SCAD Soft

SCAD Structure

ARBAT
Analysis of concrete and ferroconcrete structural members

User manual

UDC 737.30

Team of authors:
I.A. Belokopytova, S.V. Girenko, E.Z. Kriksunov, M.A. Mikitarenko,
M.A. Perelmuter, L.N. Skoruk

ARBAT. Analysis of members of concrete and ferroconcrete structures.
USER MANUAL. Version 5.1.

The user manual describes the functionality of the **ARBAT** software application, its controls, and recommendations on its usage.

The application is oriented at structural design engineers with basic computer skills.

Table of contents

1. Introduction 4
   1.1 Structural assessment 4
   1.2 The main window 6
   1.3 Menus 8
   1.4 Settings 8
   1.5 Working with tables 10
   1.6 Saving data 11
2. Information modes 12
   2.1 Concrete class 12
   2.2 Concrete grade 12
   2.3 Reinforcement 12
   2.4 Service factors for concrete 13
   2.5 Deflection limits 13
3. General management 14
   3.1 Building a cross-section 14
   3.2 Concrete data 15
   3.3 Reinforcement data 16
   3.4 Crack resistance 18
   3.5 Safety factor for responsibility 18
   3.6 Generating a report 18
4. Appraisal 20
   4.1 Resistance of ferroconcrete sections 22
   4.2 Resistance of concrete sections 27
   4.3 Appraisal of a beam 27
   4.4 Appraisal of a single-span beam 32
   4.5 Beam deflection 33
   4.6 Deflection of a single-span beam 34
   4.7 Appraisal of a column 34
   4.8 Appraisal of a slab 39
5. Local strength 42
   5.1 Local compression (SNiP 2.03.01-84*) 42
   5.2 Local compression (SP 52-101-03) 44
   5.2 Punching (SNiP 2.03.01-84*) 47
   5.3 Punching (SP 52-101-03) 50
   5.4 Tearing 54
   5.5 Fixings 56
   5.6 Short cantilevers 60
6. Reinforcement proportioning 65
   6.1 Reinforcement proportioning for a beam 65
   6.2 Reinforcement proportioning for a single-span beam 69
   6.3 Reinforcement proportioning for a column 69
7. Geometrical characteristics 74
8. APPENDIX 75
   8.1 On seismic actions 75
   8.2 Design code documents implemented in the ARBAT environment 75
   8.3 Formula calculator 77
   8.4 Converter of measurement units 78
   8.5 Discrete reinforcement 79
9. References 80

1. Introduction

The **ARBAT** software application is intended to perform a load-bearing ability analysis or to proportion reinforcement for members of concrete and ferroconcrete structures, to calculate the deflections in ferroconcrete beams, to verify the local strength of members of ferroconcrete structures (including fixings) in compliance with requirements of one of the following codes of regulations:

- SNiP 2.03.01-84*;
- SNiP 2.03.01-84* that takes into account the modifications made in Ukraine;
- SNiP 52-01-2003 (SP 52-101-03).

The analysis involves limit states of first and second group (strength and crack resistance) for design stress combinations (DSC) selected automatically for particular design loads in compliance with SNiP 2.01.07-85*, “Loads and actions” [2] and SNiP 2.03.01-84*, “Concrete and ferroconcrete constructions” [5] (SNiP 52-01-2003 [8]).

All modes of the application require design values of loads (forces) to be specified. The analysis of second limit state involves rated values of the loads. The analysis of each loading pattern employs user-specified safety factors for load in order to switch from design values to rated ones.

The proportioning of reinforcement and the verification are performed for beams, columns, or slabs made of heavy-weight, fine-grain and light-weight concrete, reinforcement steel of classes A-I (A240), A-II (A300), A-III (A400), A-IV, A-V and A-VI, and reinforcement wire of class Vr-I (V500) and reinforcement of classes A400S, A500S (A500).

Also, the **ARBAT** application is a reference manual that provides data concerning the range and properties of reinforcement, rated and design values of the concrete strength, service factors for concrete, and allowable deflection limits.

The controls and procedures used to prepare data and to document results, which are implemented in the application, are exactly the same as those in other computer-aided design and analysis applications included in the **SCAD Office®** system. The applications are based on common multi-tab windows and dialog boxes. To switch to a mode, click on its tab or use an appropriate menu item.

1.1 Structural assessment

Any set of regulations can be represented as a list of inequalities of the following type:

$$F_j(S,R) \le 1, \qquad (j = 1, \ldots, \mathrm{n}),$$

where F_j is a function of principal variables that implements j-th verification; S are generalized loads (loading actions or effects); R are generalized resistance/strength numbers.

Based on the values of functions F_j, we can introduce the concept of a **limitation utilization factor** (K) and thus represent the analysis criterion as

$$\max_j K_j \le 1,$$

where all required analyses are included. The value of K_j itself will define a reserve of strength, stability, or another design parameter available for a particular element (joint, part, cross-section etc.). If the requirement of the design code is met excessively, then the K_j factor is equal to the fraction of the design parameter which is exhausted (for example, K_j = 0.7 corresponds to the reserve of 30%). If the regulatory design requirements are not met, the value of $K_j > 1$ evidences a violation of some requirement, i.e. it describes an extent of overloading. Thus, K_j is the left-hand part of the design inequality in the form presented above (Fig. 1).

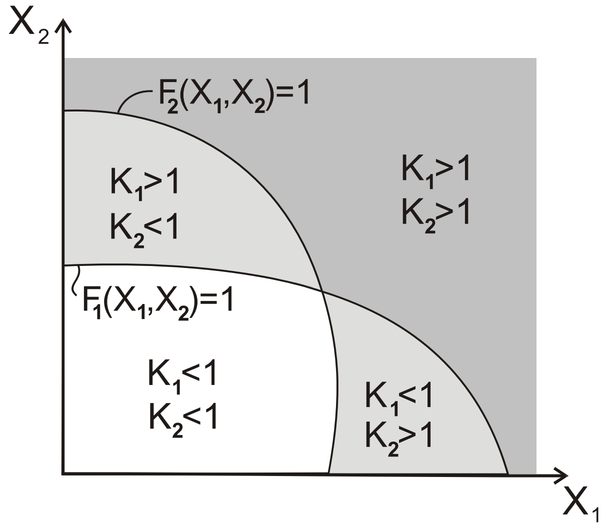


Fig. 1. *A geometric illustration of the validity area for two variables*

All values of the K_j factors obtained by analysis are available for reviewing in the **Criteria Diagram** dialog box (Fig. 2) or in a full report of the respective analysis. Appropriate dialog boxes display the value of $K_{\max}$ — the maximum (hence the most dangerous) value of K_j detected — and indicate an analysis type (such as strength or stability) that has produced this maximum.

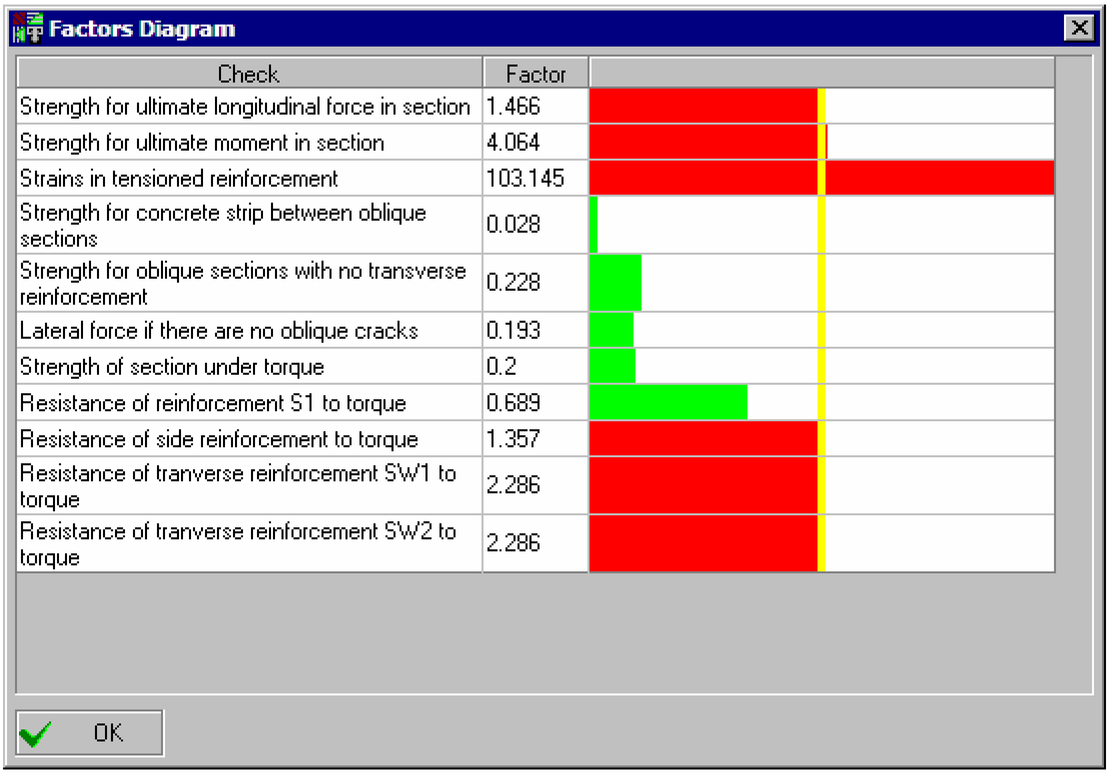


Fig. 2. *An example of the* **Criteria Diagram**

The data presented in the diagram of factors enable the structural engineer to make a proper decision and thus to make modifications needed by the structure in question.

1.2 The main window

When the application is started, the first thing seen on the screen is its main window (Fig. 3) where a working mode is to be selected. Each of the available modes is activated by its appropriate button.

A particular set of design regulations is selected from the appropriate list. The set of regulations that has been selected is displayed in the bottom left corner of the active mode window. This version of the software implements the analysis based on SNiP 2.03.01-84*, SNiP 52-01-2003 (SP 52-101-03), and SNiP 2.03.01-84* that takes into account modifications made for the territory of Ukraine.

All working modes can be classified into four groups:

- reference modes united under the **Information** group header;
- analysis modes which implement verifications of cross-sections and members with given reinforcement and which make up the **Appraisal** group;
- analysis modes which make up the **Local Strength** group and implement the verification (appraisal) of strength of structural members near an area where the load is applied;
- analysis modes which proportion the reinforcement and are gathered together as the **Reinforcement Proportioning** group.

Fig. 3. *The main window of the application*

The reference modes include the following:

- **Concrete class** — helps browse values of the rated and design strength of various classes of concrete for limit states of first and second groups as per SNiP 2.03.01-84* or SP 52-101-03;
- **Concrete grade (SNiP II-21-75)** — shows information similar to that of the previous mode but related to the concrete grades, according to SNiP II-21-75;
- **Reinforcement** — provides data concerning the range, rated and design strength of various classes of reinforcement;
- **Service factors** — provides information from the tables of SNiP 2.03.01-84*;
- **Deflection limits** — provides the deflection limit values from Table 19 of SNiP 2.01.07-85*;
- **Geometric characteristics —** determines the geometric characteristics (area, moments of inertia, parameters of a transformed section etc.) of a given concrete or ferroconcrete section;

The modes for analysis (structural appraisal) include the following:

- **Resistance of f/c sections** — to determine the load-bearing ability of sections of ferroconcrete members which have a given reinforcement;
- **Resistance of concrete sections** — to determine the load-bearing ability of sections of concrete members;
- **Deflection of beam** — to determine the deflections caused by a given load which is applied to a multiple-span beam;
- **Deflection of single-span beam** — to determine the deflections caused by a given load which is applied to a single-span beam;
- **Appraisal of beam** — to appraise the load-bearing ability of a multiple-span beam with a given reinforcement;
- **Appraisal of single-span beam** — to appraise the load-bearing ability of a single-span beam with a given reinforcement;
- **Appraisal of column** — to appraise the load-bearing ability of a column with a given reinforcement;
- **Appraisal of slab** — to appraise the load-bearing ability of a slab supported along its contour and having a given reinforcement;

The **Local Strength** group includes the following modes:

- **Local compression —** appraises the load-bearing ability of structural members against local compression;
- **Thrusting —** appraises the load-bearing ability of slab structures against thrusting;
- **Tearing** — appraises the load-bearing ability of junctions between constructions against tearing;
- **Fixings** — appraises the load-bearing ability of fixings;
- **Short cantilevers** — checks how short cantilevers resists to a lateral force.

The **Reinforcement Proportioning** group includes:

- **Reinforcement proportioning for beam** — designs the reinforcement for a multiple-span beam;

- **Reinforcement proportioning for single-span beam** — designs the reinforcement for a single-span beam;
- **Reinforcement proportioning for column** — designs the reinforcement for posts or columns.

