoidal loads on the plates in the local coordinate system;
- thermal loads on bars.

Limitations

Tekla describes loads on plates as a polygon and the values of forces or moments in each of its vertices. Loads distributed along the line are described as a segment and the values of forces or moments at the ends of the segment. These loads will be exported only if the coordinates of the vertices of the polygon (or segment) coincide with the coordinates of the vertices of the plate.

IFC

When importing files of the **IFC** format the formats of the 2x3, 4, 4x1, 4x2 versions are supported both in the text form, and in the **XML** format.

The data on coordinates and imposed constraints are imported for *nodes*.

The data on geometry (coordinates of the beginning and end), orientation of the local axes, hinges, and parameters of the cross-section are transferred for *bar* elements. A bar element can have several representations: geometric and analytical, preference is always given to the analytical one. Two types of sections are supported: parametric and sections in the form of polygons; the latter are implemented as a cross-section created by **Consul**.

The data on geometry (including the data on holes) and thickness are transferred for *plate* elements.

If the plate elements are represented as a set of polygons (IFC class - IfcFacetedBrep), the transformation into the analytical model is performed in the following way. Two “largest” faces are found. If these faces are parallel, they are considered as the top and bottom of the plate, the contour of the plate is assumed to have the shape of the larger face in the plane in the middle between the found upper and lower faces. If the selected faces are not parallel, such an element is ignored.

The following types of loads can be imported:

- nodal loads (forces and moments);
- uniformly distributed loads on bars.

If the model has plates with the number of nodes greater than 4, or plates with holes, a project of the “Standard” (*spr*) type cannot be created. In this case you can create only the project of the “Forum” (*opr*) type.

List of the main classes implemented by IFC:

- Nodes:
  - IfcStructuralPointConnection
- Bars:
  - IfcStructuralCurveMember
  - IfcStructuralCurveMemberVarying
  - IfcColumn
  - IfcBeam
  - IfcMember
- Plates:
  - IfcWall
  - IfcWallStandardCase
  - IfcPlate
  - IfcSlab
  - IfcRoof
  - IfcFooting
  - IfcOpeningElement
- Loads:
  - IfcStructuralLoad
  - IfcStructuralLoadStatic
  - IfcStructuralLoadLinearForce
  - IfcStructuralLoadSingleForce

CIS/2

The import of the CIS/2 format is implemented with the help of the conversion of the file of this format into the file of the IFC format and subsequent import of the IFC-file. Therefore, all limitations dealing with the IFC-format are applicable to the CIS/2 format as well and are described in the respective section.

SDNF

When importing files of the SDNF format, the 2.0 and 3.0 versions of this format are supported.

The information on geometry (coordinates of the beginning and end), orientation of the local axes, rigid inserts, hinges, and section is transferred for *bar* elements. The format implies that a rolled steel profile is used as a section; the profile is selected with the help of the special table of correspondences between the assortments of **SCAD** and Tekla.

The information on geometry (including the data on holes and fillets) and thickness is imported for *plate* elements.

The import of loads uniformly distributed over the whole length of the bar is implemented for *loadings*. The format does not imply that there are other types of loads.

If the model has plates with the number of nodes greater than 4, or plates with holes, a project of the “Standard” (*spr*) type cannot be created. In this case you can create only the project of the “Forum” (*opr*) type.

GMSH

When importing files of the GMSH format both text and binary types of this format are supported; and the formats starting from the 2.0 version are supported as well.

The data on the coordinates are imported for *nodes*.

The list of the main and additional nodes belonging to these elements can be read for all *finite* elements.

The following types of GMSH elements are supported:

- bars — types in GMSH: 1, 8, 26, 27, 28;
- 3-node plates — types in GMSH: 2, 9, 20, 21, 22, 23, 24, 25;
- 4- node plates — types in GMSH: 3, 10, 16;
- tetrahedrons — types in GMSH: 4, 11, 29, 30, 31;
- 6-node prisms — types in GMSH: 6, 13, 18;
- 8-node prisms — types in GMSH: 5, 12, 17, 92, 93, 32, 33, 7,14, 19.

GTC/GTCX/SMLX

The library of Graitec company or the library of the Autodesk company are used to import the files of the GTC, GTCX or SMLX format. In order to be able to import the files of the GTC, GTCX or SMLX format, the respective products of the Graitec company (AdvanceSteel and/or AdvanceConcrete) or Autodesk company (Autodesk AdvanceSteel and/or Autodesk AdvanceConcrete) must be installed on the computer.

The information on geometry (coordinates of the beginning and end), orientation of the local axes, rigid inserts, hinges, physical properties of the material, cross-section is imported for *bar* elements. Curved bars are approximated by a set of ordinary bars. Three types of sections are supported: rolled steel, parametric and compound sections. In the case of rolled steel and compound sections, a special table of correspondences between the assortments of AdvanceSteel and **SCAD** is used for selecting the profile.

The data on geometry (including holes and distortions of sides), thickness and physical properties of the material are imported for *plate* elements.

If the model has plates with the number of nodes greater than 4, or plates with holes, a project of the "Standard" (*spr*) type cannot be created. In this case you can create only the project of the "Forum" (*opr*) type.

Femap

The following is implemented when importing the files of the FNFF format (Femap neutral file format).

The format does not provide the possibility of specifying the units of measurement, therefore it is assumed that all the data are in the SI system.

The information on the coordinates and constraints is imported for *nodes*.

The data on geometry (numbers of the start and end nodes), orientation of the local axes, hinges, rigid inserts, properties of the material, parameters of the cross-section are imported for *bar* elements. In the case of a parametric section its sizes are transferred. Otherwise, only the area and the moments of inertia I_Y and I_Z are transferred. A suitable rolled steel section is selected in **SCAD** based on these data.

The data on geometry, properties of the material and thickness are imported for *plate* elements.

The data on geometry and the properties of the material are imported for *solid* elements.

The data on geometry (numbers of the first and second nodes), orientation of the local axes, and stiffness in all directions are imported for the *elastic constraints*.

The data on nodes and imposed constraints are imported for *rigid bodies*.

For all types of *loads*, except for the loads on nodes, only the forces are transferred, the moments are not provided by the format. The following types of loads can be imported:

- nodal loads;
- uniformly distributed and trapezoidal loads on bars in the local and global coordinate systems;
- uniformly distributed and trapezoidal loads on plates in the local and global coordinate systems;
- uniformly distributed loads on the faces of solid elements in the global coordinate system.

STAAD

When importing the files of the STAAD format the following data can be transferred.

Support of the commands: TO, REPEAT, REPEAT ALL.

The coordinates and constraints are transferred for *nodes*.

The information on geometry (numbers of the start and end nodes), orientation of the local axes, rigid inserts and hinges, physical properties of the material, parameters of the cross-section is transferred for *bar* elements.

The information on geometry (node numbers), thickness and the data on the material are imported for *plate* elements.

The data on geometry (node numbers) and the properties of the material are imported for *solid* elements.

The data on geometry (node numbers) and imposed constraints are imported for *rigid* bodies.

The information on parametric sections and rolled steel profiles is transferred for the *cross-sections* of bars. A list of correspondences between the profiles from the assortment of **SCAD** and STAAD is used during the import of rolled steel sections.

The following types of loads can be imported:

- nodal loads;
- concentrated loads on bars;
- uniformly distributed and trapezoidal loads on bars;
- triangular loads on bars (such loads are transformed into two trapezoidal ones);
- concentrated loads on plates;
- uniformly distributed and trapezoidal loads on plates;
- uniformly distributed loads on the faces of solid elements.

ABAQUS

The following is implemented when importing the files of the **Abaqus input file** format.

Coordinates and constraints are imported for *nodes*.

The data on geometry (numbers of the start and end nodes), orientation of the local axes, material, hinges, rigid inserts (through the eccentricities of sections), parameters of the cross-section are imported for *bar* elements. The format enables to create 3-node bars. If the middle node is not used by other elements, it is ignored, if it is used, the bar is divided in two.

The data on geometry, material, thickness and additional nodes are imported for *plate* elements.

The data on geometry, material and additional nodes are imported for *solid* elements.

The data on geometry (list of nodes) and imposed constraints are imported for *rigid* bodies.

The data on geometry (one or two nodes depending on the type of constraint) and stiffness properties are imported for the *elastic constraints*.

The data on the coefficient C_1 are transferred for the *subsoil parameters*, in the case of bars this parameter is specified for the local Z axis.

The *groups* of nodes and elements are imported.

The following types of the inertial properties are imported for the *dynamic actions*:

- masses and moments of inertia for nodes;
- concentrated and distributed masses for bar and plate elements.

The following types of loads are imported:

- loads in a node;
- uniform and trapezoidal loads on bars in the global and local coordinate systems;
- loads on plates uniformly distributed along the line;
- uniformly distributed and trapezoidal loads on plates in the global and local coordinate systems;
- uniformly distributed loads on plates in the local and global coordinate systems;
- uniformly distributed loads on the face of a solid element in the global coordinate system;
- uniformly distributed loads on solid elements in the global coordinate system.

  Thermal loads are not supported due to the different ways of their specification in ABAQUS and **SCAD**.

Some types of loads in ABAQUS can be specified in their own coordinate system. In such cases the load is transferred into the global coordinate system.

Since **SCAD** operates with values in the SI system, and the **Abaqus input file** format does not imply the possibility of specifying the dimension, an additional file with specified conversion factors is used during the import to recalculate the dimensional values. It is an ordinary text file but with the **.ccf** extension. All the dimensional values from the inp-file will be multiplied by the respective factor during the import. If a certain factor is not specified, it is assumed that it is equal to one. If there is no file with

factors, all factors are assumed to be equal to one. An individual ccf-file can be prepared for each imported ABAQUS model. The following rules for working with the ccf-file will be used during the import:

- **SCAD** will browse the directory with the data file (inp-file) to find a file with the same name but with the **.ccf** extension;
- if there is no such file, **SCAD** tries to find the *abaqusunits.ccf* file in this directory;
- if the *abaqusunits.ccf* file is not found, all factors are assumed to be equal to one.

ccf-file is a text file in the ANSI coding. It is necessary to observe the following rules when creating it:

- line breaks are denoted as 0x0D 0x0A;
- blank lines are allowed;
- maximum line length — 1000 characters;
- lines starting with ** (two stars in a row) are considered as comments;
- "." (period) is used as a decimal separator;
- each line with the description of the factor looks as follows: <name of the factor><space><value>[comment];
- the names of the factors are case-insensitive.

It is recommended to use Windows Notepad for creating and editing ccf-files. It enables to avoid problems with coding and line breaks.

The following factors are currently provided:

LENGTH — conversion factor for linear sizes, such as coordinates, sizes of sections, plate thickness, etc. **SCAD** implies that all linear sizes are given in meters.

FORCE — conversion factor for force. **SCAD** implies that all forces are given in newtons.

DENSITY — conversion factor for specific weight. Despite the fact that it is a derivative dimension, it is specified separately. It is used only for specifying the specific weight of materials. **SCAD** implies that the specific weight is given in N/m^3.

YOUNGMODULE — conversion factor for the Young's modulus. Despite the fact that it is a derivative dimension, it is specified separately. It is used only for specifying the Young's modulus of materials. **SCAD** implies that the Young's modulus is given in N/m^2.

ANGLES — conversion factor for angles. **SCAD** implies that all angles are given in degrees.

TEMPERATURE — conversion factor for temperatures. It is used for specifying the linear expansion coefficient of materials. **SCAD** implies that the temperature is given in degrees Celsius.

MASS — conversion factor for masses in the inertial loads: *Mass, *Rotary Inertia, *Nonstructural Mass.

All derivative dimensional values, for example, pressure or moment of forces, are recalculated with the help of these factors. The exceptions are: DENSITY, YOUNGMODULE, MASS.

An example of the contents of the ccf-file is given below, which enables to import the inp-file with the ABAQUS data "given" in millimeters, radians and kilograms.

```
**example.ccf
**example of a file with factors
Angles 0.0174532925199433
**Force
Force 9.8 convert kilograms to newtons
** Linear sizes
LENGTH 0.001 convert millimeters to meters
```

ANSYS

When importing the files of the ANSYS CDB format the following data can be transferred:

The coordinates and imposed constraints are transferred for *nodes*.

The data on geometry (numbers of nodes), orientation of the local axes, material, rigid inserts (through the eccentricities of sections), sections are imported for *bar* elements. The format enables to create 3-node bars. If the middle node is not used by other elements, it is ignored, if it is used, the bar is divided in two.

The information on geometry (node numbers), thickness and the data on the material are imported for *plate* elements.

The data on geometry (node numbers) and the properties of the material are imported for *solid* elements.

The data on geometry (node numbers) are imported for *rigid* bodies.

The data on geometry (node numbers) are imported for the *elastic constraints*.

The masses and moments of inertia for nodes are imported for the *dynamic actions*.

The following types of loads can be imported for *loadings*:

- nodal loads;
- concentrated loads on bars;
- uniform and trapezoidal loads on bars. Offsets from the ends of the bars are supported for the trapezoidal loads;
- uniformly distributed and trapezoidal loads on plates;
- uniformly distributed and trapezoidal loads on the sides of the plates;
- uniformly distributed loads on the faces of solid elements.

The support of loads with the help of surfaces is implemented as well (SURF elements). A finite element consisting from the same nodes as the loaded surface is selected for such loads, the load is applied to the found element.

SAF

SAF is a data format implemented by the authors of ARCHICAD to describe the analytical model. The following is implemented when importing the files of the SAF format:

Coordinates and constraints are imported for nodes.

The data on geometry (numbers of the start and end nodes), orientation of the local axes, material, hinges, rigid inserts (through the eccentricities of sections), parameters of the cross-section are imported for bar elements. Curved bars are approximated by a set of ordinary bars. Four types of sections are supported: rolled steel, parametric, compound sections and sections defined by a polygon with holes.

The data on geometry, material, midplane displacement, thickness are imported for plate elements. If regions of a different thickness are specified for the plate, then these regions are replaced with new plates, and a hole is created in the original plate.

The data on geometry (list of nodes) and imposed constraints are imported for rigid bodies.

The data on geometry (node numbers) and stiffness properties are imported for the elastic constraints.

The following types of *loads* can be imported:

- nodal loads;
- concentrated loads on bars;
- uniform and trapezoidal loads on bars.
- uniformly distributed and trapezoidal loads on plates;
- uniformly distributed and trapezoidal loads on the sides of the plates.