1.3 Menus

The menus are used to customize the application, to invoke a desired working mode, or to perform an action. There are five pull-down menus: **File**, **Modes**, **Settings**, **Tools**, **Help**.

The **File** menu contains the following command items:

Menu — switches from any working mode to the application's main window;

Exit — shuts down the current session of the application.

The **Modes** menu can be used to invoke any of the reference or analysis modes available in the application (its items duplicate the respective buttons of the main window).

The **Settings** menu is used to open the **Application Settings** dialog box where settings and preferences for the application can be customized (this menu duplicates the respective button of the main window).

The **Tools** menu can be used to invoke the standard Windows calculator, a formula calculator, and a measurement units converter.

The **Help** menu advises how to control the **ARBAT** application, explains how to use the Windows help system, and provides information about the application (No. of its version and the date of the last modification).

The use of the reference help and actions performed with the service tools are described in the Appendix.

1.4 Settings

The **Application Settings** dialog box (Fig. 4) is called up via the **Settings** menu or via the respective button located at the bottom of the **Main Window**; it contains three tabs: **Units of Measurement**; **Report and Languages**; **Visualization**.

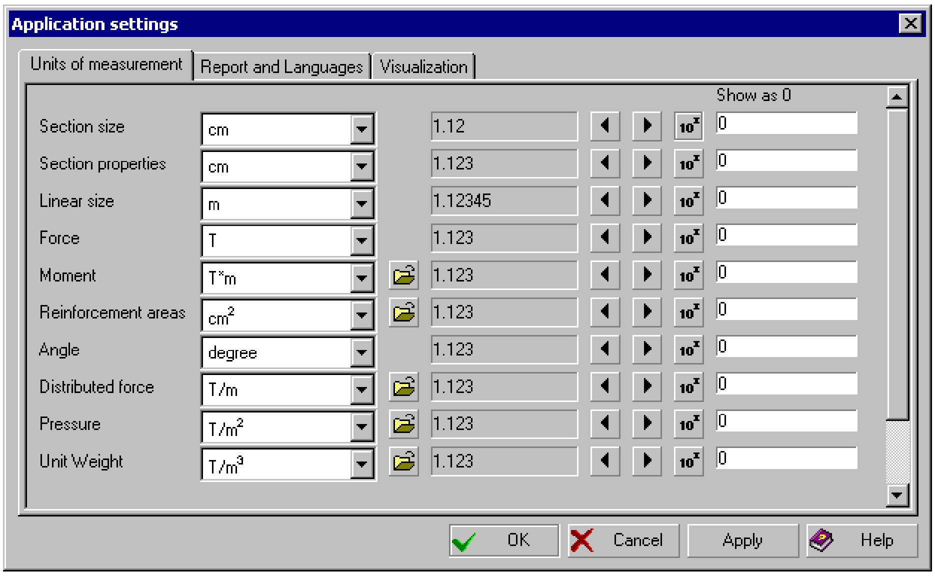
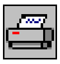


Fig. 4. *The* **Units of Measurement** *tab of the* **Application Settings** *dialog box*

The **Units of Measurement** tab is used to assign units for entering source data and subsequently reviewing the results of the analysis. The units of measurement can be changed at any time when working with the application. To assign simple units such as linear sizes or forces, use appropriate drop-down lists. When the units are compound, the respective lists display the current units, and the assignments are made in the **Set up Units of Measurement** dialog boxes (Fig. 5). The latter dialog boxes are opened by clicking one of the buttons  on the right of the respective drop-down list. To define the compound units, choose the desired simple ones in the lists and exit the dialog by clicking the **OK** button.

The **Report and Languages** dialog box (Fig. 6) is used to choose a language for the user interface, a form of representation for the report, a format for the report document etc. It includes the following controls:

 **View/Edit** — calls up a viewer/browser application for viewing the report, associated with the report's specific format and filename extension;

 **Print** — prints out the report without displaying it on the screen;

 **Report Type** — a drop-down list suggests to choose a file format for the report document. The RTF files come in two versions: Word 7 (Word Pad) or Word 97 and newer; the DOC, HTML, and PDF files are also available. To view or print the PDF files, you need to have the Adobe Acrobat Reader application installed (the application is freeware and can be downloaded at http://www.adobe.com);

the **Paper**, **Margins**, and **Orientation** groups are used to customize the format of the report document;

the **Headers/Footers** group is used to refer to an RTF file that contains headers and footers to be used in the report. This file can be prepared by a user.

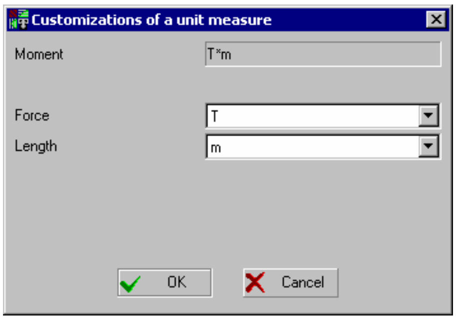


Fig. 5. *The* **Set up Units of Measurement** *dialog box*

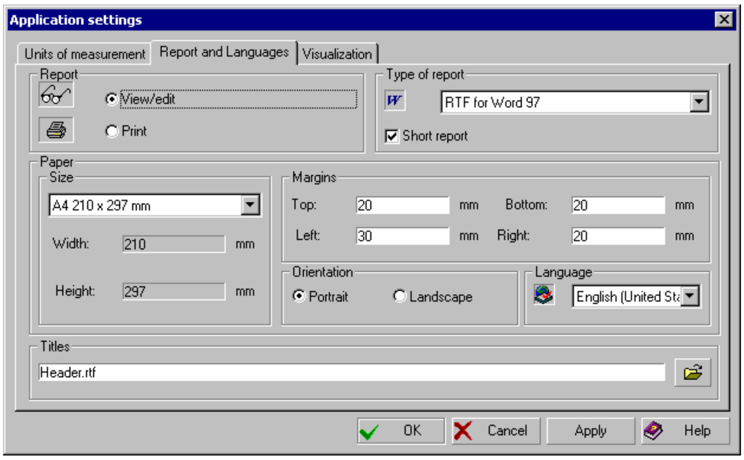


Fig. 6. *The* **Report and Languages** *tab of the* **Application Settings** *dialog box*

The **Visualization** tab is used to choose a font for the text messages on the screen and in the report. Double left clicking the line with the currently selected font opens the standard **Font** dialog box where the font is to be set up.

1.5 Working with tables

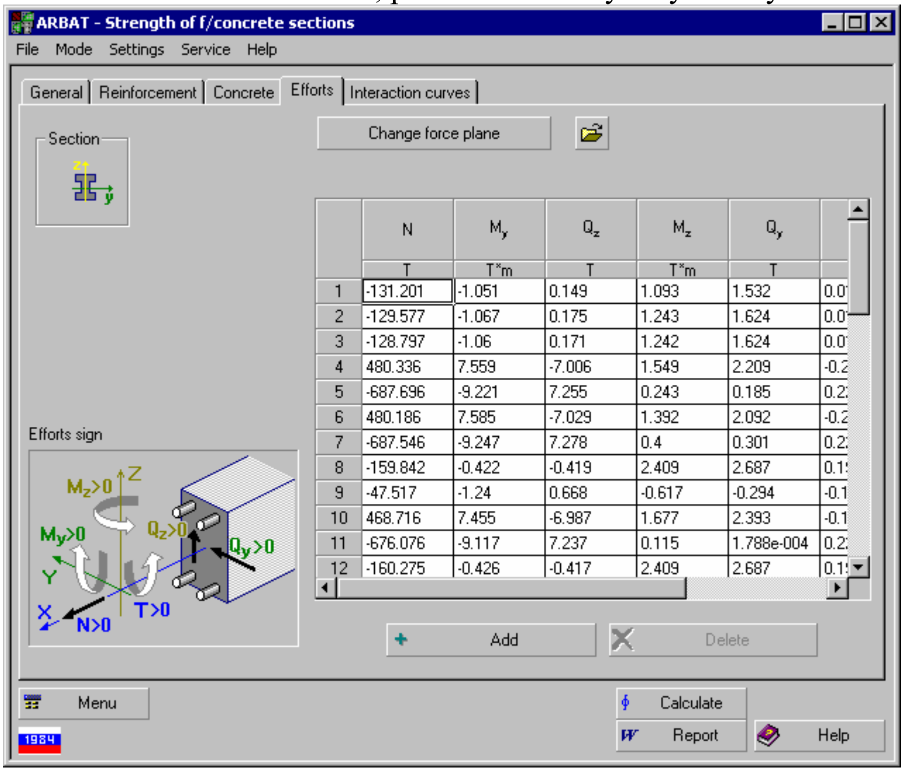
In most cases, source data for any analysis are specified in tables (Fig. 7). The following general rules are used for working with the tabular data:

the data are entered in the table as decimal numbers; a particular separator between the integral and fractional parts of the number (either comma or period) depends on the settings of the Windows environment;

in the cases when the number of rows in the table is assigned by the user, the table has the **Add** and **Delete** buttons next to it; the former adds a new row after the selected one, and the second deletes the selected row or rows;

to select one or more successive rows, put the mouse pointer on the No. of the first one, click and hold the left mouse button, and drag the pointer across the Nos. of the rows to be selected;

to switch between the cells of the table, press the **Tab** key on your keyboard.



	N	M_y	Q_z	M_z	Q_y	
	T	T*m	T	T*m	T	
1	-131.201	-1.051	0.149	1.093	1.532	0.0
2	-129.577	-1.067	0.175	1.243	1.624	0.0
3	-128.797	-1.06	0.171	1.242	1.624	0.0
4	480.336	7.559	-7.006	1.549	2.209	-0.2
5	-687.696	-9.221	7.255	0.243	0.185	0.2
6	480.186	7.585	-7.029	1.392	2.092	-0.2
7	-687.546	-9.247	7.278	0.4	0.301	0.2
8	-159.842	-0.422	-0.419	2.409	2.687	0.1
9	-47.517	-1.24	0.668	-0.617	-0.294	-0.1
10	468.716	7.455	-6.987	1.677	2.393	-0.1
11	-676.076	-9.117	7.237	0.115	1.788e-004	0.2
12	-160.275	-0.426	-0.417	2.409	2.687	0.1

Fig. 7. *An example of a table of stresses*

New rows are added after the selected one, therefore you need to do the following to add a new row before the very first one of the table:

- select the first row in the table and click the **Add** button to add a new one after it;
- select the first row in the table and press the **Ctrl+Insert** keys together; this will copy the contents of the first row to the **Clipboard**;
- select the second (new) row of the table and press the **Shift+Insert** keys together; this will insert the contents of the Clipboard into the cells of the second row, and now the first row can be filled with other data as necessary.

The sequence described above can be used also to copy one or more selected rows of a table.

1.6 Saving data

All analysis modes make it possible to save the entered data in an external file. To do it when in a particular mode, choose the menu item **File|Save As**. This will open a standard Windows dialog box for choosing a folder and a file to store the data. The filename and the extension are specified by the user.

To load the previously saved data later, use the menu item **File|Open**.

2. Information modes

The reference (information) modes provide data about materials listed in the SNiP documents. All values in the respective tables are presented in the same units of measurement as in the design codes; they do not depend on the settings of the application.

2.1 Concrete class

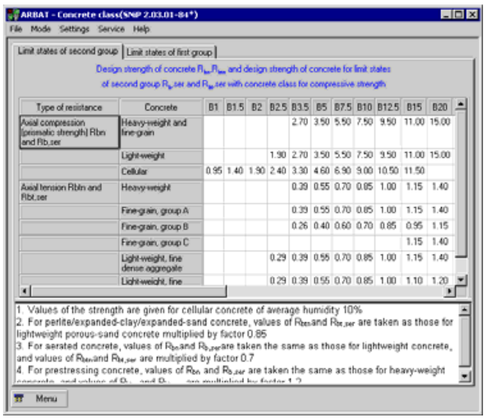


Fig. 8. *The dialog box of the* **Concrete Class** *mode*

A two-tab dialog box of this mode (Fig. 8) contains data presented in Section 2 (Tables 12 and 13) of SNiP 2.03.01-84* [5] (Tables 5.1 through 5.3 of SP 52-101-03 [9]).

2.2 Concrete grade

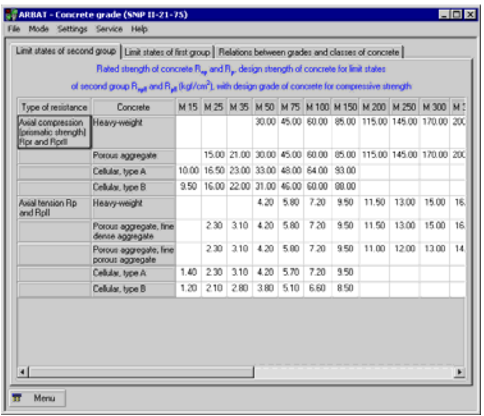


Fig. 9. *The dialog box of the* **Concrete Grade** *mode*

This mode (Fig. 9) can be used to obtain information about the rated and design strength of concrete of various grades in compliance with SNiP II-21-75 [4]. Also, the mode provides data about the relation between the grades and the classes of concrete according to GOST 26633-91 [11].

2.3 Reinforcement

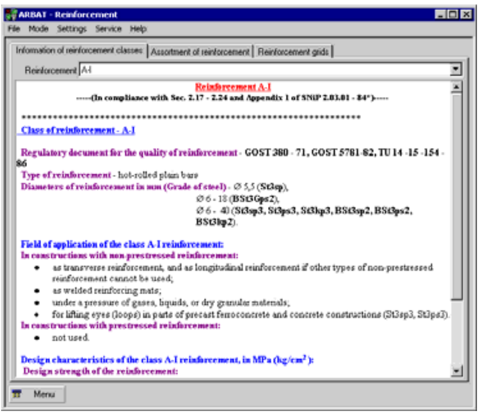


Fig. 10. *The* **Reinforcement** *mode*

This mode presents data about classes of reinforcement and the range of reinforcement products, including an effective area of the cross-section for a particular number of rebars, a theoretical weight per one meter of length, and diameters of reinforcement of various classes. Also, a separate tab presents information about bar-mat reinforcement according to GOST 23279-85 [12].

2.4 Service factors for concrete

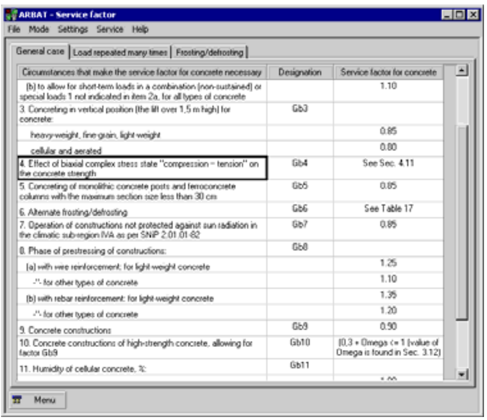


Fig. 11. *The* **Service Factors** (for concrete) *mode*

This mode (Fig. 11) provides information about service factors included in Tables 15, 16, and 17 of SNiP 2.03.01-84* [5] (Sec. 5.1.10 of SP 51-101-03 [9]).

2.5 Deflection limits

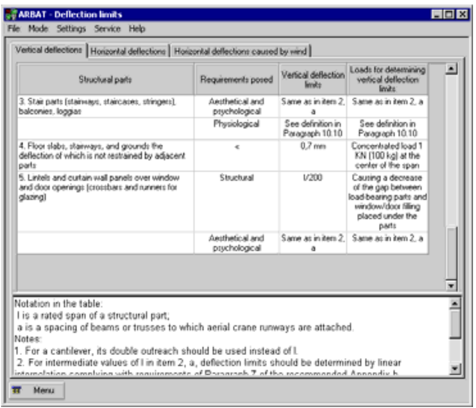


Fig. 12. *The* **Deflection Limits** *mode*

This mode (Fig. 12) provides information about limit values of vertical deflections and also of horizontal deflections caused by cranes or wind; this information is included in Tables 19, 21, and 22 of SNiP 2.01.07-85* [2].

3. General management

Some actions or groups of controls in the application are of a fairly general character and can be used in various working modes for analysis or appraisal. These include actions of building a cross-section, obtaining data about concrete or reinforcement, creating a report etc. The actions and groups of this kind are described below.

3.1 Building a cross-section

Operations that constitute building a cross-section for a structural member can be found in any working mode of the application.

The application works with six types of member sections; their schematics are shown in Fig. 13. To choose a desired section shape, click its respective button in the group, and then specify appropriate sizes and the thickness of the cover (in the reinforcement proportioning modes, the cover is not needed; instead, you have to specify the distance to the center of gravity of the reinforcement).

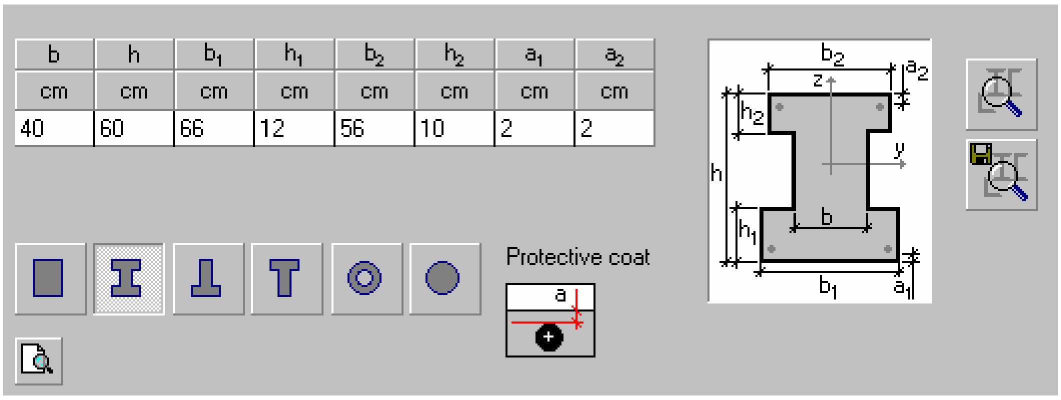
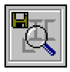


Fig. 13. *Section types*

As you work with the application, you can gradually create a database of user-defined sections. To put a new section in the database, use the button  that calls up the **User-Defined Sections** dialog box where you specify a name for the section (Fig. 14).

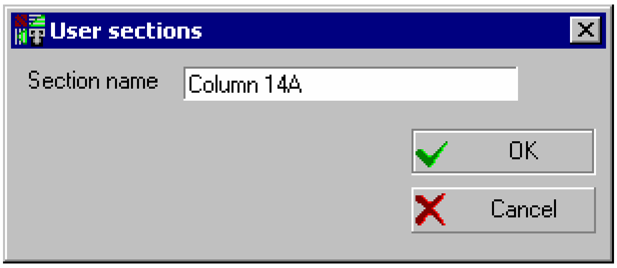
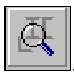


Fig. 14. *The* **User-Defined Sections** *dialog box*

The application does not check the uniqueness of the names used, so the user has to do it by herself.

To access the database and load a previously created section, use the button . The dialog box of the database of user-defined sections (Fig. 15) contains a table of section names and buttons for previewing —  — or renaming (**Rename**) a desired section. To select a section, mark the respective row in the list and click the **Apply** button.

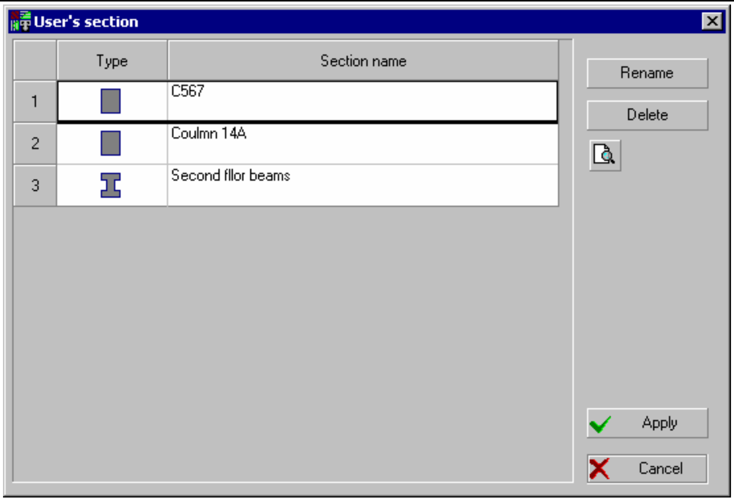


Fig. 15. *A dialog box that lists user-defined sections*

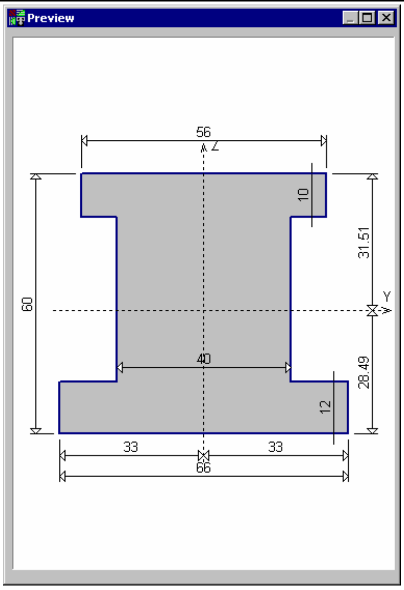
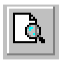


Fig. 16. *The* **Preview** *dialog box*

Clicking the button  lets you view a section in the **Preview** dialog box (Fig. 16).

As the **ARBAT** application works with ferroconcrete sections, it needs to have also information about concrete and rebars specified by the user. See below how to provide this information (Sections 3.2, 3.3).

3.2 Concrete data

Concrete

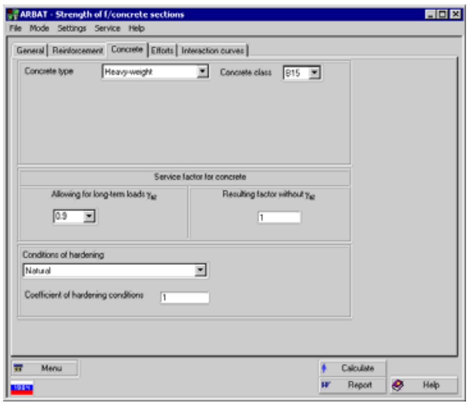


Fig. 17. *The* **Concrete** *dialog box*

The following information should be specified in all analysis modes when preparing data on the **Concrete** tab: type of concrete, class of concrete, service factors for concrete, hardening conditions, and the hardening condition factor (see Fig. 17).

A class of heavy-weight or fine-grained concrete is selected from the **Concrete class** list. For light-weight concrete, you need to choose first its grade by average density, and then its class and its aggregate.

If the analysis complies with SP 52-101-03, only heavy-weight concrete will be available because this document does not regulate the design based on other types of concrete.

The service factor for concrete, $\gamma_{\beta2}$ (or $\gamma_{\beta1}$ if the analysis is based on SP 52-101-03), takes account of the load duration. The value of the factor is set equal to either 1 or 0.9 (item 2*a* in Table 15 of SNiP 2.03.01-84* and Sec. 5.1.10*a* of SP 52-101-03), the default value being 1. In cases when a different value should be used for this factor according to conditions of the analysis, it can be specified directly in the list field.

If a load combination includes short-term loads, the analysis should use $\gamma_{\beta2}$ = 1.1 (independently of what value of it has been already specified) in compliance with regulations of item 2*b* in Table 15 of SNiP 2.03.01-84*.

The service factor for concrete, γ_{β}, is a product of all concrete service factors from Table 15 of SNiP 2.03.01-84* (Section 5.1.10 of SP 52-101-03), except for $\gamma_{\beta2}$ ($\gamma_{\beta1}$). The default value of it is 1.

If the initial elasticity modulus of concrete is different from its table value, then a hardening condition factor should be specified; it is used to make corrections to the value of the modulus (it is to be set only for natural hardening).

3.3 Reinforcement data

Section reinforcement data include information about the classes of longitudinal and transverse reinforcement, service factors for reinforcement, and data concerning the arrangement of the rebars.

The reinforcement class is to be selected from a drop-down list, the contents of which depends on the choice of a particular design code (see Fig. 18). The service factors for reinforcement should be entered by the user; the default value for those is 1.

Reinforcement	Class	Service factor
Longitudinal	A-III	1
Transverse	A-I	1

Fig. 18. *Reinforcement data*

The rebar arrangement information is specified, as a rule, on a separate tab named **Reinforcement** (Fig. 19).