SLI

When importing the files of the SLI format the following data can be transferred.

The coordinates, constraints and belonging to the groups of merged displacements are transferred for *nodes*.

The information on geometry (numbers of the start and end nodes), orientation of the local axes, rigid inserts and hinges, physical properties of the material, parameters of the cross-section, subsoil parameters is transferred for *bar* elements.

The information on geometry (node numbers), thickness, the data on the material, subsoil parameters are imported for *plate* elements.

The data on geometry (node numbers) and the properties of the material are imported for *solid* elements.

The data on geometry (node numbers) and imposed constraints are imported for *rigid* bodies.

The information on parametric sections and rolled steel profiles is transferred for the *cross-sections* of bars.

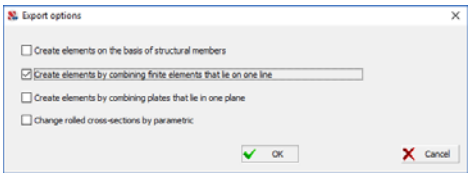
The following types of *loads* can be imported:

- nodal loads;
- concentrated loads on bars;
- uniformly distributed and trapezoidal loads on bars;
- concentrated loads on plates;
- uniformly distributed loads on plates;
- uniform heating of bars and plates;
- variation of the temperature across the section of the bar or across the thickness of the plate.

35.2. Export SCAD Data

The export of the data from **SCAD** is performed both in the formats of some design software (ABAQUS, STAAD, ...), and in the standard formats or in the formats which became "de facto" standard (SDNF, IFC, ...). A brief description of the possibilities of export is given below.

It should be noted, that for some formats (for example, GTC, IFC, SDNF, …) it is possible to combine finite elements into structural members. When the export operation is invoked, a dialog box appears where the user has to select the options which define the rules for creating structural members.

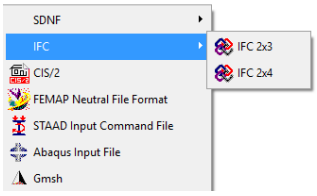


The following options are provided:

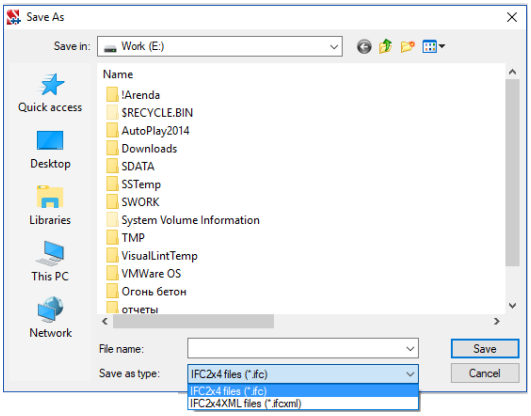
- creating structural members on the basis of those created in **SCAD** from rolled steel profiles;
- creating structural members by combining bars with the same rigidity that lie on one line and are successively adjacent to each other;
- creating structural members by combining plate finite elements with the same rigidity that lie in one plane, form a connected region, contacting by common nodes;
- rigidities can be transformed into the parametric form for the programs which do not provide the specification of the stiffness properties of bars on the basis of the rolled steel profiles (otherwise, only the values of the axial stiffness, bending stiffness etc. will be transferred).

IFC

When exporting a **SCAD** enlarged model into the IFC format, the 2x3 and 4 versions of this format are supported:



The export can be also performed into the text form or into the XML-format:



Depending on the position in space the type of bar — beam (horizontal element) or column (element that occupies any position in space except for the horizontal one), is specified during the export of bar elements. In terms of the IFC it corresponds to the IfcBeam or IfcColumn. The information on the coordinates of the beginning and end of the bar, orientation of the local axes and the cross-sectional shape is transferred. The type and sizes are exported for the parametrically defined cross-sections. The rolled steel profiles are transferred as polygons. A fictitious cross-section in the form of a square with a side of 10 cm is created for the bars with the type of rigidity incompatible with the IFC format (for example, the numerical description).

Depending on the position in space a vertical wall, inclined wall, floor, or an arbitrary flat element (in terms of the IFC — IfcWallStandardCase, IfcWall, IfcSlab or IfcPlate, respectively) is created during the export of plate elements. The information on geometry (including the data on holes) and thickness is transferred.

Limitations:

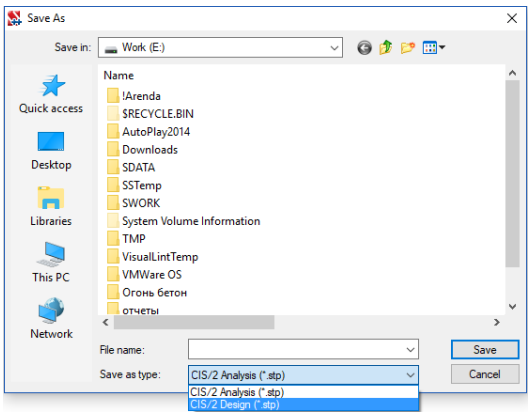
IFC does not provide the presence of solid elements.

IFC does not provide the specification of the majority of loads possible in **SCAD**, therefore the loads are not exported.

IFC does not provide a standardized method of specifying the physical properties of the material, therefore the materials are not exported.

CIS/2

When exporting an enlarged model created by **SCAD** into the CIS/2 format the user can select the type of the created CIS/2-model: analytical (Analysis) or graphical (Design).



The following data are exported when the **analytical** model is used:

- coordinates of nodes and imposed constraints;
- coordinates of the beginning and end of each bar, orientation of the local axes, hinges;
- the cross-sectional shape of the bar is transferred as a parametric description or as a polygon (if the rolled steel profiles were used in **SCAD**);

- a fictitious cross-section in the form of a 10×10 cm square is created for the bars with the type of rigidity incompatible with the CIS/2 format (for example, the numerical description);
- rigid inserts are modeled by an additional bar with the special properties of the material.

The following data are exported when the **graphical model** is used:

- coordinates of the beginning and end of each bar, orientation of the local axes, hinges;
- the cross-sectional shape of the bar is transferred as a parametric description or as a polygon (if the rolled steel profiles were used in **SCAD**);
- a fictitious cross-section in the form of a square with a side of 10 cm is created for the bars with the type of rigidity incompatible with the CIS/2 format (for example, the numerical description);
- rigid inserts are modeled by an additional bar with the special properties of the material;
- geometry and thickness of each plate finite element.

Limitations:

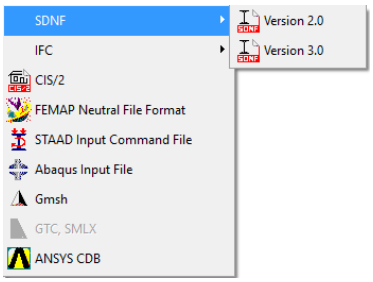
CIS/2 does not provide solid elements, therefore all the **SCAD** solid elements will not appear in the CIS/2-model after the export.

CIS/2 does not provide the specification of the majority of loads possible in **SCAD**, therefore the loads are not exported.

CIS/2 does not provide a standardized method of specifying the physical properties of the material, therefore the materials are not exported.

SDNF

When exporting a **SCAD** model into the SDNF format, the 2.0 and 3.0 versions of this format are supported:



The following data are exported:

- coordinates of the beginning and end of each bar;
- point of the orientation of the bar;
- rigid inserts and hinges in bars;
- the SDNF format implies the use of bar cross-sections only in the form of the rolled steel profiles. A special table of correspondences between the assortments of rolled profiles of **SCAD** and Tekla is used during the export. Only the information on the coordinates of the beginning and end of the finite element, hinges and rigid inserts is exported for the **SCAD** bar elements with the cross-sectional shape different from that of the rolled steel section;
- geometry (including the information on holes) and thickness of plate elements..

Limitations:

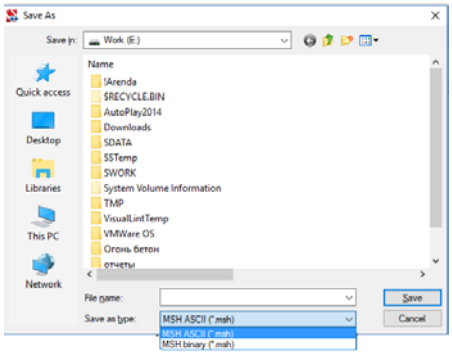
SDNF does not allow to transfer the information on solid and special finite elements.

Export of the load is not implemented, since only one type of load (trapezoidal the beginning and end of which coincide with the ends of the bar) is provided in the SDNF format.

SDNF does not provide the specification of the properties of the material, therefore the materials are not exported.

GMSH

The user can select the type of the created msh-file: text (ASCII) or binary (Binary), during the export of the model created by **SCAD** into the GMSH format:



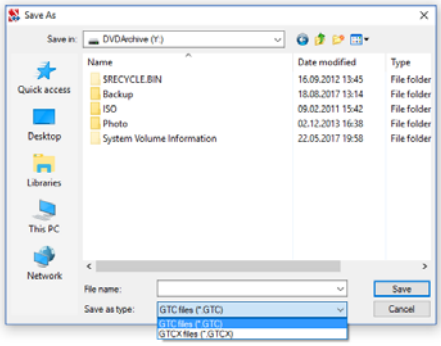
The information of the coordinates of nodes and the geometry of bar, plate and solid elements is exported.

Limitations:

Such data as loads, hinges, physical properties of materials etc. are not supported by the msh format and are not exported.

GTC/GNCX/SMLX

The library of Graitec company or the library of the Autodesk company are used to export the **SCAD** model into the GTC format. In order to be able to export into the GTC format, the respective products of the Graitec company (AdvanceSteel and/or AdvanceConcrete) or Autodesk company (Autodesk AdvanceSteel and/or Autodesk AdvanceConcrete) must be installed on the computer. The user can select the format for export (GTC, GTCX or SMLX):



The following parameters are transferred when exporting the data on the materials:

- name of the material (a file with the list of names known in AdvanceSteel and the respective properties of the materials are used when selecting the name);
- Young's modulus;
- Poisson's ratio;
- linear expansion coefficient;
- density;
- type: steel, concrete, timber, custom. During the export **SCAD** specifies the type on the basis of the analysis of the value of the Young's modulus.

Profiles for bars

The following parameters are transferred for profiles:

- sizes, for parametric sections;
- name, for rolled steel profiles. A list of correspondences between the profiles from the assortment of **SCAD** and AdvanceSteel is used during the export;
- name of the rolled steel profile and the distance between the profiles (for compound sections).

Finite elements

Bars

The following parameters can be transferred for bar finite elements:

- coordinates of the beginning and end of the bar;
- orientation;
- material;
- profile;
- hinges;
- rigid inserts.

Plates

The following parameters can be transferred for plate finite elements:

- geometry (including holes);
- thickness;
- material.

Limitations:

The GTC format does not provide the information on solid elements and loads on the elements.

Femap

The following data can be exported into the FNFF format (Femap neutral file format):

coordinates of nodes and imposed constraints;

the following information is transferred **for the bar finite elements** (they are assigned the element type number 5):

- node numbers;
- orientation of local axes;
- information on the rigid inserts;
- physical properties of the materials and the cross-sectional shape;

the following information is transferred **for the 3- and 4-node plate elements** (they are assigned the element type number 17):

- data on the thickness;
- physical properties of the materials;

physical properties of the material are transferred **for the solid finite elements** (they are assigned the element type number 25 with the following topological subtypes: number 6, 7 or 8 – for a tetrahedron, triangular prism and quadrangular prism respectively);

the node numbers and the directions of constraints are transferred **for the rigid bodies** (the element type number 29 is used);

the data on nodes and parameters of rigidity in all directions are transferred **for the elements modeling an elastic constraint between the nodes** (the element type number 6 is used).

The following geometric properties are transferred during the export of the data on the cross-sectional shape of the bar:

- area;
- perimeter;
- moments of inertia;
- torsional inertia moments;
- conventional shear areas.

The geometric dimensions are also transferred for the parametric sections. The FNFF format does not allow to transfer the information on the sections of bars with a different shape.

The following parameters are transferred during the export of data on the physical properties of the materials:

- type: isotropic or orthotropic;
- Young's modulus;
- Poisson's ratio;

- shear modulus;
- thermal expansion coefficient;
- density.

The following options are implemented during the export of loads:

- export of nodal forces and moments;
- transfer of the trapezoidal and uniformly distributed loads on the bars (in the local or global coordinate systems);
- transfer of the trapezoidal and uniformly distributed loads on the plates (in the local or global coordinate systems);
- export of the loads on the faces of the solid elements in the global coordinate system; the loads specified in the local coordinate system are transformed into the global coordinate system during the export.

Limitations:

The FNFF format does not allow to transfer the information on the loads on elements as moments.

The FNFF format does not provide the transfer of the load on a part of the bar, therefore it is assumed that such a load acts on the entire length of the bar.

The uniformly distributed loads on the solid elements are ignored.

STAAD

The following data are transferred during the export into STAAD:

coordinates of nodes and the imposed constraints;

the following information is transferred **for the bar finite elements**:

- node numbers;
- orientation of the local axes;
- information on the rigid inserts;
- physical properties of the materials;
- cross-sectional shape;

the following information is transferred **for the 3- and 4-node plate elements**:

- thickness;
- physical properties of the materials;

the following information is transferred **for the solid elements**:

- node numbers;
- physical properties of the materials;

the node numbers and the directions of constraints are transferred **during the export of rigid bodies**;

the following parameters are transferred during the export of the data on the physical properties of the materials:

- type: isotropic or orthotropic;
- Young's modulus;
- Poisson's ratio;
- shear modulus;
- thermal expansion coefficient;
- density.

The geometric dimensions of the parametric sections are transferred during the export of data on the cross-sectional shape. A special table of correspondences between the assortments of rolled profiles of **SCAD** and STAAD is used during the export. Other types of cross-sections of bars implemented in **SCAD** are not exported due to the limitations of the STAAD format.

The following data are transferred during the export of loads:

- nodal forces and moments;
- uniformly distributed, trapezoidal and concentrated (in the global and local coordinate systems) loads on bar finite elements;
- concentrated loads on plates;

- uniformly distributed and trapezoidal loads on plates;
- uniformly distributed loads on the face of solid finite elements.