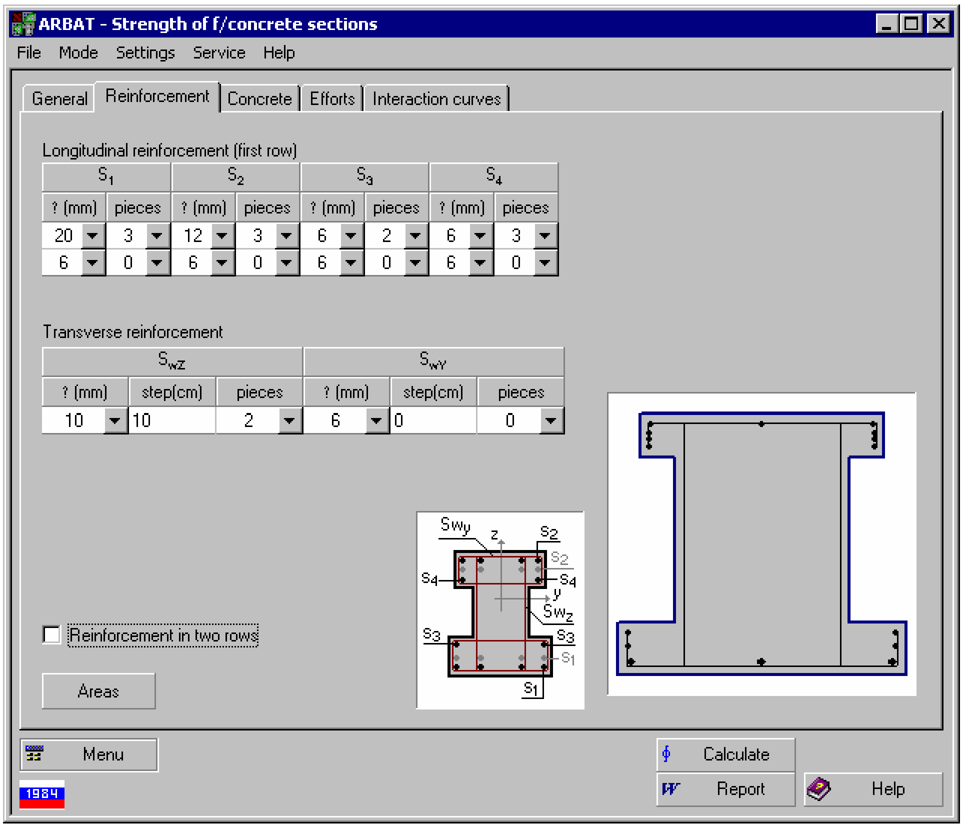


Fig. 19. *An arrangement of rebars*

Three tables are used to specify the properties of reinforcement: **Longitudinal reinforcement (first row)**, **Longitudinal reinforcement (second row)**, and **Transverse reinforcement**.

The **Longitudinal reinforcement (first row)** table contains two lines. The first line of the table presents information about the diameters and the number of rebars in the first row of the bottom, top, and side longitudinal reinforcement. The second line in the table is filled only when the first rows of the bottom (S_1), top (S_2), and/or side (S_3) reinforcement contain rebars of different diameters (the software allows the first row of reinforcement to have rebars of two different diameters).

If the **Reinforcement in two rows** checkbox is ticked, the **Longitudinal reinforcement (second row)** table appears in the window. It specifies the diameter and the number of rebars in the second row of the bottom and top longitudinal reinforcement, and the clear distance between the first and second rows of the bottom (Δ_1) and top (Δ_2) reinforcement.

Data concerning transverse reinforcement are entered in the **Transverse reinforcement** table.

When you are filling in any of the tables, the data entered are interpreted according to these rules:

- the absence of longitudinal reinforcement is indicated by specifying a zero number of the rebars;
- a zero value of the transverse reinforcement spacing is interpreted as the absence of reinforcement of this kind;
- the diameter and the number of rebars are selected from appropriate drop-down lists.

The shape of the section specified and the arrangement of reinforcement in it are displayed in a schematic shown in the window. Use the **Reinforcement Areas** button to see a table that lists areas of the specified reinforcement (Fig. 20).

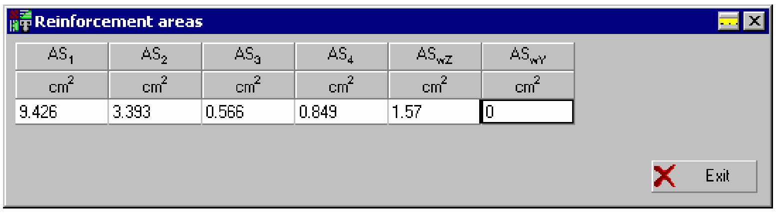


Fig. 20. *The* **Reinforcement Areas** *dialog box*

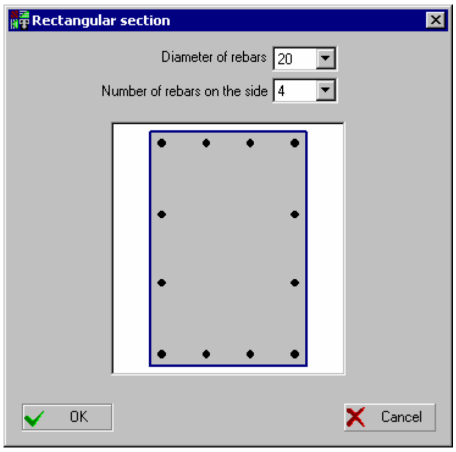
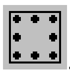


Fig. 21. *The* **Reinforcement of Rectangular Section** *dialog box*

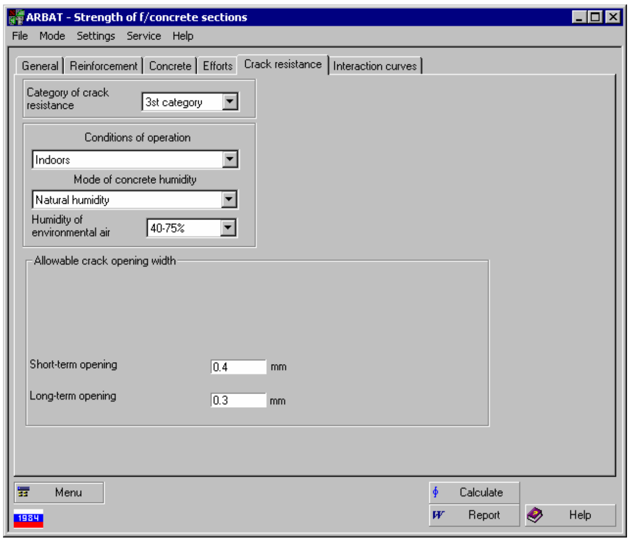
A “uniform” reinforcement can be specified for rectangular sections; this consists of a uniform number of rebars of the same diameter along each face of the section and is used extensively to reinforce columns. To specify the number and the diameter of rebars, use the **Reinforcement of Rectangular Section** dialog box (Fig. 21) which opens by clicking the button . Exiting this dialog will have the **Longitudinal reinforcement (first row)** table filled automatically.

If the number of rebars is greater than two, both lines of the table will be filled. The first line describes two rebars at the corners of each reinforcement S_1 and S_2, and all rebars of reinforcement S_3. The second line contains information about intermediate rebars of reinforcement S_1 and S_2. This arrangement of data in the tables permits to vary the diameter of the corner rebars easily.

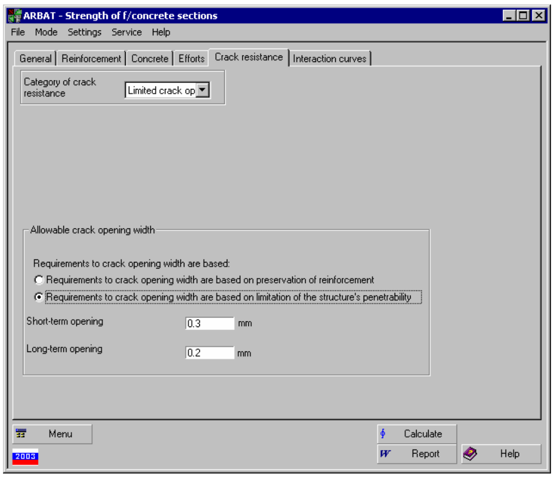
The software checks that the design requirements of Sec. 5.12 of SNiP 2.03.01-84* (Section 5.3 SP 52-101-03) are complied with, and if it is not the case, a warning message is displayed; the requirements define the minimum distance between the rebars. The user can ignore the warning that the requirements are violated by clicking the **Ignore** button, and negative consequences of this ignoring will not be analyzed.

3.4 Crack resistance

In cases when a crack formation and opening analysis is required, the user should provide data about a crack resistance category, an allowable crack opening width etc. All these data are entered on the **Crack Resistance** tab (Fig. 22).



(*a*)



(*b*)

Fig. 22. *Crack resistance data*

If 1st category of crack resistance is set (no cracks), then no additional information is required on this tab.

If the analysis follows SNiP 2.03.01-84* [5] and 3rd category of crack resistance is set (Fig. 22*a*), then you should use appropriate lists to set the construction's operating conditions, a humidity regime for concrete, and the environmental air humidity. Next, set an admissible width of short-term and long-term crack opening (the default values are recommended by SNiP 2.03.01-84* on the basis of operating conditions and humidity for the construction).

If the analysis is to follow SP 52-101-03 [9] and the *Limited crack opening width* item is selected in the drop-down list (Fig. 22*b*), you should specify requirements to the crack opening width (*based on preservation of reinforcement* or *based on limited permeability of constructions*). This will automatically set the maximum allowable widths of long-term and short-term crack opening recommended by Sec. 7.2.3 of SP 52-101-03. The user can edit those values if need be.

3.5 Safety factor for responsibility

All modes of analysis require data about the safety factor for responsibility to be specified in compliance with GOST 27751-88 [1]. The factor can be selected from a drop-down list in accordance with the nature of a particular structural project (it can be of *high*, *normal* or *low* responsibility) or entered by the user directly (in nontrivial situations).

3.6 Generating a report

All analysis modes of the application have the **Report** button. Clicking this button starts the following actions, if there are no mistakes in the source data:

- all analyses are performed;
- an RTF (Rich Text Format) file is created, which contains a list of the source data specified and results of the analysis. Depending on what option is chosen in the **Options** dialog box (whether the **Full messages** or **Short messages** mode is active), the report document will (or will not) contain results of certain intermediate calculations (such as geometric properties of the sections used, values of some, not only the maximum one, of the load-bearing ability utilization factors etc.;

- a Windows application associated with the RTF files is launched. Depending on what is set in the **Options | Other** dialog box (**Print** or **View/Edit**), the application will either print the document immediately or suggest it for viewing and possible editing. In the latter case, it is the user who is responsible for making a hard copy (she can use the printing command of the external application).

Note. The .rtf filename extension is usually associated with the WordPad application. If MS Word is installed on the computer, then it is Word that will be associated with this format. There are differences between the RTF format used by MS Word v.7 or WordPad and that used by MS Word 97. Therefore, our software provides you with the opportunity to choose any of the RTF formats by selecting **Options | Other**.

4. Appraisal

The appraisal modes are used to perform a check of sections of ferroconcrete structural members for compliance with requirements of SNiP 2.03.01-84* using the following criteria[1]:

- the strength against a longidutinal moment in the section — Sec. 3.15–3.20, 3.27–3.28;
- the strength against an ultimate longitudinal force in the section — Sec. 3.26;
- the longitudinal force that takes account of a deflection, with the flexibility $L_0 / i > 14$ — Sec. 3.24, 3.6 of SNiP, Sec. 3.54 of the Guide to SNiP 2.03.01-84* [5];
- the strength against an ultimate longitudinal force that takes account of reinforcement over the height of the section — Sec. 3.64 of the Guide to SNiP 2.03.01-84*;
- the moment resisted by the section during the crack formation — Sec. 4.5;
- the crack opening width (short-term) — Sec. 4.14, 4.15;
- the crack opening width (long-term) — Sec. 4.14, 4.15;
- the oblique crack opening width (short-term) — Sec. 4.17;
- the oblique crack opening width (long-term) — Sec. 4.17;
- the stresses in transverse reinforcement — Sec. 4.17;
- the strength of an oblique strip between oblique cracks — Sec. 3.30;
- the strength of oblique sections without transverse reinforcement — Sec. 3.32;
- the strength of an oblique crack — Sec. 3.31 of SNiP, Sec. 3.31 of the Guide to SNiP 2.03.01-84*[2];
- the strength that takes account of the resistance of concrete in a tensioned area — Sec. 3.8;
- the strength of the section under a torque;
- the resistance of reinforcement to a torque;
- the resistance of side reinforcement to a torque;
- the resistance of transverse reinforcement to a torque.

Note that SNiP 2.03.01-84* provides two strength analysis opportunities for flexural members: one for a so-called general case (Sec. 3.28*) and one based on simplified formulas which work only for a uniaxial state of stress. The application uses the general case always when appraising flexural members, except for the **Section Resistance** mode with the **Reinforcement areas** option (see below).

The reason for this is as follows. For a single-row reinforcement, there is little difference between these two cases of analysis. However, if the reinforcement layout is more complicated (for example, there are rebars on the sides of a rectangular section), then the result will depend on how we refer to those rebars — as a second row of the bottom/top reinforcement or as a side reinforcement (the discrepancy may be as large as 10 to 20%).

The check of sections of concrete structural members for compliance with regulations of SNiP 2.03.01-84* is based on the following criteria:

- the strength against an ultimate longitudinal force in the section — Sec. 3.1–3.5 of SNiP;
- the longitudinal force that takes account of the deflection, with the flexibility $L_0 / i > 14$ — Sec. 3.3, 3.6 of SNiP, Sec. 3.54 of the Guide to SNiP 2.03.01-84*;
- the strength against an ultimate longitudinal force, out of the bending plane — Sec. 1.21, 3.2, 3.3 SNiP;
- the strength against an ultimate moment in the section — Sec. 3.8;
- the strength that takes account of the resistance in a tensioned area of concrete — Sec. 3.5;
- the strength of an oblique strip between oblique cracks — Sec. 3.30;
- the strength of oblique sections without transverse reinforcement — Sec. 3.32;

[1] The analysis for resistance to a torque is based on formulas suggested by Moersch and Rausch (see [19]).

[2] The application uses this section of the Guide instead of Section 3.31 of SNiP because the recommendations from the Guide produce more accurate results.

- the shear force in the absence of oblique cracks — Sec. 4.4 of the Guide to SNiP 2.03.01-84*.

If the analysis is to comply with SP 52-101-03, the appraisal of sections of ferroconcrete structural members will use the following criteria:

- the strength against an ultimate longitudinal force in the section — Sec. 6.2.25, 6.2.31;
- the strength of concrete in tension;
- the strength against an ultimate moment in the section — Sec. 6.2.25, 6.2.31;
- the strength against an ultimate longitudinal force in the section — Sec. 6.2.19;
- the strength against an ultimate moment in the section — Sec. 6.2.9-6.2.15;
- the strains in compressed concrete — Sec. 6.2.21-6.2.31;
- the strains in tensioned reinforcement — Sec. 6.2.21-6.2.31;
- the height of a tensioned area of concrete — Sec. 4.1.2a, 6.2.30;
- the strains in tensioned concrete — Sec. 6.2.30, 6.2.31, 7.2.11;
- the longitudinal force that takes account of a deflection, with the flexibility $L_0/i>14$ — Sec. 6.2.16;
- the strength that takes into account the resistance of concrete in a tensioned area — Sec. 6.1.9, 6.1.12;
- the moment accepted by the section when a crack appears — Sec. 7.2.7;
- the crack opening width (short-term) — Sec. 7.2.3, 7.2.4, 7.2.12;
- the crack opening width (long-term) — Sec. 7.2.3, 7.2.4, 7.2.12;
- the strength against an ultimate longitudinal force in the section — Sec. 6.2.3, 6.2.8, 6.2.16, Sec. 3.50 of the Guide to SP 52-101-03 (further, the Guide to SP);
- the strength against an ultimate longitudinal force in the section, that takes account of reinforcement over the height of the section — Sec. 6.2.17, Sec. 3.58 of the Guide to SP;
- the strength of a concrete strip between oblique sections — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- the strength against a shear force resisted by concrete only — Sec. 6.2.34, Sec. 3.52, 3.71 of the Guide to SP;
- the strength of oblique sections without transverse reinforcement — Sec. 6.2.34, Sec. 3.52,3.71 of the Guide to SP;
- the strength of an oblique section — Sec. 6.2.34, Sec. 3.52,3.71 of the Guide to SP;
- the shear force in the absence of oblique cracks — Sec. 4.28 of the Guide to SP;
- the strength of a concrete strip between oblique sections, against Q_z — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- the strength of a concrete strip between oblique sections, against Q_y — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- the strength of a section under a torque — Sec. 6.2.37;
- the resistance of reinforcement S_1 to a torque;
- the resistance of reinforcement S_2 to a torque;
- the resistance of side reinforcement to a torque;
- the resistance of transverse reinforcement S_{W1} to a torque;
- the resistance of transverse reinforcement S_{W2} to a torque.

The sections of concrete structural members are appraised using the following criteria:

- the strength against an ultimate longitudinal force in the section — Sec. 6.2.25, 6.2.31;
- the strength of concrete in tension;
- the strength against an ultimate moment in the section — Sec. 6.2.25, 6.2.31;
- the strength against an ultimate longitudinal force in the section — Sec. 6.2.19;
- the strength against an ultimate moment in the section — Sec. 6.2.9–6.2.15;

- the strains in tensioned reinforcement — Sec. 6.2.21–6.2.31;
- the strains in tensioned reinforcement — Sec. 6.2.21–6.2.31;
- the height of a tensioned area of concrete — Sec. 4.1.2a, 6.2.30;
- the strains in tensioned reinforcement — Sec. 6.2.30, 6.2.31, 7.2.11;
- the longitudinal force that takes account of a deflection, with the flexibility $L_0/i>14$ — Sec. 6.2.16;
- the strength that takes account of the resistance of concrete in a tensioned area — Sec. 6.1.9, 6.1.12;
- the moment accepted by the section when a crack appears — Sec. 7.2.7;
- the crack opening width (short-term) — Sec. 7.2.3, 7.2.4, 7.2.12;
- the crack opening width (long-term) — Sec. 7.2.3, 7.2.4, 7.2.12;
- the strength against an ultimate longitudinal force in the section — Sec. 6.2.3, 6.2.8, 6.2.16, Sec. 3.50 of the Guide to SP;
- the strength against an ultimate longitudinal force in the section, that takes account of reinforcement over the height of the section — Sec. 6.2.17, Sec. 3.58 of the Guide to SP;
- the strength of a concrete strip between oblique sections — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- the strength against a shear force resisted by concrete only — Sec. 6.2.34, Sec. 3.52, 3.71 of the Guide to SP;
- the strength of oblique sections without transverse reinforcement — Sec. 6.2.34, Sec. 3.52, 3.71 of the Guide to SP;
- the strength of an oblique section — Sec. 6.2.34, Sec. 3.52,3.71 of the Guide to SP;
- the shear force in the absence of oblique cracks — Sec. 4.28 of the Guide to SP;
- the strength of a concrete strip between oblique sections, against Q_z — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- the strength of a concrete strip between oblique sections, against Q_y — Sec. 6.2.33, Sec. 3.52 of the Guide to SP;
- the strenght of the section againts a torque — Sec. 6.2.37.

4.1 Resistance of ferroconcrete sections

This mode implements a function of determining the load-bearing ability of any of the sections available in the application's environment, depending on the arrangement, diameter (area), and class of reinforcement, the class of concrete, the operating conditions, and the admissible crack opening width.

In a general case, the analysis takes into account a longitudinal force, bending moments, a torque, and shear forces that act in the principal planes of inertia.

SNiP 2.03.01-84* does not regulate any check of ferroconcrete members for limit states of second group under the action of moments in two planes. Therefore, if the crack resistance appraisal mode is turned on (with the selected design code being SNiP 2.03.01-84*), then only an eccentrical compression/tension with the eccentricity in one plane is considered.

General

The appearance of the **General** tab depends on the **Reinforcement areas** checkbox on the **Resistance of f/c sections** button of the main window. If the checkbox is off, the source data for the analysis will consist of rebar arrangement information, and the **General** tab (Fig. 23) should have these pieces of data specified:

- the geometric and effective lengths of members;
- the random eccentricities;
- the shape and sizes of the section;
- the thickness of the reinforcement's protective coating;
- the service factors for reinforcement;

- the class and the diameter of longitudinal reinforcement (a two-row arrangement can be used);
- the class, the diameter, and the spacing of transverse reinforcement, and the number of stirrups.

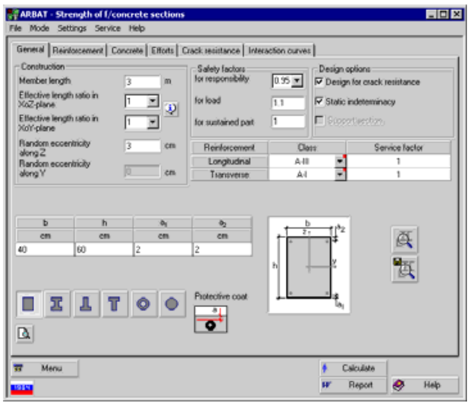


Fig. 23. *The* **General** *tab* (the **Reinforcement areas** checkbox is disabled)

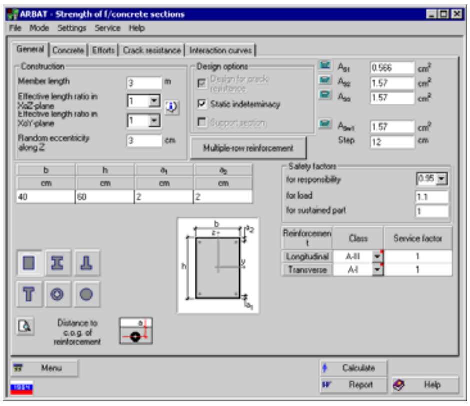


Fig. 24. *The* **General** *tab* (the **Reinforcement areas** checkbox is enabled)

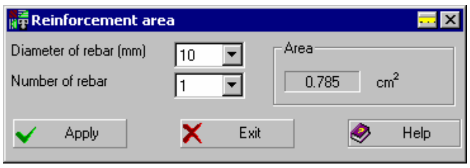


Fig. 25. *The dialog box of the* **Reinforcement Area** *calculator*

Checkboxes are used to indicate that a check is needed for the second limit state (**Crack resistance analysis**), to point out that a section belongs to a statically determine construction, or to state that there is second row of reinforcement.

SP 52-101-03 makes it possible to do the check that takes into account the crack resistance under moments in two planes, therefore the checkbox is always on for this design code document.

The analysis makes use of the given safety factors for responsibility and for load, and of the sustained-part factor.[3]

Note that the value of the *sustained-part factor* can be even less than 0 or greater than 1. It can be seen from the following example. Suppose some combination includes a sustained loading in which the longitudinal force is 1.0T and a short-term loading the action of which produces the longitudinal force of –0.1T. Then the combination that consists of the sum of the two loadings will have the overall longitudinal force of 0.9T while the sustained part of the comnbination will still be 1.0T. Thus the *sustained-part factor* is $k = 1{,}0 / 0{,}9 = 1{,}11$. Similarly, there can be a situation in which $k < 0$.

The formwork sizes of the section and the thickness of the cover are specified following standard rules as described in Section 3.1.

The effective length factors are specified according to Sec. 3.25 of SNiP 2.03.01-84* (Sec. 4.2.6 of SP 52-101-03). The button provides reference data concerning the effective lengths according to Sec. 3.25 of SNiP 2.03.01-84* (Sec. 6.2.18 of SP 52-101-03).

If the random eccentricity is set to zero, the analysis will use its value calculated in compliance with Sec. 1.21 of SNiP 2.03.01-84* (Sec. 1.2.6 of SP 52-101-03). If a nonzero random eccentricity is specified, then the greater of the two values will be used: either 1 cm or one specified (according to Sec. 3.50 of the Guide to SNiP 2.03.01-84* and Sec. 3.6 of the Guide to SP 52-101-03).

To specify the properties of reinforcement and the arrangement of rebars, use a standard group of controls and the **Reinforcement** tab (see Section 3.3).

If the **Reinforcement areas** checkbox on the **Resistance of sections** button in the main window is on,

[3] Sec. 1.12* of SNiP 2.03.01-84* says “...sustained loads include also a part of the full value of short-term loads...». The ARBAT application takes account of that part of the loads by using a *sustained-part factor*.

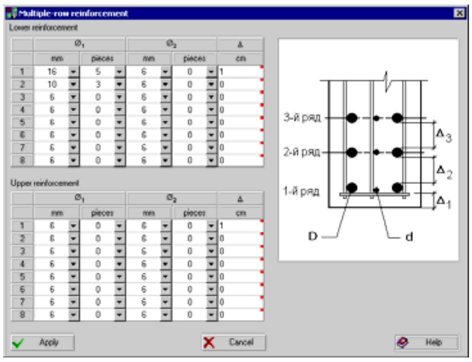


Fig. 26. *The* **Multiple-Row Reinforcement** *dialog box*

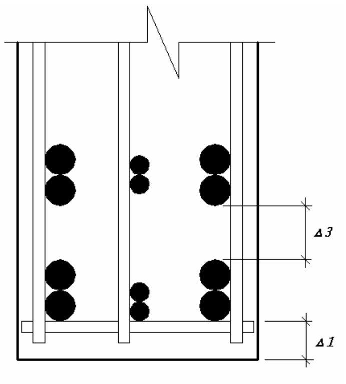


Fig. 27. *An arrangement of rebars of different diameters*

then the required source data include areas of longitudinal and transverse reinforcement (the **General** tab looks as shown in Fig. 24). Also, this mode does not require specifying a thickness of the cover; instead, **distances to the centers of gravity** of rebars should be specified.

The area of reinforcement can be calculated using the **Reinforcement Area** calculator (Fig. 25). The calculator is launched by clicking one of the buttons to the left of the respective edit fields.

To calculate the reinforcement area with the calculator, choose a line in the top list that corresponds to the needed rebar diameter and specify the number of rebars in the bottom list. Clicking the **Apply** button will close the calculator and put the calculated value of the area in the respective edit field of the **General** tab.

The tab contains the **Multiple-Row Reinforcement** button that serves to enter data for multiple-row reinforcement. Clicking this button will open a dialog box under the same name, which is shown in Fig. 26.

This dialog is used to determine the distance from the center of gravity of the longitudinal reinforcement section to the nearest face of the structural member, when there are multiple rows of reinforcement in the section of the structural member (a column or a beam). Rebars of different diameters can be placed in the same row.