Limitations:

The loads on elements specified as moments are not exported.

The loads on plates and solid finite elements in the local coordinate system are transformed into the global coordinate system during the export, because STAAD does not support the possibility of working with such loads in the local coordinate system.

ABAQUS

The following data are transferred during the export from **SCAD** into ABAQUS with the help of the Abaqus input file format:

coordinates of nodes and the imposed constraints (using the *Boundary ABAQUS section);

the following information is transferred for the bar finite elements:

- node numbers;
- orientation of the local axes;
- information on the physical properties of the materials;
- cross-sectional shape.

The transfer of the data on rigid inserts and hinges is not implemented, since ABAQUS does not have a complete analogue of these concepts. The correspondence between the types of bar elements is given in the table below.

Type of the **SCAD** finite element	Type of the ABAQUS finite element
1	T2D2
2, 3	B21
4	T3D2
other	B31

3- and 4-node plates are exported (including the axisymmetric finite elements). The data on the thickness and physical properties of the materials are exported as well. The correspondence between the types of plate elements is given in the table below.

Type of the **SCAD** finite element	Type of the ABAQUS finite element
61, 64	CAX4
62	CAX3
Three-node plates	S3
Four-node plates	S4

The data on the physical properties of the materials are exported for the **SCAD** solid finite elements. The correspondence between the types of solid elements is given in the table below.

Type of the **SCAD** finite element	Type of the ABAQUS finite element
Triangular prism (type 34)	C3D6
Quadrangular prism (type 36)	C3D8
Tetrahedron (type 32)	C3D4

The following table of correspondences is used when transferring the data on the cross-sectional shape of bar finite elements with a parametric form.

Type in **SCAD**	ABAQUS section	Type in **SCAD**	ABAQUS section
S0	RECT	S4	ARBITRARY
S1	I	S5	BOX
S2	I	S6	PIPE

S3	I	S7	GENERAL

Only the information on stiffness properties is transferred for all other parameters of rigidity of bars (implemented in **SCAD**).

The correspondence between the types of finite elements given in the table below is used during the export of the data on elastic constraints.

Type of the **SCAD** finite element	Type of the ABAQUS finite element
51	Spring1
55	SpringA

The data on the groups of nodes and groups of elements are exported.

Type in **SCAD**	Type in ABAQUS
Group of elements	*Elset
Group of nodes	*Nset

The subsoil parameters C_1 are exported with the help of the *FOUNDATION object of ABAQUS. Only the data for the local Z axis are transferred for bars (due to the absence of a complete analogue).

When transferring the information on the physical properties of the materials the transfer of the parameters of both isotropic and orthotropic materials is implemented. The following data are transferred:

SCAD	ABAQUS
Specific weight	*Density
Young's modulus	*Elastic
Poisson's ratio	*Elastic
Linear expansion coefficient	*Expansion

The following table contains the information on the export of the data on loads.

SCAD	ABAQUS
Static loads on nodes	
Nodal loads	*Cload
Static loads on elements	
Loads on bars	
Distributed, in the global coordinate system, along the X, Y, Z axes	PX, PY, PZ
Distributed, in the local coordinate system, along the Y, Z axes	P1, P2
Moments and other loads on bars	Not supported due to the absence of an analogue in ABAQUS
Loads on plates	
Distributed, in the local or global coordinate system, along the X, Y, Z axes	TRVEC
Distributed along the line in the local or global coordinate system, along the X, Y, Z axes	EDLD
Moments and other loads on plates	Not supported due to the absence of an analogue in ABAQUS

SCAD	ABAQUS
Loads on solid elements	
Distributed over the entire volume, in the global coordinate system, along the X, Y, Z axes	BX, BY, BZ
Distributed on the face in the global coordinate system, along the X, Y, Z axes	TRVEC
Dynamic loads on nodes	
Masses along the X, Y, Z axes	*MASS
Moments of inertia of the mass along the X, Y, Z axes	*Rotary Inertia
Dynamic loads on elements	
Distributed masses on bars	*Nonstructural Mass, units=MASS PER LENGTH
Distributed masses on plates	*Nonstructural Mass, units=MASS PER AREA
Distributed masses on solid elements	*Nonstructural Mass, units=MASS PER VOLUME

ANSYS

When exporting the data into the ANSYS CDB format the following data can be transferred:

Coordinates and imposed constraints are transferred for *nodes*.

The data on geometry (node numbers), orientation of the local axes, material, hinges, rigid inserts, sections are transferred for *bar* elements. The format does not provide the specification of hinges and rigid inserts directly on the element, therefore they are modeled with the help of the additional nodes and constraints between them.

The data on geometry, material, thickness, and additional nodes are transferred for *plate* elements.

The data on geometry, material, and additional nodes are exported for *solid* elements.

The data on geometry and imposed constraints are transferred for *rigid* bodies.

The masses and moments of inertia for nodes are transferred for the *dynamic actions*.

The following types of loads are transferred for *loadings*:

- loads in a node;
- uniform and trapezoidal loads on bars in the global and local coordinate systems;
- uniform and trapezoidal loads distributed along the line on plates in the global and local coordinate systems;
- uniformly distributed and trapezoidal loads for plates in the local and global coordinate systems;
- distributed loads on the face of a solid element in the global coordinate system.

Some loads can not be specified directly for a finite element, in such cases an additional SURF-element consisting from the same nodes as the given finite element is created, and the load is applied to it.

SAF

When exporting the data into the SAF format the following data can be transferred:

Coordinates and imposed constraints are transferred for nodes.

The data on geometry (numbers of the start and end nodes), orientation of the local axes, material, hinges, rigid inserts, sections are transferred for bar elements.

The data on geometry, material, thickness, and midplane displacement are transferred for plate elements.

Limitations:

The SAF format does not provide the information on solid elements.

Rigid bodies are not transferred because the format does not allow their full transfer.

The data on the following special elements are not transferred: peripheral finite elements, node dampers, ...

SLI

The following data are transferred during the export into SLI:

the following information is transferred **for the nodes**:

- coordinates of nodes;
- imposed constraints;
- belonging to the groups of merged displacements

the following information is transferred **for the bar finite elements**:

- node numbers;
- orientation of the local axes;
- information on the rigid inserts;
- information on the hinges;
- physical properties of the materials;
- cross-sectional shape;
- subsoil parameters

the following information is transferred **for the 3- and 4-node plate elements**:

- thickness;
- physical properties of the materials;
- subsoil parameters

the following information is transferred **for the solid elements**:

- node numbers;
- physical properties of the materials

the node numbers and the directions of constraints are transferred **during the export of rigid bodies**;

the following parameters are transferred **during the export of the data on the physical properties of the materials**:

- type: isotropic or orthotropic;
- Young's modulus;
- Poisson's ratio;
- shear modulus;
- density.

The geometric dimensions, area and moments of inertia are transferred **during the export of data on the cross-sectional shape**. Section name is also transferred for rolled steel profiles.

the following data are transferred **during the export of loads**:

- nodal forces and moments;
- uniformly distributed, trapezoidal and concentrated (in the global and local coordinate systems) loads on bar finite elements;
- concentrated loads on plates;
- uniformly distributed loads on plates;
- uniform heating of bars and plates;
- variation of the temperature across the section of the bar or across the thickness of the plate.

Limitations:

The loads on solid finite elements are not exported.

Appendix

List of Assortments of Rolled Profiles

Assortment name	File name in the root directory	Abbreviation for a text file
GOST	russian.prf	Russian
Reduced GOST	Russhort.prf	Russhort
Old Russian	Rus_old.prf	Rus_old
Assortment of the Chelyabinsk Metallurgical Plant (STO ASChM 20-93)	Aschm.prf	Aschm
British Standard Sections	British1.prf	British1
Overseas Shapes	British3.prf	British3
Welded profiles	WELD.PRF	WELD
OTUA	Otua.prf	Otua
DIN	Din.prf	Din
ARBED	ARBED.PRF	ARBED
ASTM	Usa.prf	Usa
Indian	India.prf	India
Japanese Steel	Japan.prf	Japan
Zinc coated steel wire ropes GOST	Cable.prf	cable
Rebars GOST	RebarRU.prf	RebarRu
Rebars of Belarus Republic	RebarBY.prf	RebarBY
Rebars ENV 10080:1996	RebarEC.prf	RebarEC
Poland	POLAND.PRF	POLAND
Thermal roll-formed steel profiles	TERM2007.PRF	TERM2007
Cold-formed profiles	TU1121.PRF	TU1121
Cold-formed zinc-coated steel profiles	TU112101.PRF	TU112101
Ukrainian cold-formed profiles	Ukraine.PRF	Ukraine

This table provides the names of the sections of the assortments of rolled profiles which are used in the text file:

Section name	Content
List of the sections of the full assortment of GOST profiles	
ce_equal	Equal angle GOST 8509-93
ci_uneq	Unequal angle GOST 8510-86*
pu_typep	Channel with parallel flanges GOST 8240-89
pu_vagon	Channel GOST 5267.1-90
pu_uklon	Channel with sloped inner flange surfaces GOST 8240-89
P_col_k	Column I-beam GOST 26020-83
P_uklon	I-beam with sloped inner flange surfaces GOST 8239-89
P_add_d	Additional series I-beam GOST 26020-83
P_norm_b	Regular I-beam GOST 26020-83
P_wide_h	Wide flange I-beam GOST 26020-83
tavr_KT	Column T-bar TU 14-2-685-86
tavr_ST	T-bar TU 14-2-685-86
diam	Longitudinally electric welded Circular Hollow Sections GOST 10704-91
diamold	Circular Hollow Sections GOST 10704-91 (reduced list)
pipe8732	Seamless hot finished steel Circular Hollow Sections GOST 8732-78
	Circular Hollow Sections GOST P 54157-2010
cg	Bent channels with equal flanges GOST 8278-83 (steels C239-C245)
cg2	Bent channels with equal flanges GOST 8278-83 (steels C255-C275)
OKV	Square Hollow Sections TU 36-2287-80

Section name	Content
ORECT	Rectangular Hollow Sections TU 67-2287-80
PU_Ec97	Economical channels with parallel flanges GOST 8240-97
PU_LT97	Light channels with parallel flanges GOST 8240-97
PU_Spc97	Special channels GOST 8240-97
PU_typ97	Channel with parallel flanges GOST 8240-97
PU_ukl97	Channel with sloped inner flange surfaces GOST 8240-97
PU_19425	Special steel channel GOST 19425-74*
P_19425	Special I-beam GOST 19425-74*
OKV66	Square Hollow Sections GOST 12336-66
OKV54157	Square Hollow Sections GOST P 54157-2010
ORECT66	Rectangular Hollow Sections GOST 12336-66
ORECT54157	Rectangular Hollow Sections GOST P 54157-2010
kv_truba	Square Steel Hollow Sections GOST 8639-68
pr_truba	Rectangular Steel Hollow Sections GOST 8645-68
OKV77	Square Hollow Sections GOST 25577-83*
ORECT77	Rectangular Hollow Sections GOST 25577-83*
OKV30245	Square Hollow Sections GOST 30245-94
REC30245	Rectangular Hollow Sections GOST 30245-94
okv2003	Steel bent closed welded square section GOST 30245-2003
orect2003	Steel bent closed welded rectangular section GOST 30245-2003
kv82_truba	Square Steel Hollow Sections GOST 8639-82
Round	Calibrated round steel GOST 7417-75
2590	Round Hot-rolled Steel Bars GOST 2590-88
S_259188	Square Hot-rolled Steel Bars GOST 2591-88
okv2012	Steel bent closed welded square section GOST 30245-2012
orct2012	Steel bent closed welded rectangular section GOST 30245-2012
19771_1	Bent equal angle GOST 19771-93 (tab.1)
19771_2	Bent equal angle GOST 19771-93 (tab.2)
19772_1	Bent unequal angle GOST 19772-93 (tab.1)
19772_2	Bent unequal angle GOST 19772-93 (tab.2)
259006	Round Hot-rolled Steel Bars GOST 2590-2006
S_259106	Square Hot-rolled Steel Bars GOST 2591-2006
Pipe1381	Circular Hollow Sections TU 1381-103-05757848-2013
DTV016SH	Wide flanged I-beam TU 0925-016-00186269-2016
DTV036SH	Wide flang I-beam TU 0925-036-00186269-2016
DTVR016K	Column I-beam TU 0925-016-00186269-2016
DTVR016S	Regular I-beam TU 0925-016-00186269-2016
DTVR016N	Regular I-beam TU 0925-016-00186269-2016
DTVR036K	Column I-beam TU 0925-036-00186269-2016
DTVR036N	Regular I-beam TU 0925-036-00186269-2016
DTVR036S	Pile I-beam TU 0925-036-00186269-2016
DTVR016U	Narrow I-beam TU 0925-016-00186269-2016
TU016_U	Narrow flange I-beam TU 24107-016-00186269-2017
TU016_N	Regular I-beam TU 24107-016-00186269-2017
TU016_Sr	Medium flange I-beam TU 24107-016-00186269-2017
TU016_Sh	Wide flange I-beam TU 24107-016-00186269-2017
TU016_K	Column I-beam TU 24107-016-00186269-2017
TU036_N	Regular I-beam TU 24107-036-00186269-2017
TU036_K	Column I-beam TU 24107-036-00186269-2017
TU036_Sh	Wide flange I-beam TU 24107-036-00186269-2017
TU036_Sv	Pile I-beam TU 24107-036-00186269-2017
G57837DB	Additional series I-beam GOST R 57837-2017
G57837DK	Additional series column I-beam GOST R 57837-2017