If there are rebars of different diameters in the first row (one nearest to the face), the rebars of the smaller diameter are moved toward the face of the member, onto a line tangential to the rebars of the greater diameter (Fig. 27). The rebars in the next rows, if they have different diameters, are assumed to have their centers belonging to one straight line.

The table is used to specify the diameter of the rebars and their number in the current row, and the clear distance from the rebars of that row to those of the preceding row. The clear distance is replaced with the concrete cover to reinforcement for the rebars of the first row. If the distance of zero is specified between the rebars ($\Delta = 0$), the rebars are assumed to be immediately adjacent to one another.

After the **Multiple-Row Reinforcement** dialog is closed by clicking the **Apply** button, the calculated results appear in the **Distance to cent.grav. of reinforcement** edit fields on the **General** tab. To edit the data, you can call up the **Multiple-Row Reinforcement** dialog box again, where you will find the information last entered.

Having prepared the source data, you can launch the analysis by clicking the **Calculate** button.

Figs. 28 through 30 present arrangements of longitudinal and transverse reinforcement and symbols of reinforcement used in drawings of various sections.

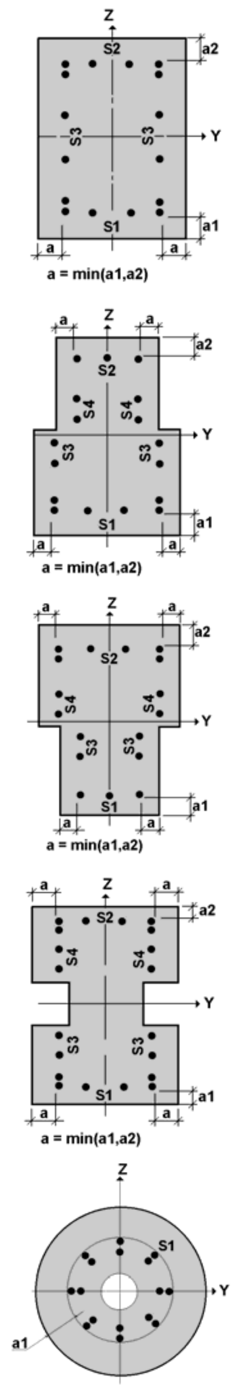


Fig. 28. *Arrangements of longitudinal reinforcement*

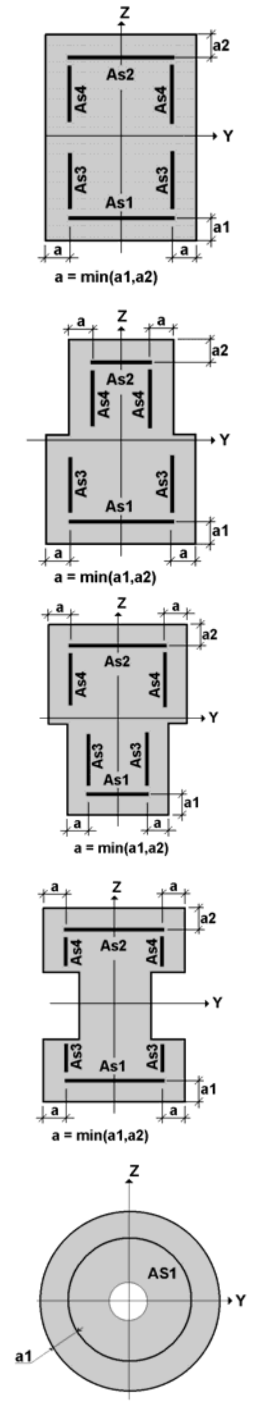


Fig. 29. *Arrangements of “areas” of longitudinal reinforcement*

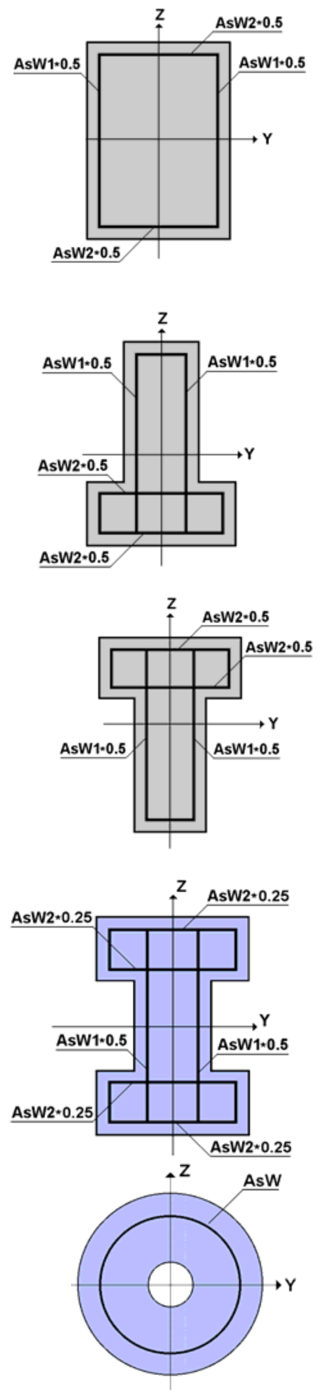


Fig. 30. *Arrangements of transverse reinforcement*

Concrete

Properties of concrete are specified on the **Concrete** tab in the same way as described in Sec. 3.2.

Stresses

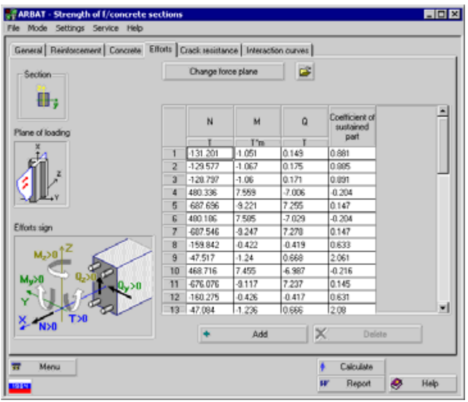


Fig. 31. *The* **Stresses** *tab*

The **Stresses** tab (Fig. 31) is used to enter stresses acting in a cross-section of the structural member. It presents a schematic of the section that depicts its principal axes of inertia and positive directions of the stresses. The tab contains a table used to define stresses caused by one or more loadings on the section. The number of lines in the table corresponds to the number of loadings. The table can be filled also by importing data from **SCAD** that describe design stress combinations (DSCs). A file with the .rsu extension is created in the **Element Information** mode of the SCAD software system and can be imported by clicking the button located above the table.

The **Section Resistance** mode provides a capability of changing the plane of loading (the **Change the plane of loading** button clicking which will replace M_y by M_z and Q_y by Q_z).

Crack Resistance

Crack resistance data are specified on the **Crack resistance** tab according to rules defined in Section 3.4.

Interaction Curves

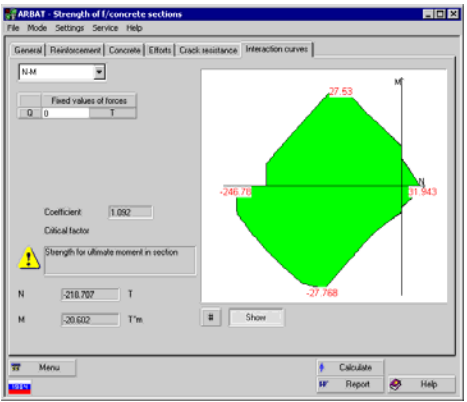


Fig. 32. *The* **Interaction Curves** *tab*

Results of the analysis are displayed on the **Interaction Curves** tab (Fig. 32) and represented as a curve that bounds the section's load-bearing ability area under the action of a couple of stresses selected by the user. The couple is assumed to be admissible if the load-bearing ability utilization factor for the section is $K_{max} \le 1$. All the other stresses are assumed to equal the values that have been specified for them in the **Fixed values of stresses** group.

When the appraisal is based on SNiP 2.03.01-84* and takes account of crack resistance, two couples of stresses can be used: $N–M_y$ (the longitudinal force and the bending moment) or $N–Q_z$ (the longitudinal force and the shear force). The results concerning the latter couple will be displayed only if the diameter and the spacing of transverse reinforcement have been specified in the **General** tab.

If the crack resistance analysis is not required or the appraisal checks have been done in compliance with SP 52-101-03, then the interaction curve can be built for the following stress couples:

$N–M_y$; $N–M_z$; $M_y–M_z$; $N–Q_z$; $N–Q_y$.

The mouse pointer can be used to review the whole stress variation area shown on the diagram. Every position of the pointer conforms to a couple of numerical values of the stresses, which are displayed in fields to the left of the diagram. At the same time, a maximum value of the utilization factor that corresponds to those values is displayed together with the check type that has produced this maximum.

Clicking the right mouse button while the mouse pointer is in any position in the interaction curve's display field produces a stress diagram for all appraisals being performed.

A report can be generated to document the results of the analysis (the **Report** button).

4.2 Resistance of concrete sections

This mode is similar to the **Resistance of Ferroconcrete Sections** mode. A natural difference between them is that there is no need to specify the reinforcement data (Fig. 33). The current version of the software implements the analysis of rectangular and T-shaped sections.

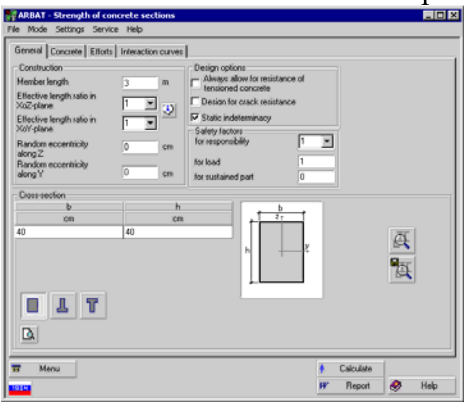


Fig. 33. *The* **General** *tab*

Also, a special type of analysis is available. It is turned on by the **Always allow for resistance of concrete in tension** checkbox. The usage of the checkbox is obvious from its name. Formally, its application leads to a violation of SNiP which allows the resistance of concrete against tension to be taken into account only in a pure bending analysis. On the other hand, if we followed the SNiP regulations directly, there could be a whimsical situation when the presence of even a minimal longitudinal force made it impossible to take into account the tensional behavior of concrete and thus decreased the calculated load-bearing ability by leaps. We suggest that the user make her own decision whether to use this nonstandard approach or not.

4.3 Appraisal of a beam

This mode is to appraise the strength and crack resistance of a multiple-span continuous beam of a constant cross-section in compliance with regulations of SNiP 2.03.01-84* (SP 52-101-03). Bending of the beam in one plane of loading under distributed or concentrated loads is under consideration. Loads are combined into loadings (loading patterns) which can be classified by their physical origin and properties into permament, temporary but sustained, short-term, wind, and snow. The checks of all sections involve automatically created design stress combinations (DSC). DSC factors which take into account the nature of the loading are assigned by the software automatically on the basis of regulations from SNiP 2.01.07-85*.

The beam is supposed to be free from longitudinal forces; only the following force actions are taken into account:

M, a bending moment;

Q, a shear force.

The analysis can be done for a beam of a rectangular, tee or double tee section. A particular arrangement of the rebars should be specified. The number and the diameter of the rebars can be different on different segments along the beam.

Data are prepared on the tabs named **General**, **Loads**, **Concrete**, **Crack Resistance**, and **Segments**, and the results are reviewed on the **Appraisal Results** tab.

General

The **General** tab (Fig. 34) is used to assign the number and lengths of the spans, define the class and service factors for longitudinal and transverse reinforcement, specify the section's type and the thickness of the cover.

The number of spans (five at the most) is selected from the repective list. The presence of cantilevers or immuring (clamped ends) is indicated by checking the respective checkboxes. The lengths of the spans and cantilevers are specified in the appropriate edit fields.

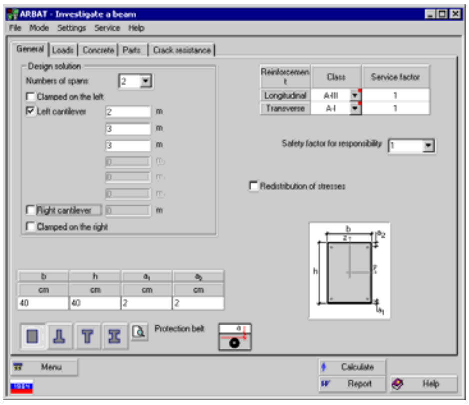


Fig. 34. *The* **General** *tab*

To assign the geometric properties of the beam, follow these steps:

- set a desired number of spans in the **Number of spans** list (at most five, not counting the cantilevers);
- the cantilevers (if there should be any) are defined by checking the respective checkboxes, **left cantilever** and/or **right cantilever**;
- enter the lengths of the spans and cantilevers in the respective edit fields.