Section name	Content
G57837Sh	Wide flange I-beam GOST R 57837-2017
G57837_B	Regular I-beam GOST R 57837-2017
G57837_K	Column I-beam GOST R 57837-2017
G57837_S	Pile I-beam GOST R 57837-2017
C_32931	Circular Hollow Sections GOST 32931-2015
S_32931	Square Hollow Sections GOST 32931-2015
R_32931	Rectangular Hollow Sections GOST 32931-2015
C_58064	Circular Hollow Welded Sections GOST R 58064-2018
List of the sections of the reduced GOST	
ce_equal	Equal angle GOST 8509-93
ci_uneq	Unequal angle GOST 8510-86*
pu_typep	Channel with parallel flanges GOST 8240-89
P_col_k	Column I-beam GOST 26020-83
P_uklon	I-beam with sloped inner flange surfaces GOST 8239-89
Pu_uklon	Channel with sloped inner flange surfaces GOST 8240-89
P_add_d	Additional series I-beam GOST 26020-83
P_norm_b	Regular I-beam GOST 26020-83
P_wide_h	Wide flange I-beam GOST 26020-83
tavr_KT	Column T-bar TU 14-2-685-86
tavr_ST	T-bar TU 14-2-685-86
diam	Longitudinally electric welded Circular Hollow Sections GOST 10704-91
cg	Bent channel with equal flanges GOST 8278-83 (steels C239-C245)
cg2	Bent channels with equal flanges GOST 8278-83 (steels C255-C275)
OKV	Square Hollow Sections TU 36-2287-80
ORECT	Rectangular Hollow Sections TU 67-2287-80
List of the sections of the assortment of the Chelyabinsk Metallurgical Plant	
d1	Regular I-beam STO ASChM 20-93
d2	Wide flange I-beam STO ASChM 20-93
d3	Column I-beam STO ASChM 20-93
d4	I-beam R 40-93
d5	Special I-beam STO ASChM 20-93
List of the sections of the old assortments	
l1	Equal angle OST 14-1926
l2	Equal angle OST 14-1932
L3	Unequal angle OST 15-1926
L4	Unequal angle OST 15-1932
i1	I-beam OST 16-1926
I2	I-beam OST 16-1932
c1	Channel with sloped inner flange surfaces OST 17-1926
C2	Channel with sloped inner flange surfaces OST 17-1933
Dtavr39	I-beams OST 10016-39
Channel39	Channel OST 10017-39
Equal39	Equal angle OST 10014-39
EnEqual39	Unequal angle OST 10015-39
IPB	IPB Shapes DIN 1025
IPB1	IPB1 Shapes DIN 1025
IPBv	IPBv Shapes DIN 1025
IPE	IPE Shapes DIN 1025
IPEo	IPEo Shapes DIN 1025
IPEv	IPEv Shapes DIN 1025
List of the sections of the assortment of the British Standard Sections	

Section name	Content
ub_bs4	Universal Beams BS 4-1:1993
uc_bs4	Universal Columns BS 4-1:1993
ubp_bs4	Universal Bearing Piles BS 4-1:1993
rhs_uk	Rectangular Hollow Section
chs_uk	Circular Hollow Section
shs_uk	Square Hollow Section
joists	Joists
equal	Equal angle BS EN 10056-1:1999
unequal	Unequal angle
channel	Channel
t_ub_bs4	Structural Tees cut from UB's
t_uc_bs4	Structural Tees cut from UC's
Circular	Circular Hollow Sections EN10219-2:2006(E)
Square	Square Hollow Sections EN10219-2:2006 (E)
Rectangl	Rectangular Hollow Sections EN10219-2:2006 (E)
List of the sections of the assortment of the Overseas Shapes	
astm	ASTM W Shapes (Universal beams and columns)
ipe	IPE Shapes (European universal beams)
he	HE Shapes (European universal beams and columns)
rhs_os	Rectangular Hollow Section EN10210 (OS)
chs_os	Circular Hollow Section EN10210 (OS)
shs_os	Square Hollow Section EN10210 (OS)
List of the sections of OTUA	
ce_5677	Equal angle (NF A 45-009)
ci_5778	Unequal angle (NF A 45-010)
P_ipn	IPN Shapes (NF A 45-209)
P_pa	PA Shapes (NF A 45-205)
P_ipea	IPE-A Shapes (NF A 45-205)
P_ipe	IPE Shapes (NF A 45-205)
P_iper	IPE-R Shapes (NF A 45-205)
P_heaa	HEA-A Shapes (NF A 45-201)
P_hea	HEA Shapes (NF A 45-201)
P_heb	HEB Shapes (NF A 45-201)
P_hem	HEM, HEC Shapes (NF A 45-201)
Dp_pa	Structural Tees cut from PA (NF A 45-205)
Dp_ipea	Structural Tees cut from IPE-A (NF A 45-205)
Dp_ipe	Structural Tees cut from IPE (NF A 45-205)
Dp_iper	Structural Tees cut from IPE-R (NF A 45-205)
Dp_heaa	Structural Tees cut from HEA-A (NF A 45-211)
Dp_hea	Structural Tees cut from HEA (NF A 45-201)
Dp_heb	Structural Tees cut from HEB (NF A 45-201)
Dp_hem	Structural Tees cut from HEM, HEC (NF A 45-201)
Upn	Channel UPN (NF A 45-202)
Upna	Channel UPN-A (NF A 45-202)
Uap	Channel UAP (NF A 45-255)
Uapa	Channel UAP-A (NF A 45-255)
round	Circular Hollow Section
rect	Rectangular Hollow Section
square	Square Hollow Section
List of the sections of DIN	
Equal	Equal angle DIN 1028
Unequal	Unequal angle DIN 1029

Section name	Content
Dtavr_1	Beam DIN 1025
Dtavr_2	IP DIN 1025
Dtavr_3	IP DIN 1025 (9%)
Chanel	Channels DIN 1026
Round	Circular Hollow Section DIN 2448
Rect	Rectangular Hollow Section DIN 59410
Square	Square Hollow Section DIN 59410
List of the sections of ARBED	
ARB_Equal	Equal angle Euronorm 56-77
ARB_Unequal	Unequal angle Euronorm 57-78
ARB_ipe	European I-beams (IPE)
ARB_ipn	European standard beams (IPN) NF A 45-209
ARB_he	European wide flange beams (HE)
ARB_hl	European wide flange beams (HL)
ARB_hd	Wide flange columns (HD)
ARB_hp	Wide flange bearing piles (HP)
ARB_w	American wide flange beams (W)
ARB_ub	British universal beams (UB) BS 4-1:1993
ARB_uc	British universal columns (UC) BS 4-1:1993
ARB_uap	Channels with parallel flanges NF A 45-255
ARB_upn	European standard channels NF A 45-202
List of the sections of ASTM	
Equal	Equal angle
Unequal	Unequal angle
Dtavr_HP	H-Piles ASTM A6/A6M-98
Dtavr_M	Miscellaneous Shapes
Dtavr_S	American Standard Shapes
Dtavr_W	Wide Flange Shapes ASTM A6/A6M-98
Tavr_MT	Miscellaneous Tees
Tavr_ST	American Standard Tees
Tavr_WT	Wide Flange Tees
Sh_C	American Standard Channels
Sh_MC	Miscellaneous Channels
Tube_P	Pipe
Tube_PX	Extra Strong Pipe
Tube_PXX	Double-Extra Strong Pipe
Tube_TS	Tube Steel (Square)
Tube_TR	Tube Steel (Rectangular)
List of the sections of welded profiles	
tv	Welded I-beam TU U 01412851.001-95
P_NORM_B	Welded I-beam instead of GOST 26020-83 (TU 0925-001-97638531-2016)
D1	Welded I-beam instead of STO ASChM 20-93 (TU 0925-001-97638531-2016)
P_COL_K	Column I-beam instead of GOST 26020-83 (TU 0925-001-97638531-2016)
D3	Welded I-beam instaed of STO ASChM 20-93 (TU 0925-001-97638531-2016)
P_COL_KP_WIDE_H	Broad-flanged beam instead of GOST 26020-83 (TU 0925-001-97638531-2016)
D2	Broad-flanged beam instead of STO ASChM 20-93 (TU 0925-001-97638531-2016)
TU_CTO	Welded I-beam TU 0925-001-97638531-2016
D1_002.tot	Welded I-beam instead of STO ASChM 20-93 (TU 0925-002-62017235-2016)
D2_002.tot	Wide flange I-beam instead of STO ASChM 20-93 (TU 0925-002-62017235-2016)
D3_002.tot	Welded I-beam instead of STO ASChM 20-93 (TU 0925-002-62017235-2016)

Section name	Content
List of the sections of the Indian assortment	
Equal	Rolled steel equal angle
UnEqual	Unequal angle
ISHB	Rolled Steel Beams ISHB
ISJB	Rolled Steel Beams ISJB
ISLB	Rolled Steel Beams ISLB
ISMB	Rolled Steel Beams ISMB
ISJC	Rolled Steel Channels ISJC
ISLC	Rolled Steel Channels ISLC
ISMC	Rolled Steel Channels ISMC
Round	Circular Hollow Sections
List of the sections of the Japanese assortment	
E_Angles	Equal Angle
V_Angles	Unequal Angle
V2_Angles	Unequal Angle 2
SF_Shapes	Short Flange Shapes
MF_Shapes	Middle Flange Shapes
WF_Shapes	Wide Flange Shapes
Channel	Channel
SF_Tee	Short Flange Tee
MF_Tee	Middle Flange Tee
WF_Tee	Wide Flange Tee
Pipe	Circular Hollow Section
Rect_T	Rectangular Hollow Section
Square_T	Square Hollow Section
List of the sections of the assortment of GOST zinc coated steel wire ropes	
T2-36_1	Load-bearing closed wire rope with two layers of Z-shaped wire and a core of TK type GOST 18901-73*
T2-37_1	Double strand wire rope of LK-RO type. TU 14-4-902-78
T2-38_1	Load-bearing closed wire rope with one layer of Z-shaped wire and a core of TK type GOST 3090-73*
T2-38_2	Closed wire rope with a layer of wedge and Z-shaped wire and a core of TK type. GOST 7675-73*
T2-38_3	Closed wire rope with layers of wedge and Z-shaped wire and a core of TK type. GOST 7676-73*
T2-39_1	Closed spiral strand zinc coated steel wire ropes. TU 14-4-1216-82
T2-40_1	Single strand wire rope of LK-O type. GOST 3062-80*
T2-40_2	Single strand wire rope of TK type. GOST 3063-80*
T2-40_3	Single strand wire rope of T12 type. GOST 3064-80*
T2-40_4	Double strand wire rope of LK-O type. GOST 3066-80*
T2-40_5	Double strand wire rope of LK-3 type. GOST 7667-80*
T2-40_6	Double strand steel wire rope of DK-RO type. GOST 7669-80*
T2-41_1	Double strand wire rope of LK-R type. GOST 14954-80*
List of the sections of the assortment of GOST rebars	
A_1	Rebars GOST 5781-82*
A_2	Rebars GOST 5781-82*
A_3	Rebars GOST 5781-82*
A_4	Rebars GOST 5781-82*
A_5	Rebars GOST 5781-82*
A_6	Rebars GOST 5781-82*
At_7	Rebars GOST 10884-81
Bp_1	Rebars GOST 6727-80

Section name	Content
Bp_2	Rebars GOST 7348-81
B_2	Rebars GOST 7348-81
A400C	Rebars STO ASChM 7-93
A500C	Rebars STO ASChM 7-93
K_7	Cables GOST 13840-68
K_19	Cables TU 14-4-22-71
List of the sections of the assortment of Belarus Republic rebars	
S400-500	Rebars S400, S500 TU RB 190266671.001-2002
S400500	Rebars S400, S500 TU RB 04778771.001-97
S800	Rebars S800 TU RB 04778771.001-97
S500	Rebars S500 STB1341-2002
List of the sections of the assortment of ENV 10080:1996 rebars	
B500A	Rebars B500A ENV 10080:1996
B500B	Rebars B500B ENV 10080:1996
B450C	Rebars B450C ENV 10080:1996
Assortment of China	
Equal	Equal angle GB/T 706-2008
UnEqual	Unequal angle GB/T 706-2008
Angle2T	Unequal angle type II GB/T 706-2008
Dtavr	I-beam with sloped inner flange surfaces GB/T 706-2008
Channel	Channel with sloped inner flange surfaces GB/T 706-2008
Round	Circular Hollow Sections GB/T 6728-2002
Square	Square Hollow Sections GB/T 6728-2002
Rectangl	Rectangular Hollow Sections GB/T 6728-2002
Assortment of Poland	
Angle_Eq	Equal angle PN-EN 10056-1:2000
Equal	Equal angle PN-84/H-93401
IPE	I-beam PN-91/H-93419
HEA	Structural Tees cut from HEA PN-H-93452:2005
HEB	Wide flange I-beam PN-H-93452:2005
HEC	Structural Tees cut from HEC PN-H-93452:2005
Chanel	Channels PN-86/H-93403
Tube_car	Square Hollow Sections PN-EN 10219-2:2000
Thermal roll-formed steel profiles	
CTO_TPGC	Profile TPGC CTO 42481025 006-2007
CTO_TPP	Profile TPP CTO 42481025 006-2007
Cold-formed profiles	
TU1121U	Profile LP TU 1121-009-46216359-2010
TU1121S	Profile LC TU 1121-009-46216359-2010
TU1122_U	Profile U TU 1122-002-10836231-2014
TU1122_S	Profile C TU 1122-02-10836231-2014
TU1120_U	Profile U TU 1120-001-37820873-2012
TU1120_S	Profile C TU 1120-001-37820873-2012
TU1120S	Profile PC TU 1120-001-82861223-2009
G8282_83	Steel roll-formed C-shaped equal flange sections GOST 8282-83
Z13229N1	Steel bent Z-shaped sections GOST 13229-78 tab.3
Z13229N2	Steel bent Z-shaped sections GOST 13229-78 tab.4
Z13229R1	Steel bent Z-shaped sections GOST 13229-78 tab.1
Z13229R2	Steel bent Z-shaped sections GOST 13229-78 tab.2
G8281_T1	Steel roll-formed unequal channels GOST 8281-80 tab.1
G8281_T2	Steel roll-formed unequal channels GOST 8281-80 tab.2
G8283_T1	Bent steel hat equal flange sections GOST 8283-93 tab.1

Section name	Content
G8283_T2	Bent steel hat equal flange sections GOST 8283-93 tab.2
Cold-formed zinc-coated steel profiles	
TU1121PC	Profile PC TU 1121-001-13830080
TU1121PH	Profile PN TU 1121-001-13830080
TU1121TC	Profile TC TU 1121-001-13830080
TU1121TH	Profile TN TU 1121-001-13830080
TU1122AC	Profile AC TU 1122-002-82866678-2013
TU1121E1	Profile E TU 1121-001-99651760-2015 (steel 250/C255)
TU1121E2	Profile E TU 1121-001-99651760-2015 (steel 350/C345)
TU1122AE	Profile AE TU 1122-002-82866678-2013
TU1122Z1	Profile AZ TU 1122-002-82866678-2013 tab.2
TU1122Z2	Profile AZ TU 1122-002-82866678-2013 tab.3
Extruded of aluminium alloys GOST 17576-97	
Alum	Aluminium-alloy extruded profiles GOST 17576-97
Ukrainian cold-formed profiles	
Light_C	Profile C manufacture by Light House-Ukraine
Light_U	Profile U manufactured by Light House-Ukraine
Stal_C	Profile C manufactured by Stalex
Stal_Cw	Profile Cw manufactured by Stalex
Stal_U	Profile U manufactured by Stalex
Stal_Z	Steel bent Z-manufacture by Stalex
Stal_Zw	Steel bent Zw-manufacture by Stalex
tabl_C	Profile C manufacture by Pruszynski

Extensions of the Files in the Working Directory of SCAD

Table 1 provides the extensions for the files with the initial data used in the software.