The shape of the section is defined following standard rules described in Section 3.1.

A special checkbox, **Allow for redistribution of stresses**, makes it possible to switch to a mode where the curves of the moments and shear forces are calculated taking into account a possible redistribution of the stresses. This mode complies with the recommendation (see [20]) that sets the maximum level of moment (force) redistribution equal to 30%. It should be noted that the phenomenon of redistribution is based on plastic hinges, i.e. an unlimited crack opening width. Therefore, when the **Allow for redistribution of stresses** checkbox is on, the crack resistance analysis is always off.

Loads

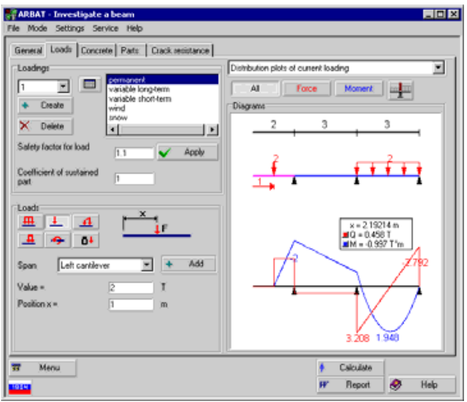


Fig. 35. *The* **Loads** *tab*

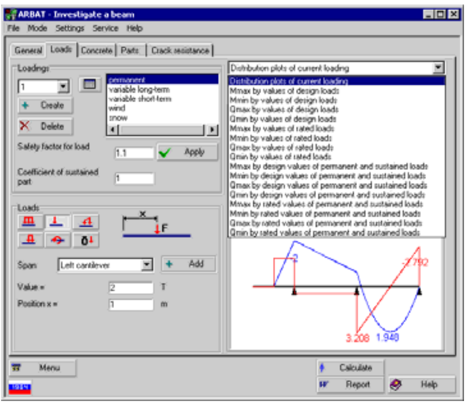


Fig. 36. *The* **Loads** *tab* with the list of combinations opened

The appraisal of a given reinforcement makes use of DSCs built from multiple loadings. The loadings are added on the **Loads** tab (Fig. 35) following these steps:

- click the **Create** button (which means 'create a loading');
- choose a type of the loading from the list (permanent, temporary but sustained, short-term, wind, or snow);
- assign the load type (click a button that depicts a desired distributed or concentrated load);
- enter the design value of the load;
- specify the sustained-part factor for snow or a short-term load;
- specify the safety factor for load;
- use the **Span** drop-down list to choose a span or a cantilever where the load is to be applied (the selected span will be displayed in red);
- click the **Add** button;
- create and add other loads included in the current loading.

Also, the application provides the capability of defining a self-weight load on the beam (button).

Depending on the load's type, its properties may include:

- an intensity of the load — for distributed loads;
- an intensity, a location, and a width of application — for a distributed load upon a part of the span;

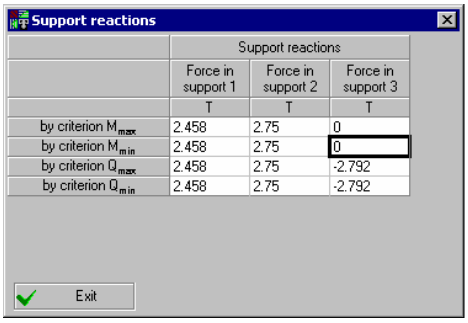


Fig. 37. *The* **Support Reactions** *dialog box*

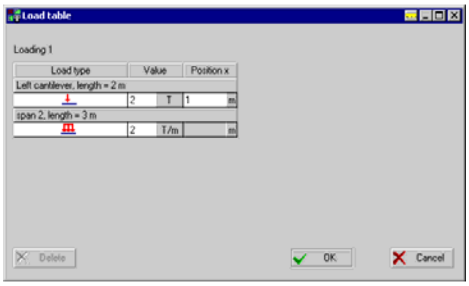


Fig. 38. *The* **Table of Loads** *dialog box*

- a magnitude of the force and its location in the span — for a concentrated force;
- a magnitude of the moment and its location in the span — for a concentrated moment;
- a magnitude of the load at the beginning and at the end, a binding point (a point of application) — for a trapezoid load.

No additional data are required to specify the self-weight load.

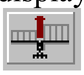
All loads newly specified will belong to the current loading until the next one is created. It should be noted that the **Sustained-part factor** and the **Safety factor for load** relate to the whole loading, and the calculation of DSC will use the last entered value. Having changed any of the factors, click the **Apply** button.

To delete the current loading, click the **Delete** button.

To edit a previously added loading, choose its No. from the list of Nos. of loadings.

If the **Diagrams** field (the list of data to be displayed) has the **Distribution curves for current loading** item selected, then entering every new load will be followed by displaying the loading's schematic and the distribution curves of the moments and shear forces for the current loading.

The **All**, **Force**, and **Moment** buttons let you choose a mode for displaying the distribution curves. The moment and shear force diagrams can be displayed together, or you can have only moments or only shear forces displayed.

Also, clicking the button  will display the calculated values of the support reactions (Fig. 37).

In addition to the distribution curves of the moments and of the shear forces for each of the involved loadings, the application calculates stresses which can be caused in every section of the beam by the combination of the given loads. The combining rules comply with SNiP 2.01.07-85*. The list of the combinations (Fig. 36) is located at the top of the distribution curve dialog box and contains:

- extreme values of the moments and their respective shear force values;
- extreme values of the shear forces and their respective moment values.

The combinations are defined for either design or rated values of the loads, and also for the case when only the permanent and sustained loads are applied, with their design or rated values taken into account.

As you move the mouse across the distribution curve area, the screen shows values of the moment and the shear force in the section (Fig. 35) which conforms to the current position of the pointer.

If a load's value should be changed or the load itself removed from a loading, use the table of loads to do it (the button  in the **Loadings** group). The **Table of Loads** dialog box (Fig. 38), that opens by clicking this button, displays the type, the magnitude, and the location of this particular load. Changes made to the properties of the load will be confirmed and saved after you exit the table and click the **OK** button.

Concrete

Properties of concrete are specified on the **Concrete** tab in the way defined in Section 3.2.

Crack resistance

Crack resistance data are specified on the **Crack Resistance** tab according to the rules defined in Section 3.4.

Segments of a beam

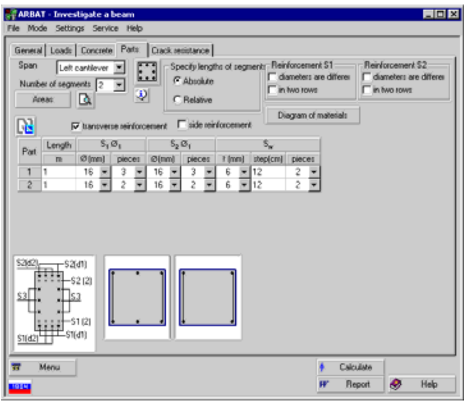


Fig. 39. *The* **Segments** *tab*

To specify reinforcement for a beam, each span or cantilever of it is divided into a number of segments (five at the most). The reinforcement is assumed to stay the same within one segment. Each span or cantilever can be divided into its particular number of segments. The numbering of the segments goes from left to right. Source data concerning the arrangement of reinforcement on the segments are specified on the **Segments** tab (Fig. 39). This information is to be entered separately for each span (cantilever). The dialog box contains the following groups of controls:

- **Span**, where the number of segments on each span or cantilever is to be defined;
- **Lengths of segments specified as**, where you use radio buttons to define the way the lengths are specified:

  Absolute — to specify the lengths of the segments in units of length (the units of length for the spans are defined in the **Application Settings** dialog box);

  Relative — to specify the lengths of the segments as percentage of the total span length;
- **Bottom reinforcement** S_1 — this group is used to define the following, using appropriate checkboxes:

  Different diameters — the presence of rebars of different diameters in the same row of the bottom reinforcement;

  In two rows — the bottom reinforcement should be installed in two rows (different diameters are allowed only in the first row of the bottom reinforcement). If the **In two rows** checkbox is turned on, you should specify the clear distance between the rows of reinforcement in the **Distance between rows** edit field (always in millimeters);
- **Top reinforcement** S_2 — similarly to the bottom reinforcement;
- the **Transverse reinforcement** checkbox — when turned on, the checkbox indicates there is transverse reinforcement on the segment;
- the **Side reinforcement** checkbox — when turned on, the checkbox indicates the presence of side

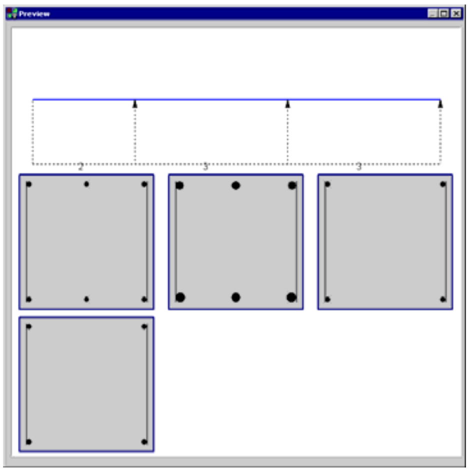


Fig. 40. *The* **Reinforcement Layout** *dialog box*

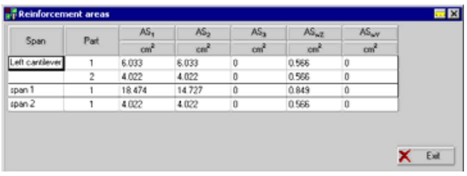


Fig. 41. *The* **Reinforcement Areas** *dialog box*

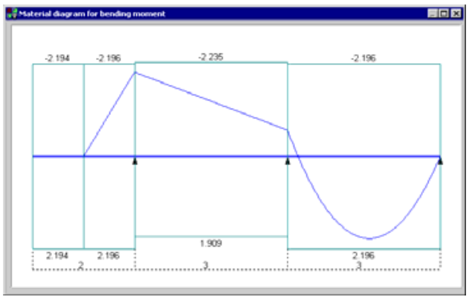


Fig. 42. *A material distribution curve*

reinforcement on the segment;

- the table of reinforcement properties, where the data should be entered for each particular span selected in the **Span** list. The number of columns in the table depends on settings made for the current span, and the number of lines depends on the number of segments the span is divided into.

The lower part of the dialog displays data concerning the arrangement of reinforcement on segments of the current span.

The reinforcement layout thus specified goes through verification in the **Reinforcement Layout** dialog box (Fig. 40) which opens after clicking the button . Reference information that includes examples of beam reinforcement can be obtained with the button .

A “uniform” reinforcement can be specified for rectangular-section beams using the button (see Section 3.3).

The button lets you export the longitudinal rebar arrangement data to a format which can be then imported into the AllPlan FT design software developed by Nemetchek. clicking this button will open a standard Windows dialog box that asks for a folder and a filename. Specify that information, and the data will be saved to a file with the **.a** extension.

Follow these steps to enter data on the **Beam Segments** tab:

- choose the name of a span (cantilever) from the **Span** list, for which the data will be entered;
- set the number of segments in the selected span in the **Number of segments** list;
- choose a radio button with the desired length specification method in the **Lengths of segments specified as** group;
- turn on the checkboxes in the **Bottom reinforcement S_1** and **Top reinforcement S_2** groups to define additional features of reinforcement in the span (if the reinforcement is arranged in two rows, the distance between the rows should be specified);
- turn on the **Transverse reinforcement** and/or **Side reinforcement** checkboxes if you need transverse and/or side reinforcement to be defined;
- fill in the table by specifying the span lengths, the diameter and the number of rebars;
- repeat the above steps for other spans of the beam.

Having finished the specification of source data for all spans, click the **Calculate** button.

Information about the areas of reinforcement **AS_1, AS_2,...** can be obtained in the **Reinforcement Areas** dialog box (see Fig. 41) which appears after you click the **Areas** button.

This tab also contains a button that calls up a dialog box displaying a distribution curve of materials (Fig. 42). The curve is built under the assumptions that the safety factor for load is 1.1 and the sustained-part factor is 1.0.

Appraisal results

This tab (Fig. 43) opens automatically as soon as you activate the appraisal mode (click the **Calculate** button).

Results of the analysis that has been performed are displayed as a table. The first and second columns of the table list spans and segments. For each segment, the **Criterion** column gives the description of a criterion for which the limitation utilization factor achieves its maximum, and the **Factor** column gives the value. The last column of the table presents a graphical representation of the coefficients, where red bars denote factors greater than one.

The dialog box provides the capability of switching to the deflection analysis mode (click the **Deflections** button to do it) where all source data including the loadings are passed automatically. When switching from the multiple-span appraisal and reinforcement proportioning modes to the deflection analysis mode, all loadings are passed to the latter. The load for the analysis is a combination of the loadings specified in the respective dialog box (Fig. 44). By default, all loadings in the combination, except for the first one, have their factors equal to zero.

A report can be generated on the basis of the analysis data (the **Report** button).