Files of the initial data **Table 1**

Extension name	Content
.spr	Initial data of the **SCAD** project
.vpr	Initial data of the **SCAD** project with the variations of models
.opr	Initial data of the project of the **FORUM** type
.mpr	Initial data describing the erection of the structure (**Erection**)
.txt	Initial data of the **SCAD** project in the text format
.sec	Data on sections created by **Section Builder**
.cns, .con	Data on sections created by **Consul**
.tns	Data on sections created by **TONUS**
.crs	Data of **Cross**
.spc	Files with accelerograms
.spd	Files with the spectra of the dynamic amplification factors
.thi	File with the time function (for the direct integration mode)
.bak	File with the copy of the model in the state before the last saving of the data on the disk
.AutoSave **.AutoSave1** **.AutoSave2** **...**	Files with the copies of the data which is created by autosaving

Extension name	Content
.SCADModelColor	File with the data on the color scale settings
.SCADFilters	File with the filter settings
.SCADDocMaker	File with the data for generating a report
.SCADFonts	File with the fonts settings
.SCADMatrix	File with the model position

Files which can be divided into the following three groups depending on the type of the information they contain are saved in the working directory (**SWORK** by default) in the process of performing the calculation:

- working files (stiffness matrix and its right-hand sides, internal formats of finite elements etc.);
- results of analysis (displacements, forces, oscillation modes etc.);
- results of generating tables in the text format.

The first two groups contain the files in the binary format, and the third one — in the text format.

All files in the working directory are identified by the name of the problem file and a three-character extension, where the first letter **f** (see Table 2) is used for binary files, and **p** — for text files (see Table 3).

It is necessary to know the file names in order to transfer the problem together with the working files to another computer or to remove the unnecessary results from the working directory. Obviously, you can always delete the print files. If there is not enough disk space, you can also delete the file with the stiffness matrix (extension **.f01**), which usually has the largest size. The graphical analysis of the results of the calculation will remain accessible, but mode of continuing the calculation will not work.

When outputting the graphic files generated by the **Save the screen image** or **Save the video with the animation** operations, and also when creating documents for MS Word or MS Excel, the file names are specified by the user. Their extensions depend on the type of the information (**.wmf, .emf** — when saving the screen image, **.avi** — video file, **.rtf** — when exporting to MS Word, **.xls** — when exporting to MS Excel).

Binary files of the results **Table 2**

Extensions	Content
F00	Internal formats of finite elements after the control
F01	Stiffness matrix
F02	Working file for storing intermediate calculations
F03	Working file for storing intermediate calculations
F04	Natural frequencies of oscillations Oscillation modes Inertial loads Distribution of the weights of masses
F05	Frontal method
F06	Frontal method. Auxiliary information.
F07	Values of displacements
F08	Values of forces
F10	Design combinations Unified design combinations
F11	Stiffness matrix for Lanczos
F12	Dynamics for Lanczos
F14	Accelerations for dynamic loadings
F15	Accelerations of points
F16	Stability factors of safety Buckling modes Unsupported lengths of elements
F17	Sorted displacements
F19	Principal and equivalent stresses
F20	Principal and equivalent stresses from combinations of loadings
F21	Design combinations

Extensions	Content
	Unified design combinations
F22	New DCF for printing
F23	Old DCF for printing
F24	Right-hand sides
F25	Sorted displacements
F28	DCF for progressive collapse
F29	Design combinations of displacements
F30	Design combinations of reactions in constraints
F31	Results of the nonlinear processor necessary for postprocessing
F32, F33, F36	Working files of the nonlinear processor
F34	Working file of the calculated displacements for the nonlinear processor
F35	Working file of the data of the previous problem
F39	Results of the calculation in the “Fire Resistance” postprocessor
F40	Results of the calculation in the “Steel” postprocessor
F41	Erection data
F42	Erection dynamics
F43	Erection deformations
F44	Erection forces
F45	
F50	Polynomials (working files)
F51	Energy
F53	Loads from the structural fragment in constraints from combinations of loadings
F54	Reactions in constraints
F55	From combinations of loadings: Stability factors of safety Buckling modes Unsupported lengths of elements
F56	Loads from the structural fragment from a combination of loadings
F57	Loads from the structural fragment
F58	Displacements from combinations of loadings
F59	Forces from combinations of loadings
F60	Design combinations of deflections
F71	Response spectra
F72	AFC
F73	
F81-F84	Results of integration of the equations of motion
F91	Check "Steel"
F92	Check "R/c"
F93	Check of the progressive collapse "Steel"
F94	Check of the progressive collapse "R/c"
F95	Results of the selection of steel
F96	Results of the reinforcement selection

Text files of the results **Table 3**

Extension name	Content
P01	Analysis log
P02	Initial data
P03	Errors in the initial data
P04	Service information
P05	Values of displacements
P06	Values of forces
P07	Values of reactions

Extension name	Content
P08	Design combinations and unified design combinations
P09	Frequencies of natural oscillations
P10	Oscillation modes
P11	Inertial loads
P12	Superelement structure
P13	Information on the state of the material
P14	Principal and equivalent stresses for forces
P15	Principal and equivalent stresses from combinations of loadings
P16	Explanatory note
P17	Distribution of the weights of masses
P18	Stability factors of safety
P19	Unsupported lengths of elements
P20	Buckling modes
P21	Soil stiffness parameters
P22	From combinations of loadings: values of forces
P23	From combinations of loadings: values of displacements
P24	From combinations of loadings: principal and equivalent stresses
P25	Reactions on the foundation
P26	From combinations of loadings: reactions on the foundation
P27	From combinations of loadings: stability factors of safety
P28	From combinations of loadings: unsupported lengths of elements
P29	From combinations of loadings: buckling modes
P30	Values of pulses
P31	Accelerations
P32	DCF
P33	Reactions in constraints
P34	Reactions in constraints from combinations of loadings
P35	Design combinations of displacements
P36	Design combinations of displacements in constraints
P37	DCF for the progressive collapse
P38	Design combinations of deflections
P39	Accelerations
P71,P72,P73	Response spectra
P97-399	Reinforcement

"Hot" Keys and Context Menu Settings

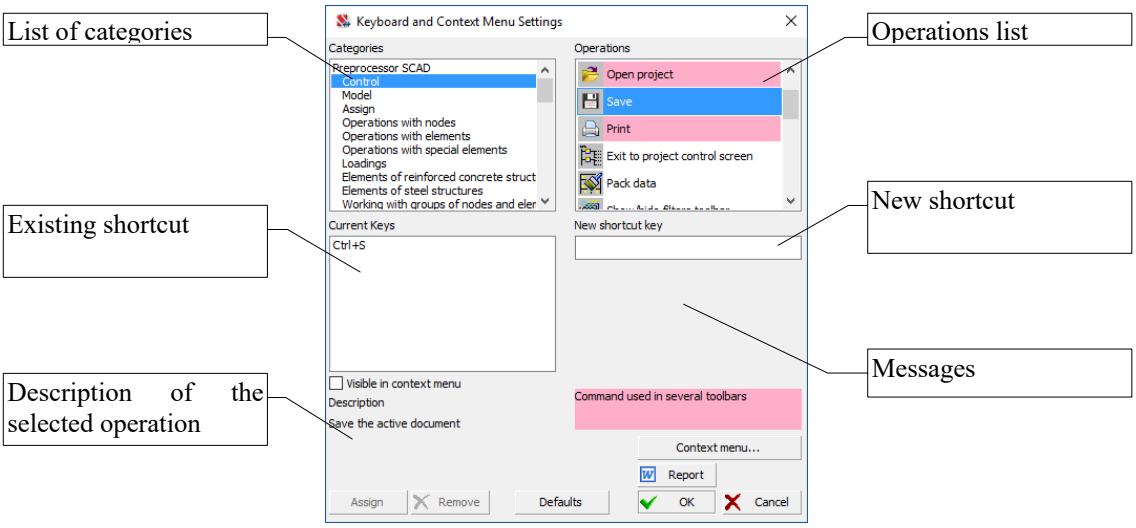
The list of “hot” keys of SCAD++ is given in the table below.

Perform the operation		Enter
Cancel		Escape
Save project		Ctrl+S
Save project as…		Ctrl+Shift+S
Open project		Ctrl+O
Undo the last operation		Ctrl+Z
Redo the last undone operation		Ctrl+Y or Ctrl+Shift+Z
Print		Ctrl+P
Delete selected objects		Delete
Close the active project		Ctrl+W

Exit to the project tree		Ctrl+T
Information on the element		Ctrl+Shift+F
Information on the node		Ctrl+F
Help		F1

You can also create and modify your own sets of hot keys for the vast majority of operations implemented in the SCAD pre- and postprocessor. Hot keys can be repeatedly reassigned, including those that have been defined during the installation. Generally, rules for assigning the hot keys used in any Windows-application apply to SCAD as well. For example, one or more modifier keys — Ctrl, Alt or Shift, can be used as a prefix (first keys in a combination). Moreover, one operation can be invoked by several sets of hot keys.

The **Keyboard Settings** dialog box, which is invoked from the respective item of the **Settings** menu, enables to create and modify the sets of hot keys.



Dialog Box **Keyboard and Context Menu Settings**

It is necessary to perform the following operations to create a new combination of hot keys:

- invoke the **Keyboard Settings** dialog box;
- select the name of the toolbar, filter toolbar, or menu which includes the operation in the **Sections** list;
- select the necessary operation (if the hot keys are already assigned for this operation, they will be displayed in the **Assigned key combinations** field);
- place the cursor in the **New key combination** field;
- press the keys of the combination;
- click the **Assign** button.

To replace an existing combination enter a new set of keys according to the above rules, and then delete the previous one. To delete a combination select it in the **Assigned key combinations** field and click the **Delete** button in the dialog box.

If the new key combination has already been used for invoking another operation, the respective message will appear in the message field of the dialog box.

The **Default** button puts the set of hot keys into the default state described above. The **Report** button creates a document with a list of all assigned hot keys.

Some commands are repeated in several toolbars (for example, the *Display the design model* command). These commands are shown in a list on a pink background. The setting of hot keys for such commands automatically leads to their setting in all toolbars.

*The above-mentioned **Keyboard and Context Menu Settings** dialog box, which can be invoked from the **Settings** menu, enables to create and edit the context menu.*

It is necessary to perform the following actions to add the button invoking the operation to the menu*:*

- *invoke the **Keyboard and Context Menu Settings** dialog box;*
- *select the name of the toolbar, filter panel or menu the operation belongs to in the **Sections** list;*
- *select the line with the name of the necessary operation in the **Operations** list;*
- *check the **Add to the context menu** checkbox;*
- *repeat the above actions if it is necessary to add other buttons to the menu.*

*The menu can be edited (buttons can be removed or their order can be changed) in the respective dialog box, which is invoked by clicking the **Context menu** button… The order of operations can be changed by clicking the **Up/Down** buttons, and the **Delete** button enables to remove the operation from the menu.*

The set of operations available in the context menu depends on the mode of the software. For example, the preprocessing operations will be automatically excluded from the menu when working in the postprocessor.

The information on hot keys (and settings of dialog boxes **Information on the Node, Information on the Element**) is saved in the file "%systemdrive%\users\All Users\SCAD Soft\SCADACCEL.tbl". If it is necessary to transfer the settings to another computer, you just have to copy this file (**SCAD++** has to be closed).

TABLE OF CONTENTS

1. SCAD OFFICE SYSTEM 4
1.1.Structure CAD 5
Project 5
Functional Modules 5
Processor and Finite Element Library 6
Graphical Tools for Generating a Design Model 8
Enlarged Design Models and Connection with Other Systems 8
Groups 9
Filters 9
Graphical Postprocessor 10
Documenting the Results 10
1.2.Design and Analysis Programs 10
Kristall 10
ComeIn 11
ARBAT 12
Decor 13
ZAPROS 13
1.3.Design and Engineering Programs 14
Monolit 14
COMET 15
1.4.Auxiliary Programs 15
Section Builder 15
Consul 15
TONUS 15
SEZAM 16
West 16
Slope 16
Cross 16
1.5.Service Functions 17

2. THEORETICAL FOUNDATIONS 18
2.1.Structure and Its Design Model 18
General Information 18
Design Model According to the Slope Deflection Method 19
Principal and Auxiliary Unknowns 21
External and Internal Constraints 21
Junction between Elements and Nodes 22
Groups, Substructures, Super-Elements 22
Loads and Actions 23
2.2.Basic Relationships of the Finite Element Method 23
Linear Static Analysis 23
Additional Constraints 25
Rigid Bodies 25
Dynamical Analysis 28
Accounting for Damping 30
2.3.Solving Systems of Equations 31
2.4.Standard Dynamic Load Cases 32
Wind Load 33
Seismic Actions 34
Pulse Loads 35
Harmonic Excitation 36
Analysis with Accelerograms 36
Direct Integration of the Equations of Motion 36
2.5.Stability of Equilibrium 38
Governing Equations 38
Single-Parametric Loading 39
Analysis of a Role Played by Particular Subsystems 40
2.6.Design Combinations of Forces (DCF) 41

Bars ... 41
Membranes (Plane Stress State) ... 42
Plates ... 42
Shells ... 42
Solid Elements ... 42
Loadings ... 43
2.7.Erection ... 43
Elementary Actions ... 43
Agreement on the Loads ... 44
Variants of the Model Modification ... 44
Finite Element Procedures ... 45
2.8.Finite Element Models ... 46
Convergence of FEM ... 46
Convergence Validation for Some Models ... 48
Circumventing Singularities ... 50
Fragmentation ... 51
Counteracting the Dimensional Instability ... 52
Coupling Elements of Different Dimensionality ... 53
Effects of Merged Displacements ... 55
Allowing for a Distributive Ability of Soil ... 56
Using Rigid Inserts ... 57
References to Chapter 2 ... 60

3. FINITE ELEMENT LIBRARY ... 61
3.1.General Issues ... 61
3.2.Composition of the Finite Element Library ... 62
3.3.Bar Elements ... 65
Local Coordinate Systems of the Bar ... 65
General Description of Bar Finite Elements ... 68
Types of Bar Finite Elements ... 69
Peculiarities of Specifying the Initial Information for Bar Finite Elements ... 70
Loads on Bar Finite Elements ... 75
3.4.Flat Finite Elements for the Calculation of Deep Beams, Thin Plates, and Shallow Shells ... 77
Class of the Solved Problems ... 77
Material of Structures ... 78
Reissner-Mindlin Plates ... 79
Allowing for the Elastic Subgrade ... 80
Local Coordinate Systems ... 80
Types of Flat Finite Elements ... 80
Peculiarities of Specifying the Initial Information for Flat Finite Elements ... 86
Loads on Flat Finite Elements ... 87
3.5.Finite Elements for Solving the Spatial Problem of the Theory of Elasticity (Solid Elements) ... 92
General Information ... 92
Local Coordinate Systems ... 93
Set of Finite Elements ... 93
Peculiarities of Specifying the Initial Information for Solid Finite Elements ... 93
Loads on Solid Finite Elements ... 94
3.6.Universal Finite Elements for Solving the Axisymmetric Problem of the Theory of Elasticity ... 96
General Information ... 96
Set of Finite Elements ... 98
Peculiarities of Specifying the Initial Information for Axisymmetric Finite Elements ... 99
Local Loads ... 99
3.7.Finite Elements for the Calculation of the Piecewise Inhomogeneous throughout the Thickness (Laminated) Plates and Shallow Shells ... 101
Specifying Finite Elements ... 101
Initial Hypotheses of the Model Accounting for the Transverse Shear, Compression of Layers and Curvature 102
Initial Hypotheses of the Model Accounting for the Interlayer Shear and Curvature ... 104
Multilayer Elements in the FE Library ... 105
Peculiarities of Specifying the Initial Information for Multilayer Finite Elements ... 106

Local Loads 106
3.8.Special Finite Elements 106
General Type "Null Element" (FE Type — 154) 108
FE Modeling the Constraints of Finite Rigidity (FE Type — 51) 109
Two-Node Peripheral FE of Elastic Subgrade (FE Type — 53) 109
Single-Node Peripheral FE of Elastic Subgrade (FE Type — 54) 109
FE Modeling an Elastic Constraint between the Nodes (FE Type — 55) 110
FE Modeling the Behavior of an External Node Damper (FE Type — 56) 110
FE Modeling the Behavior of a Node-to-Node Damper (FE Type — 57) 111
Rigid Body (FE Type – 100) 112
Pseudo-Element Designed for Modeling Elements Excluded from the Design Model (FE Type — 200) 112
References to Chapter 3 112

4. STRUCTURE CAD FOR DUMMIES 116
4.1.Graphical Environment 116
Menus 117
Toolbars 119
Icons 119
Buttons with Additional Features 120
Graphs 120
Filters 120
Windows 122
Mouse 123
Cursors 123
Dialog Boxes 124
Working with Tables 125
Tabular Representation of the Data and Results 126
List of Messages 129
4.2.Undo and Redo Operations 129
4.3.Color Scales 130
Control of the Interval Color Scales 130
4.4.Solution of the Simplest Problem 132
Loading the Software 133
Creating a New Project 134
"Control" Section of the Preprocessor Toolbar 135
Creating the Design Model, Calculation and Analysis of Results 136
4.5.Calculation 142
4.6.Graphical Analysis of the Calculation Results 145
Analysis of Displacements 145
Analysis of Forces 146
4.7.Creating a Design Model from 2D Elements 147
Specification of the Boundary Conditions 149
Specification of Loads 149
Graphical Analysis of the Calculation Results 151
4.8.THUS … 153

5. CREATING A DESIGN MODEL 154
5.1.Design Models of Frame Bar Structures 155
Generate a Frame Prototype Operation 155
Multi-Storey Multi-Span Frames 155
Single-Storey Multi-Span Frames of Industrial Buildings 157
Single-span Variable I-section Frames 159
5.2.Creating Models of Plane Hinged-Bar Systems 161
Generating a Steel Truss Prototype 161
Work in the Section Tab (Additional Features) 162
Generating a Timber Truss Prototype 163
Generating Trusses with Elements with a Custom Section 164
5.3.Spatial Frames and Trusses 164
5.4.Creating a Design Model of a Grade Beam 165
5.5.Generating a Rectangular Mesh of Finite Elements on the Plane 166

5.6.Surfaces of Revolution ... 168
Description of Parametric Prototypes of Surfaces of Revolution ... 170
5.7.Creating an Analytically Defined Surface of Revolution ... 174
Rules for Entering Mathematical Formulas ... 175
Example. Radio Telescope Reflector ... 176
Example. Helical Ramp ... 176
5.8.Creating an Analytically Defined Surface ... 176
Example. Shell (Paraboloid of Revolution) ... 177
5.9.Creating Parametric Surfaces ... 178
Examples of Creating Design Models Using the Operation of Generating Parametric Surfaces ... 179
5.10.Creating a Standard Analytic Surface ... 181
5.11.Creating Standard Structures ... 184
5.12.Generating an Arbitrary Mesh of Finite Elements on the Plane ... 187
Fundamental Principles ... 187
Specifying a Meshing Contour ... 188
Transforming Bars into a Meshing Contour ... 190
Converting the Convex Hull of the Selected Nodes into the Meshing Contour ... 190
Converting Enlarged Elements (Objects) into a Meshing Contour ... 190
Parameters of Meshing ... 191
Some Nuances ... 193
Mesh Refinement ... 193
Mesh Quality Assessment ... 197
Improving the Meshing Quality ... 198
Export the Meshing Contour to GMSH ... 198
5.13.Assembling a Model from Several Models ... 198
Methods of Assembling ... 200
Rules of Assembling ... 201
Working with Loads ... 201
5.14.Copying a Design Model ... 202
5.15.Copying the Fragment of the Model ... 205
5.16.Creating Design Models from Solid Elements ... 205
5.17.Geometric Transformations ... 206
Moving ... 206
Rotation about the Given Axis ... 207
Scaling along the Given Line ... 207
Scaling in the Given Plane ... 207
Mirroring with Respect to the Given Axis ... 207
Scaling (Full) ... 208
Examples of Geometric Transformations ... 208
5.18.Specification of Grid Lines ... 211
5.19.Creating a Model on Grid Lines ... 212
5.20.Copying the Model along Trajectories ... 213
Trajectories ... 213
Copying the Model Using the Trajectory as a “Director” ... 214
5.21.Graphics Editor ... 215

6. OPERATIONS WITH NODES AND ELEMENTS ... 216
6.1.Selection of Nodes and Elements ... 216
6.2.Operations with Nodes ... 218
Create Nodes ... 219
Delete Nodes ... 220
Restore Deleted Nodes ... 220
Create Additional Nodes between Nodes ... 221
Move Nodes ... 222
Merge Nodes with the Same Coordinates ... 223
Generate Nodes along an Arc ... 224
Move the Origin ... 224
Create Nodes at the Specified Distance from the Selected Ones ... 224
Replace a Node in Elements ... 225

Select Nodes....225
Create Nodes at the Intersection of Grid Lines....226
Generate a Generator by the Formula....226
Move Nodes to the Given Plane....227
Round the Node Coordinates....228
Renumbering Nodes....229
Copy Node Properties....229
6.3.Operations with Elements....230
Create Plates....232
Create Bars....233
Create Solid Elements....235
Delete Elements....235
Restore Deleted Elements....236
Create Bars Taking into Account the Intermediate Nodes....236
Divide the Bar....236
Divide Bars at the Intersection Point....237
Combine Bar Elements....237
Create Bars along an Arc....238
Select Elements....239
Merge Duplicate Elements....239
Divide 4-node Plates....239
Divide 4-node Plates into 3-node Ones....240
Divide 3-node Plates....240
Divide Solid Elements....241
Divide Plates Taking into Account the Intermediate Nodes....242
Separate Elements....242
Attach Additional Nodes to the Elements....242
Divide Bars Taking into Account the Intermediate Nodes....243
Move Elements....243
Detach All Additional Nodes from the Faces of the Elements....244
Combine Elements....244
Divide Plates along the Intersection Line....245
Extend the Bar to the Plate....245
Chamfer Bars....245
Create Bars Perpendicular to the Selected One....246
Meshing Quality Assessment....246
Improve the Meshing Quality....246
Export Polygon from the Selected Plates....247
Renumbering Elements....247
Copy Element Properties....247
Divide Bars by a Polygon....248
6.4.Input and Assign Parameters of Special Finite Elements....249
Create a 0-element....249
Create Constraints of Finite Rigidity....250
Create an Elastic Constraint....251
Create Two-node Peripheral Elements of the Plate....252
Create Single-node Peripheral Elements of the Plate....253
Create Cable-stayed Elements....254
Create Unilateral Constraints....254
Create External Node Dampers....255
6.5.Input and Assign Parameters of Rigid Bodies....256
Create a Rigid Body....257
Change the Configuration of a Rigid Body....258
Move the Master Node....258
Delete Rigid Body....258
Add Slave Nodes....258
Remove Slave Nodes....258
Move Slave Nodes....258
6.6.Groups of Nodes and Elements....258

Create Groups ... 259
Correct the Group ... 260
Select Group ... 260
Operations with Groups ... 260
Control the Groups ... 261

7. SPECIFYING CHARACTERISTICS OF NODES AND ELEMENTS ... 263
7.1.Specifying Stiffness Properties ... 264
Specifying Stiffness Properties for Bar Elements ... 264
Numerical and Parametric Description ... 269
Specifying Stiffness Properties for Plate Elements ... 271
Specifying Stiffness Properties for Solid Elements ... 273
Specifying Stiffness Properties for Multilayer Elements ... 274
Specifying Stiffness Properties for Axisymmetric Elements ... 275
7.2.Specifying Stiffness Properties for Special Finite Elements ... 276
Color Indication of the Rigidities ... 276
Delete Duplicate Types of Rigidity ... 278
Renumber Rigidities ... 278
Physical and Mechanical Properties of the Material ... 279
7.3.Specifying the Properties of Elements ... 279
Specifying the Element Type ... 279
Specifying Nonlinear Elements ... 280
Install or Remove Rigid Inserts ... 280
Move the Plate Midplane ... 281
Install and Delete Hinges and Sliders ... 282
Specifying the Orientation Angle of the Axes of the Bar Section ... 283
Specifying Intermediate Sections for the Calculation of Forces ... 284
Change the Direction of the Local X_1 Axis of Bars to the Opposite One ... 285
Change the Direction of the Local Z_1 Axis of Plate Elements ... 285
Stresses along the Given Direction ... 285
Specifying the Direction of the Orthotropy Axes ... 288
Specifying Subsoil Parameters ... 288
Specifying the Prestressing of Elements of the Design Model ... 289
Specification of the Additional Data for Elements ... 290
7.4.Specifying Characteristics of Nodes ... 291
Installing Constraints in Nodes ... 291
Merged Displacements ... 292
7.5.Specifying Subsoil Parameters for the Elements of the Design Model with the Help of Cross ... 293
References to Chapter 7 ... 295

8. SPECIFYING LOADINGS ... 296
8.1.Specifying Static Loadings ... 296
Automatic Specification of the Self-weight ... 297
Nodal Loads ... 298
Specifying Loads for a Group of Nodes ... 299
Loads on Bar Elements ... 300
Specifying Loads for a Group of Elements ... 301
Loads on Plates ... 302
Specifying Loads for a Group of Elements ... 304
Specifying Loads on Solid Elements ... 304
Thermal Loads ... 306
Loads on Region ... 308
8.2.Analysis for the Specified Displacements ... 309
Loads through “Null Elements” ... 309
Specifying a Load by the Displacement of a Constraint ... 311
8.3.Saving Loadings ... 312
8.4.Groups of Loads ... 313
Specifying Loads Using Groups of Nodes and Elements ... 314
Generating Loadings from Groups of Loads ... 315
Specifying Factors for Groups of Loads ... 316

8.5.Deleting Loads, Loadings and Groups of Loads 317
Deleting Loads 317
Deleting Loadings and Groups of Loads....318
Pack Loadings 318
8.6.Preparing Data for the Analysis for Dynamic Actions 318
8.7.Specifying the Parameters of Dynamic Actions 322
Creating a New Loading 322
Correcting Parameters of the Existing Loading 325
Peculiarities of Specifying Data on Seismic Actions 325
Seismic Analysis according to SNiP II-7-81*....329
Seismic Analysis according to SNiP II-7-81* with Modifications from 27.12.99....330
Seismic Analysis according to SP 31-114-2004 of the Russian Federation....331
Seismic Analysis according to SP 14.13330.2011 (Updated edition of SNiP II-7-81*) of the Russian Federation 332
Seismic Analysis according to MGSN 4.19-05....333
Seismic Analysis according to SP 14.13330.2014 and SP 14.13330.2018 (Updated edition of SNiP II-7-81*) of the Russian Federation 334
Seismic Analysis according to SP 14.13330.2018 with Chanhe No.1 (Updated Edition of SNiP II-7-81*) of the Russian Federation 335
Calculation according to the Design Codes for Earthquake-Resistant Nuclear Plants NP-031-01 337
Seismic Analysis according to Ukrainian Design Codes DBN B.1.1-12:2006 338
Seismic Analysis according to Ukrainian Design Codes DBN B.1.1-12:2014 339
Seismic Analysis according to SNiP B1.2-4-98 of the Republic of Kazakhstan 341
Seismic Analysis according to SNiP RK 2.03-30-2006 of the Republic of Kazakhstan....341
Seismic Analysis according to SP RK 2.03-30-2017, SP RK 2.03-30-2017*, and NTP RK 08-01.1-2017 of the Republic of Kazakhstan 342
Seismic Analysis according to SNRA II-2.02.94 of the Republic of Armenia 344
Seismic Analysis according to SNRA II-6.02.2006 of the Republic of Armenia 345
Seismic Analysis according to SNRA AzDTN 2.3-1 of the Republic of Azerbaijan 346
Seismic Analysis according to SNT 2.01.08-99 and SNT 2.01.08-99* of Turkmenistan....346
Seismic Analysis according to KMK 2.01.03-96, KMK 2.01.03-19 of the Republic of Uzbekistan....347
Seismic Analysis according to SNiP KR 20-02:2009 of the Kyrgyz Republic....348
Seismic Analysis according to GNiP RT 22-07-2015, GNiP RT 22-07-2018 Tajikistan Republic....349
Seismic Analysis according to SP 358.1325800.2017 350
Seismic Analysis according to EN 1998-1....351
Seismic Analysis Based on an Accelerogram 353
Seismic Analysis Based on the Specified Accelerograms (3 Components)....354
Seismic Analysis Based on the on the Six-Component Action....354
Wind Pulsation....356
Wind Pulsations according to SNiP 2.01.07-85* 357
Wind Pulsations according to Moscow City Building Codes (MGSN) 358
Wind Pulsations according to SP 20.13330 (Updated Edition of SNiP 2.01.07-85*)....358
Pulse or Impact Action....359
Modal Analysis 360
Harmonic Oscillations....361
Direct Integration of the Equations of Motion 362
References:....365
Specifying Dynamic Loads and Masses....365
8.8.Correcting the Parameters of Loadings/Groups of Loads....366
References to Chapter 8 367

9. FORUM PREPROCESSOR 369
9.1.General Principles for Creating a Model 369
Toolbar....370
Loads....374
Groups....374
9.2.Structure of the Model....374
Nodes 374
Modules....374
Elements....375

9.3.Creating a New Model ... 376
New Project ... 376
Creating a New Module ... 377
Create Elements ... 377
Walls ... 378
Copying Models and Modules ... 379
Copying the Fragment of the Model ... 380
Transferring Elements from One Module to the Other ... 381
Control the Blocks ... 381
9.4.Creating Standard Structures ... 381
9.5.Operations with Nodes ... 383
9.6.Operations with Elements ... 383
Change the Stiffness Properties ... 384
Connect Walls and Floors ... 384
Correcting Openings ... 385
Delete Openings ... 385
Extend the Bar to the Plate ... 386
Expansion of the Plate by an Offset from the Faces ... 386
Divide Plates along the Intersection Line ... 387
Operations with a Pair of Planar Objects ... 388
Create Surfaces of Revolution ... 388
Division of floors and walls by beams and columns lying in their plane ... 388
9.7.Project Tree ... 389
Control Panel ... 390
View and Adjust the Parameters of Elements ... 390
Adjusting the Geometry of Elements ... 391
9.8.Display Filters ... 397
9.9.Specifying Loads ... 397
Parameters of the Load Specification Mode ... 398
Tables of Loads ... 398
Permanent Loads ... 398
Temporary Loads ... 399
Allowing for Permanent Loads ... 400
Peculiarities of Allowing for the Self-Weight of Enclosing Structures and Partitions ... 400
Specify the Types of Rooms ... 400
Delete Loads ... 401
Control of the Permanent Loads ... 401
9.10.Export the Model to SCAD ... 401
Generation of a Finite Element Model ... 402
Modeling the Joint of the Column and Floor Slab Intersection ... 403
Accounting for Loads ... 404
Results of Generation Dialog Box ... 405
9.11.Connection with Architectural and Graphic Systems ... 406
9.12.Data Transfer from the FORUM Preprocessor to Monolit ... 407
9.13.Example of Creating a Model in the FORUM Preprocessor ... 409

10. CONTROL OF THE DISPLAY OF THE DESIGN MODEL ... 413
10.1.Visualization Toolbar ... 413
Rotation of the Model ... 413
Set the Rotation Angle ... 414
Continuous Rotation of the Model ... 414
Orbit ... 414
Project the Model on the Coordinate Plane ... 414
Isometric Projection ... 415
Selecting a Planar Fragment ... 415
Select a Fragment with a Marquee ... 415
Cut on Projections ... 417
Fragmentation on the Grid Lines ... 418
Undo and Redo Fragmentation of the Design Model ... 418

Zoom in the Image 419
Full Screen Mode 419
Rendering Engine 419
Fragmentation Using the Tree 420
Fragmentation of the Selected Objects 420
Fragmentation of Not Selected Objects 420
Search of Nodes with the Given Characteristics 420
Save the Screen Image 421
Save as HTML 421
Save and Restore the View of the Model on the Screen 421
10.2.Display Information on the Design Model 422
General Comments on Displaying the Information on the Model 422
Saving and Resetting the Filters 423
Setting the Output of the Digital Information 424
Print the Design Model 424
Reset Display Filters 425
Display Information on the Nodes 425
Display Information on the Elements 432
Displaying Loads, Masses and Specified Displacements 442
Display Axes of the Global Coordinate System 446
Display Grid Lines 446
Display Dimension Lines 449
Deleting the Lines of the Invisible Contour 449
Transparency, Lightning, Perspective 450
Color Display of the Groups of Nodes and Elements 450
Color Display of the Specified Reinforcement 451
Display Groups of Structural Members 451
Cancel the Selection of Nodes and Elements 452
Navigator 452
Show Elements Included/Not Included in the Given Erection Stage 452
Prestressing 453
Color Display of Modules 453
Types of Rooms 453
Color Display of Fire Resistance Groups 453
10.3.Displaying the Results of the Calculation 454
Visualization of Results Toolbar 454
Displaying the Deflections in Bars 454
Finite Element Mesh on Isofields 454
Displaying the Deformed Model 454
Output the Values on Isolines, Isofields and Diagrams 455
Selective Output of Values on Isolines, Isofields, and Diagrams 455
Control of the Output of Isofields 456
10.4.Environment Settings 457
10.5.Color Palette Settings 462

11. CONTROL OF THE CALCULATION 464
11.1.General Parameters 464
11.2.Gaussian Method 466
11.3.Multifrontal Method 466
11.4.Iterative Method 467
11.5.PARFES 467
11.6.Stability 467
11.7.Dynamics 468
11.8.Nonlinearity 470
11.9.Elements 470
11.10.Special Settings 471
11.11.RAM Usage 471
11.12.Optimization of the Stiffness Matrix 472
11.13.Multi-threaded Calculations 472
11.14.Description of Solvers 473

Linear and Nonlinear Statics ... 473
Multifrontal Solver ... 474
PARFES ... 475
Iterative Solver ... 476
Gaussian Method ... 476
11.15.Description of the Dynamic Solvers ... 477
Subspace Iterations Method ... 477
Method of Steepest Descent ... 477
Lanczos Method ... 478
References to Chapter 11 ... 479

12. GRAPHICAL ANALYSIS OF THE STRESS STRAIN STATE ... 482
12.1.General Principles of Controlling the Display of Results ... 482
Selecting Loading ... 483
Selecting the Considered Factor ... 484
Scale of Display ... 484
Displaying Markers, Isolines, Isofields, and Isosurfaces ... 484
Output of Isolines and Isofields with a Constant Step of the Considered Factor ... 486
Units of Measurement ... 486
Displaying the Design Model ... 487
12.2.Analysis of Displacements ... 487
Animation of Stepwise and Time Processes ... 489
Determination of the Coordinates of the Center of Mass ... 490
Determination of the Tilt of the Structure ... 491
Determination of the Tilt of the Foundation Slab ... 491
Periods and Frequencies of Oscillations ... 492
Weights of Nodal Masses ... 493
Dynamic loads by oscillation modes ... 493
Accelerations ... 494
Unsupported Lengths of Bar Elements ... 494
Saving the Deformed Model ... 495
Accounting for the Initial Imperfections ... 495
Special Processing of the Output Displacements ... 496
12.3.Analysis of Forces in Bars ... 497
12.4.Analysis of Forces and Stresses in Plate Elements ... 499
Control of the Display of Forces and Stresses ... 499
Selective Marking of Force Factors ... 501
Plotting Stress Diagrams ... 501
Isofields of Stresses in Plates ... 502
Units of Measurement of Force Factors ... 504
12.5.Analysis of the Results of Postprocessing ... 505
12.6.Graphical Display of the Reactions in Constraints ... 505
12.7.Graphical Display of the Forces in Special Finite Elements ... 506
12.8.Generation of the Groups of Elements ... 506

13. DOCUMENTATION OF THE INITIAL DATA AND ANALYSIS RESULTS ... 507
13.1.Generator of Tables in a Text Format ... 507
13.2.Documenter ... 510
13.3.Reading the Results ... 513
Window of the Processor ... 513
Documentation of the Initial Data ... 516
13.4.Displacements of Nodes ... 516
Positive Directions of Displacements ... 517
Tables of Displacements ... 517
Displacements from Combinations of Loadings ... 518
13.5.Tables of Forces and Stresses in Elements ... 519
13.6.Stresses and Forces ... 522
Forces in Bars ... 522
Flat Finite Elements ... 525
Solid Finite Elements ... 531

Multilayer Shell Elements 532
Axisymmetric Finite Elements 533
Special Finite Elements 534
13.7.Reactions in Constraints 536
13.8.Natural Oscillations 536
13.9.Design Combinations of Forces (DCF) 538
13.10.DCF at the Analysis of the Progressive Collapse 540
13.11.Design Combinations of Reactions in Constraints 540
13.12.Design Combinations of Displacements (DCD) 541
13.13.Design Combinations of Deflections 541
13.14.Design Combinations of Punching 542
13.15.Loads from the Structural Fragment 542
13.16.Punching Loads 543
13.17.Stability Analysis 543
13.18.Principal and Equivalent Stresses 544
13.19.Results of the Check of the Elements of Reinforced Concrete and Steel Structures and of the Selection of Reinforcement and Rolled Steel Sections 545

14. ERECTION 546
14.1.Creating a Project 546
14.2.Erection Toolbar 546
Initialization of the New Erection Stage 548
Adjusting Settings of the Erection Stage 549
Deleting the Erection Stage 549
Adding Elements in the Current Erection Stage 549
Deleting Elements in the Current Erection Stage 549
Changing the Elastic Moduli for the Current Erection Stage 550
Creating a List of Loadings 550
Specifying and Modifying Constraints 551
Specifying Hinges 552
Deleting Hinges 552
Specifying and Changing the Subsoil Parameters 552
Specifying Merged Displacements 552
Save Information on the Current Stage as a Separate Problem 552
Control the Erection Stages 552
Creating and Modifying the Main Model 553
14.3.Analysis 554
14.4.Analysis of Results 554
Erection Stages History 555
14.5.Design Combinations of Forces 555
14.6.Calculation of Elements of Steel and Reinforced Concrete Structures 555

15. DIRECT INTEGRATION OF THE EQUATIONS OF MOTION 556
15.1.Problem Formulation 556
15.2.Specifying the Initial Information 560
Specifying the Characteristics of the Dynamic Loading 560
Specifying Nodal Actions 566
15.3.Calculation 567
15.4.Results of the Calculation 567
Results of the Calculation at a Fixed Time 567
Graphs of Displacements, Velocities and Accelerations of Nodes 568
Graphs of Forces (Stresses) in Elements 569
References to Chapter 15 569

16. VARIATION OF MODELS 570
16.1.Accounting for the Uncertainty of the Parameters of the Design Model 570
16.2.Differences between Models 570
16.3.Generate a Package of Models 570
16.4.Design Combinations of Forces 572
16.5.Results of the Calculation and Their Analysis 572
16.6.Using the Variation of Models 573

Stiffness Properties ... 573
Compliance of the Subsoil ... 573
Nodal Connections ... 574

17. COMBINATIONS OF LOADINGS ... 575

18. DESIGN COMBINATIONS ... 577
18.1.Fundamental Principles ... 577
Logical Relationship between Loadings ... 578
18.2.Criteria of DCF ... 579
Bars ... 579
Membranes (Plane Stress State) ... 580
Plates ... 580
Shells ... 581
Special Elements ... 581
Solid and Axisymmetric Elements ... 581
Numbering of the Criteria of DCF ... 581
18.3.Criteria of DCD ... 582
18.4.Criteria of Design Combinations of Deflections ... 582
18.5.Criteria of Design Combinations of Reactions in Constraints ... 582
18.6.Criteria of Design Combinations of Punching ... 582
18.7.Data Preparation ... 583
Peculiarities of Specifying Combinations for EN 1990 ... 587
Peculiarities of Specifying Combinations for Variations of Models, Erection, and Nonlinear Problems ... 587
Allowing for the Pulsation Component of the Wind Load ... 588
Crane Loads ... 588
Unification ... 589
Groups for DCF ... 590
18.8.Design Combinations of Displacements ... 592
18.9.Design Combinations of Reactions in Constraints ... 592
18.10.Design Combinations of Deflections ... 592
18.11.Design Combinations for the Progressive Collapse ... 592
18.12.Analysis of Results (DCF and DCD Postprocessor) ... 592
Create Combinations of Loadings for DCF ... 593
Show Elements the DCF of which Include the Specified Loadings ... 594
Show Combinations which Include the Specified Loading ... 594
Show Elements the DCF of which Include the Specified Combinations ... 595
Color Indication of Design Combinations of Forces ... 595
Color Indication of Design Combinations of Displacements ... 596
Color Indication of Design Combinations of Forces (Progressive Collapse) ... 596
Envelopes of the Diagram of Design Combinations of Forces ... 596
Envelopes of the Diagram of Design Combinations of Forces (Progressive Collapse) ... 596
Design Combinations of Reactions in Constraints ... 596
Information on the Element/Node ... 597

19. REINFORCEMENT OF SECTIONS OF REINFORCED CONCRETE ELEMENTS ... 599
Definition of the Main Concepts ... 599
Reinforcement of Two-dimensional Elements ... 599
Reinforcement of Bar Elements ... 600
19.1.Limitations ... 601
19.2.Selection of Design Codes ... 601
19.3.General Information about the Reinforcement Modules ... 602
Bar 2D (Uniaxial Stress State) ... 602
Bar 3D (Biaxial Stress State) ... 603
Plate. Shell ... 604
Deep Beam ... 604
19.4.Work with the Postprocessor ... 605
19.5.Structural Members and Groups of Structural Members ... 605
19.6.Input the Initial Data ... 606
Creating New Structural Groups ... 606

The Reinforcement of Bars and Reinforcement of Plates Dialog Boxes 607
Selection with Limitation 608
Accounting for the Maximum Allowable Deflections and Displacements 608
Importance Factors 609
Advanced Settings 610
Additional Information on the Initial Data 612
Correcting the Initial Data of the Structural Groups 616
Dividing a Group of Reinforcement into Two Groups or More 616
19.7.Main and Additional Groups 617
19.8.Selection of Elements of Groups 618
19.9.Active Group Visualization Mode 618
19.10.Control of the Groups of Reinforced Concrete Structural Members 618
Specify the Classes of Concrete and Reinforcement 620
Information on the Groups of Reinforcement 621
19.11.Setting the List of Reinforcement Spacing 621
19.12.Selection of Reinforcement 622
Output the Results of the Selection 623
Transformation of the Results of the Reinforcement Selection 624
Graphical Representation of Results of the Selection of Reinforcement in Bar Elements 626
Graphical Representation of Results of the Selection of Reinforcement in Plate Elements 627
Export of the Results of Selection of Reinforcement in a Plate to the Design Subsystem of ALLPLAN 629
Generating a Report 629
19.13.Reading the Results of the Reinforcement Selection 630
Bar Elements 630
Plate Elements 632
19.14.Check of the Specified Reinforcement 632
Specifying the Reinforcement Layout of Plate Elements 633
Specifying the Reinforcement Layout of Bar Elements 635
Graphical Representation of the Specified Reinforcement in Bar Elements 636
Display of the Color Map of the Specified Reinforcement in Plate Elements 637
Display Results of the Check 637
Extreme Factor Values 637
Generating a Report 638
Information on the Element – Analysis of Reinforced Concrete Structures 639
Specified Reinforcement Filter 640
19.15.Weight of the Specified Reinforcement 641
19.16.Basic Principles of the Selection of Reinforcement in the Finite Elements of a Slab and Shell 641
Selection of the Type of Reinforcement 641
Strength Check 644
Analysis for Lateral Forces 645
Determination of the Crack Opening Width 646
General Algorithm 647
19.17.Implementation of SNiP 52-101-2003 and SP 63.13330 647
19.18.Peculiarities of the Analysis according to EN 1992-1-1 648
19.19.Peculiarities of the Analysis according to DBN V.2.6-98:2009 650
19.20.Factors in Stress-Strain Models 650
19.21.Analysis of Round Sections 652
19.22.Examples of Using the Results of the Reinforcement Selection in Designing 652
19.23.Punching Analysis 653
Data Preparation, Calculation and Analysis of Results 654
Specify Punching Nodes 655
Select Punching Nodes 656
Show Punching Nodes 656
Calculation 656
Show Punching Contours 656
Display Results 657
Table of Results 658
Generating a Report 658
References to Chapter 19 659

20. CHECK OF THE LOAD-BEARING CAPACITY OF STEEL SECTIONS 660
Input the Initial Data..... 660
20.1.Setting the Parameters..... 661
Steel..... 662
Service Factors 662
Limit Slenderness..... 662
Creating New Structural Groups 663
20.2.Specifying New Structural Members..... 664
Effective Length Factors 664
Critical Moment 664
Accounting for the Maximum Allowable Deflections and Displacements 666
Importance Factors..... 667
Replacing a Section of a Structural Member..... 667
Type of the Structural Member 668
Corrosion analysis 669
Advanced Settings..... 670
Copying the Parameters of Structural Members..... 670
Control of Structural Members..... 670
20.3.Specifying Groups of Structural Members 671
Accounting for the Maximum Allowable Displacements 671
20.4.Correcting the Parameters of Structural Members and Groups of Structural Members..... 672
Copying the Parameters of Groups of Structural Members..... 673
20.5.Main and Additional Groups 673
20.6.Analysis of cold-formed profiles 674
20.7.Specifying Unification Groups 678
20.8.Selection of Elements of Groups 679
20.9.Active Group Visualization Mode 679
20.10.Control of the Groups of Steel Structural Members and Groups of Unifications 679
20.11.Changing Steels in the Groups of Structural Members..... 681
20.12.Information on the Groups of Steel Structural Members..... 682
20.13.Visualization of the Modified Sections..... 683
20.14.Restore Initial Sections 683
20.15.Changing the Design Codes..... 683
20.16.Check and Selection of Sections..... 684
20.17.On Seismic and Special Actions 694
20.18.Analysis of Custom Sections 695
20.19.Display Results 695
20.20.Generating a Report 697
20.21.Extreme values of factors..... 698
20.22.Create Elements of the Technical Specification for Steel..... 699
20.23.Steel Consumption..... 700
20.24.Selection of Sections..... 701
20.25.Information on the Results of the Selection..... 703
Information on the Work of the Postprocessor..... 703

21. FIRE RESISTANCE ANALYSIS OF STRUCTURAL ELEMENTS..... 704
21.1.Fire Resistance of Steel Structural Elements 704
21.2.Fire Resistance of Reinforced Concrete Structural Elements 705
Thermal Analysis 705
Strength Analysis 706
21.3.Results of the Fire Resistance Analysis 707
21.4.Preparation of the Initial Data, Calculation and Analysis of Results..... 707
The Fire Resistance Toolbar..... 707
Specifying the Parameters of Fire Resistance Analysis..... 708
Creating a New Group of Bar Elements..... 708
Modifying the Existing Fire Resistance Groups..... 709
Deleting the Fire Resistance Groups 709
Creating and Modifying Fire Resistance Groups of Plate Elements 709
Control of the Fire Resistance Groups 710

Selecting the Design Codes....710
21.5.Fire Resistance Analysis of Elements....710

22. CALCULATION OF LOADS FROM THE STRUCTURAL FRAGMENT....712
22.1.Input Initial Data....713
22.2.Description of Fragments....713
22.3.Graphical Analysis of the Results of the Calculation of Loads from the Structural Fragment....714

23. PROGRESSIVE COLLAPSE ANALYSIS....716
23.1.Data Preparation....717
The Progressive Collapse Dialog Box....717
Peculiarities of Using the Results of the Reinforcement Selection....718
23.2.Calculation....719
23.3.Analysis of Results....719
References to Chapter 23....720

24. STABILITY ANALYSIS....721
24.1.Problem Formulation....721
24.2.Search of the Stability Factor of Safety....722
24.3.Buckling Modes....723
24.4.Unsupported Lengths....723
24.5.Unsupported Lengths and Effective Length Factors....724
24.6.Design Codes and Stability Analysis of Bars....724
24.7.Input Data....725
24.8.Peculiarities of the Analysis for the Dynamic Loadings....726
24.9.Analysis of Results....726
References to Chapter 24....727

25. ENERGY POSTPROCESSOR....728

26. PRINCIPAL AND EQUIVALENT STRESSES....730
26.1.Principal Stresses for Different Types of Finite Elements....731
Spatial Problem of the Theory of Elasticity....731
Deep Beam Elements....731
Plates and Shells....732
Bar Elements....732
26.2.Calculating Equivalent Stresses....733
26.3.Preparing Data for the Calculation of the Principal and Equivalent Stresses....734
26.4.Graphical Display of the Results of the Calculation of the Principal and Equivalent Stresses....735
Color Display of Principal/Equivalent Stresses in Bars....736
Display of Directions of Principal Planes....737

27. NONLINEAR CALCULATIONS IN SCAD....738
27.1.General Information....738
27.2.Geometric Nonlinearity....739
Cable-Stayed Elements....740
Unilateral Constraints....743
27.3.Physical Nonlinearity....744
General Information....744
Physically Nonlinear Finite Elements....745
Material Models....745
Accounting for Geometric Nonlinearity....754
27.4.Control of the Nonlinear Analysis....755
27.5.Results of the Calculation....757
Nonlinear Loading History....758
References to Chapter 27....758

28. AMPLITUDE-FREQUENCY CHARACTERISTICS....759
28.1.Preparation of the Initial Data....760
28.2.Results of the Calculation....760

29. RESPONSE SPECTRA....762
29.1.Seismic Analysis....762

29.2.Accelerograms and Response Spectra ... 763
29.3.Input Data and Analysis of the Results ... 764
29.4.Preparation of the Accelerogram Files ... 765
References to Chapter 29 ... 765

30. INFLUENCE LINES AND SURFACES ... 766
30.1.Creating Routes ... 766
Creating a Route and Its Snap to the Existing Model ... 766
Peculiarities of the Implementation ... 767
30.2.Preparing Data for the Calculation of Influence Lines and Surfaces ... 767
30.3.Calculation ... 767
30.4.Graphical Display of the Results of the Calculation of Influence Lines and Surfaces ... 768

31. CENTERS OF RIGIDITY ... 770
31.1.Determination of the Center of Rigidity ... 770
31.2.Input and Calculation ... 770
31.3.Analysis of Results ... 771

32. DATA ARCHIVING LANGUAGE ... 772
32.1.Export and Import Text Files with the Initial Data ... 772
32.2.Architecture and Syntax of the Archiving Language ... 773
Reduction of the Volume of the Initial Information When There are Regularities ... 775
32.3.General Information on the Problem. Document 0 “Header” ... 776
Code of the Problem (Line 1) ... 777
Index of the System Type (Line 2) ... 777
Coordinate Systems (Line 3) ... 778
Calculation of Forces in the Intermediate Sections and Nodes (Line 4) ... 778
Control of the Calculation of the Design Combinations of Forces (Line 8) ... 778
Unification of the Sections of Elements (Line 12) ... 779
Organization of the Groups of Design Combinations of Forces (Line 15) ... 779
Allowable Number of Crane and Brake Loadings Included in the Design Combinations of Forces (Line 16) ... 779
Data for Stability (Line 21) ... 779
Data for Calculating the Principal and Equivalent Stresses (Line 22) ... 780
Parameters of the Calculation (Line 23) ... 780
Lists of the Deleted Nodes and Elements (Lines 31 and 32) ... 780
Units of Measurement (Line 33) ... 780
32.4.Documents Describing the Design Model ... 781
Document 1. “Elements” ... 781
Document 2. “Boundary Conditions” ... 781
Document 4. “Coordinates” ... 782
Document 5. “Constraints” ... 782
32.5.Properties of Elements ... 782
Document 3. “Stiffness Properties” ... 782
Document 31. “Rigid Inserts” ... 787
Document 32. “Position of the Principal Axes of Inertia” ... 787
Document 33. “Coordinate Systems for Calculating Stresses and Forces” ... 788
Document 34. “Subsoil Parameters” ... 788
32.6.Specifying Static Loadings ... 789
Document 6. “Types of Loads” ... 789
Document 7. “Values of Loads” ... 790
Documents 9 and 10. “Groups of Loads” ... 792
32.7.Description of Dynamic Loadings ... 792
Description of Masses in Documents 6 and 7 ... 792
Characteristics of Dynamic Loadings ... 792
Specifying Pulse Loads ... 793
Specifying Impact Loads ... 793
Files with Pulse Parameters ... 793
Document 71. “Loading History” ... 794
32.8.Auxiliary Documents ... 794
Document 21. “Grid Lines” ... 794

Document 27. “Meshing Contour”....794
Document 47. “Groups of Elements”....795
Document 48. “Groups of Nodes” 795
32.9.Erection Documents 795
32.10.Documents for Checking and Selecting Rolled Steel Sections....795
32.11.Documents for Reinforcing Sections of R/C Elements and Its Check 795
32.12.Initial Data of Postprocessors....796
Documents of Design Combinations....796
Documents of Design Combinations for Nonlinear Analysis 796
Document 36. “Combinations of Loadings”....796
Document 37. “Stability”....796
Document 38. “Principal and Equivalent Stresses” 797
Document 39. “Loads from the Structural Fragment” 797
Document 53. “Reinforcement of Reinforced Concrete Elements”....797
Document 70. Progressive Collapse....797
32.13.Special Coordinate Systems 798
32.14.Specifying a Load in the Special Coordinate Systems 799

33. SERVICE FUNCTIONS 801
33.1.Save the Fragment of the Model....801
33.2.Determine the area, perimeter, and angles of polygons....801
33.3.Special Functions of the Service Menu 801
Formula Calculator 802
Converting Units of Measurement 803
Properties of Concrete....803
Characteristics of Reinforcement....804
Reinforcement Consumption 805
33.4.Import and Export the Initial Data Prepared in the Form of the Text Description 808
33.5.Comments to the Project....808
33.6.Section Viewer 809
33.7.Graphics Editor....811
Create Graphics Objects....812
Editing Objects....814
33.8.Project Tree Service Section....815
33.9.Accelerogram Editor 816
Main Window 816
Settings....817
Editing an Accelerogram 818
Creating Plots....820
Nonlinear Spectra....820
General Properties....821
33.10.Editor of Dynamic Amplification Factor....823
33.11.CoCon – Reference Manual 824
33.12.KUST – Design-Theoretical Reference Manual....824
References to Chapter 33 825

34. SETTINGS 826
34.1.Units of Measurement 826
34.2.Express Setting of the Units of Measurement 827
34.3.Font Setting 827
34.4.Sections 828
34.5.Report and Languages 828
34.6. UI 829
34.7.EN 1993 Settings....829
34.8.Visualization Settings of the Result Tables 830
34.9.Format Pages for Printing....830
34.10.Setting the Design Codes....831
34.11.Specifying Work Directories 831
34.12.Table Settings 831
34.13.Toolbar Settings....831

35. IMPORT AND EXPORT SCAD DATA ... 833
35.1.Import Design Models ... 833
Import Graphics Files ... 836
Plugins ... 836
IFC ... 838
CIS/2 ... 839
SDNF ... 839
GMSH ... 839
GTC/GTCX/SMLX ... 840
Femap ... 840
STAAD ... 840
ABAQUS ... 841
ANSYS ... 842
SAF ... 843
SLI ... 843
35.2.Export SCAD Data ... 844
IFC ... 844
CIS/2 ... 845
SDNF ... 846
GMSH ... 847
GTC/GNCX/SMLX ... 847
Femap ... 848
STAAD ... 849
ABAQUS ... 850
ANSYS ... 852
SAF ... 852
SLI ... 853

APPENDIX ... 854
List of Assortments of Rolled Profiles ... 854
Extensions of the Files in the Working Directory of SCAD ... 861
"Hot" Keys and Context Menu Settings ... 864