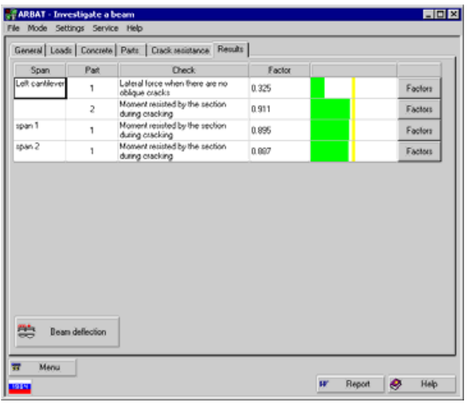


Fig. 43. *The* **Appraisal Results** *tab*

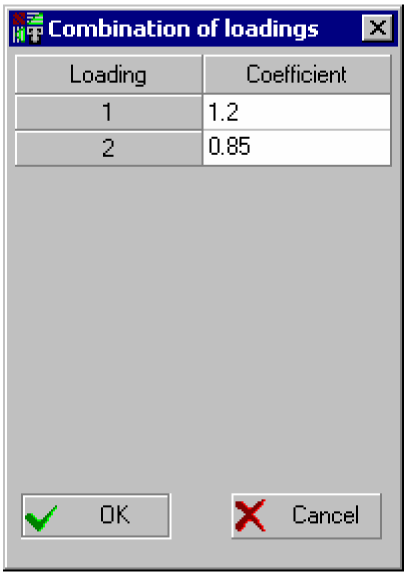


Fig. 44. *The* **Combination of Loadings** *dialog box*

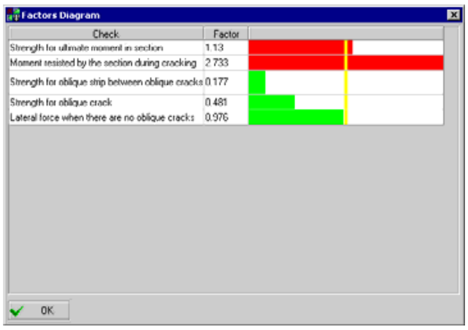


Fig. 45. *The* **Criteria Diagram** *dialog box*

Clicking the **Criteria** button in the respective row of the table that contains the results of the segment's appraisal will open the **Criteria Diagram** dialog box (Fig. 45) with a detailed information about all appraisal checks done for the segment and the respective factors.

4.4 Appraisal of a single-span beam

This multiple-tab dialog box (Fig. 46) is used to perform an appraisal of a single-span beam.

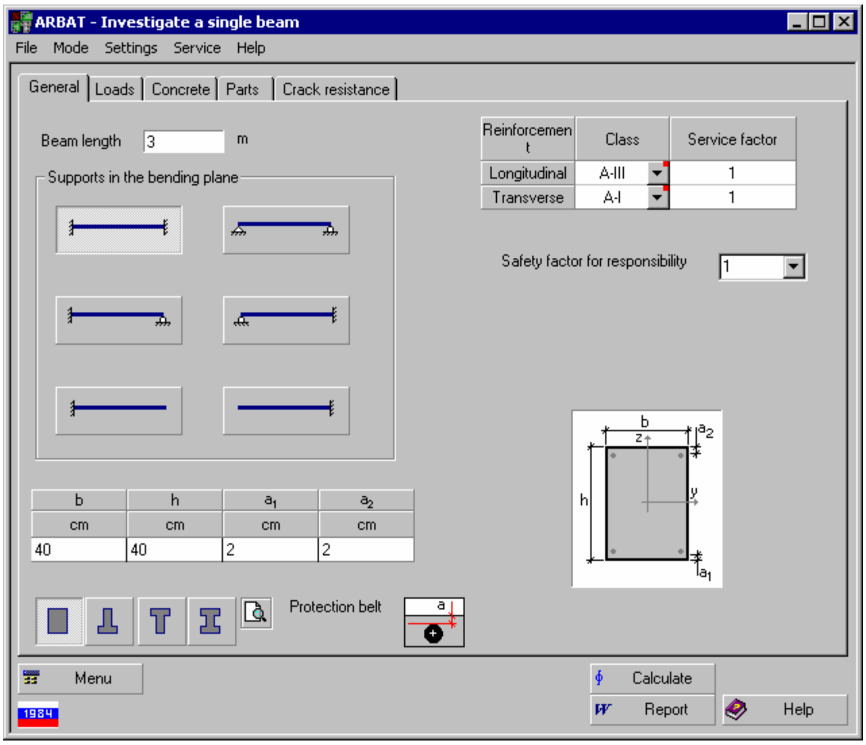


Fig. 46. *The* **Appraisal of Single-Span Beam** *dialog box*

This mode is similar to **Appraisal of Beam** (see Section 4.3). There are differences, though: only one length is specified for the beam, and the user is supposed also to choose a method of fixation of the beam in its bending plane. The fixation is selected by clicking a button that depicts the desired method of it. Also, the **Loads** tab does not require the user to indicate the span to which the load is applied.

4.5 Beam deflection

This mode is used to calculate the deflections of a multiple-span beam under a given load. The deflections are calculated for rectangular, tee, and double tee sections in compliance with Sec. 4.31 of SNiP 2.03.01-84*. (Sec. 4.3 of SP 52-101-03). The curvature takes into account cracks in the tensioned area according to Sec. 4.27 of SNiP 2.03.01-84* (Sec. 4.3.3 of SP 52-101-03). Source data are prepared on the **General**, **Loads**, **Concrete**, **Operating conditions**, and **Segments** tabs, and results are reviewed on the **Deflections** tab that opens after the calculation is done (the **Calculate** button).

The software provides the capability of switching between the appraisal modes and the reinforcement proportioning modes for multiple-span beams, on one hand, and the deflection analysis mode, on the other hand. The source data already specified in these modes will be carried over automatically.

`General`

The **General** tab is used to set a number and lengths of spans, a class and service factors for longitudinal and transverse reinforcement, a type of the beam's section and its sizes, a thickness of the cover.

This tab is similar to the respective tab in the **Appraisal of Beam** mode (see Section 4.3).

`Loads`

The deflections are calculated for a single loading that may include concentrated and distributed loads. The loads are specified on the tab under the same name; working with it is similar to working with the **Loads** tab of the **Appraisal of Beam** mode (see 4.3). The only difference is that the analysis is nonlinear (it allows for the effect of cracks on the curvature), therefore it involves only one loading.

To delete all loads, use the **Delete** button.

Concrete

Properties of concrete are specified on the **Concrete** tab according to the rules defined in Section 3.2.

Operating conditions

Data concerning the operating conditions for the construction are defined on the **Operating Conditions** tab following the rules described in Section 3.4.

Segments of beam

Segments of the beam are described on the tab under the same name as in the **Appraisal of Beam** mode (see Section 4.3).

Deflections

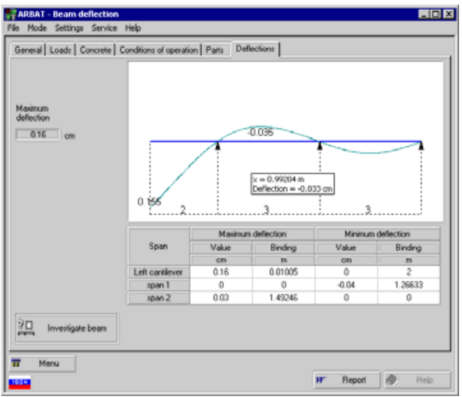


Fig. 47. *The* **Deflections** *dialog box*

This tab (Fig. 47) opens automatically after you turn on the deflection analysis mode (by clicking the **Calculate** button). Results of the analysis will be displayed as a distribution curve or as a table (after you click the **Table** button). The maximum deflection value is indicated in the respective field. A report can be generated after the results of the analysis (to get it, click the **Report** button).

If you click the **Appraisal of beam** button, you switch to the beam appraisal mode. All data concerning the section, reinforcement, loads will be preserved.

4.6 Deflection of a single-span beam

This mode is used to determine the deflections of a single-span beam. The mode is similar to the **Beam Deflection** mode (see Section 4.5). Peculiarities of the source data specification are obvious — they follow from the difference between a single-span beam and a multiple-span one.

4.7 Appraisal of a column

This mode is used to perform the checks of constant-section columns for strength and crack resistance. An eccentric compression/tension with a biaxial eccentricity is under consideration. All appraisals involve automatically created design stress combinations (DSCs).

DSC factors which take into account the nature of loading are assigned by the application on the basis of regulations from SNiP 2.01.07-85*.

SNiP 2-03-01-84* does not define any rules for the second limit state analysis in a biaxial state of stress. If this document is chosen to be the working design code, the following limitation takes place:

the check for the first limit state should take into account the action of the following types of stresses:

N, a longitudinal force;

M_y, a moment that bends the member in the XoZ-plane with its vector along the Y-axis;

M_z, a moment that bends the member in the XoY-plane with its vector along the Z-axis;

Q_z, a shear force along the Z-axis;

Q_y, a shear force along the Y-axis;

M_t, a torque with its vector along the X-axis;

the checks for the first and second limit states should take into account only these types of stresses:

N, a longitudinal force;
M_y, a moment that bends the member in the XoZ-plane with its vector along the Y-axis;
Q_z, a shear force along the Z-axis.

The analysis can be done for a column of a rectangular, tee, double-tee, annular, or round section. The position of rebars in the section is supposed to be known beforehand and constant over the length of a particular segment; it is the user who defines the number and lengths of segments that the column is to be divided into.

Source data are prepared on the **General**, **Loads**, **Concrete**, **Segments**, and **Crack Resistance** tabs, and the results are reviewed on the **Appraisal Results** tab.

General

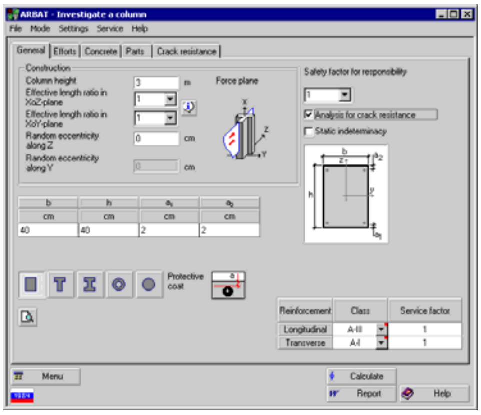


Fig. 48. *The* **General** *tab*

The **General** tab (Fig. 48) is used to specify the height of a column, its effective length factors, values of the random eccentricity. Following the standard rules (see Section 3), you provide information about the section and the reinforcement classes for the column.

Also, the state of the **Crack resistance analysis** checkbox defines whether the column should be analyzed for the second limit state, and the **Static indeterminacy** checkbox defines whether the column belongs to a statically determinate or indeterminate structure.

The effective length factors and the random eccentricities are specified following the same rules as in the **Resistance of Ferroconcrete Sections** mode (see Section 4.1).

Loads

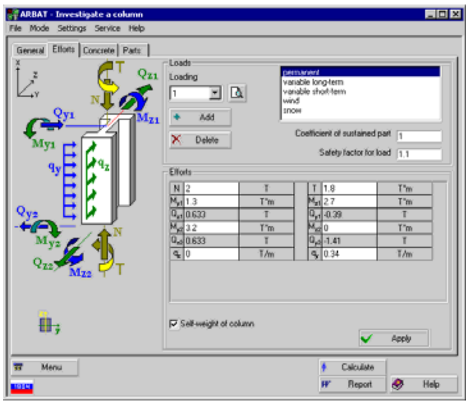


Fig. 49. *The* **Loads** *tab*

The **Loads** tab (Fig. 49) is used to specify design values of loads applied as a longitudinal force and nodal moments in the extreme sections of the column; these are to describe an interaction between the column and the rest of the structure. Another admissible type of load is a uniformly distributed lateral force or torque upon the column. Note that a positive longitudinal force corresponds to *compression* in this mode. Depending on whether you are dealing with a uniaxial or biaxial state of stress, you can specify the whole set of loads or only ones that conform to a uniaxial eccentricity.

Follow these steps to define loads for each loading:

- click the **Create** button (which means to create a loading); this will add No. of the loading to the respective list;
- choose a type of the current loading from the list of loading types (permanent, temporary but sustained, short-term, wind, snow). The type of loading defines the combination factors in compliance with SNiP 2.01.07-85*, which stand before the loads of this loading when DSCs are calculated;
- enter values that correspond to the current loading in the appropriate edit fields;

- in the **Sustained-part factor** edit field, enter a value for this factor to be used with the current loading;
- in the **Safety factor for load**, specify a value for this factor;
- click the **Apply** button.

After performing the latter operation, nodal forces Q_1 and Q_2 which ensure the equilibrium will be calculated automatically.

The **Self-weigth** checkbox helps automatically add a load caused by the column's self-weight to the current loading.

The  button lets you see the distribution curves for the current loading upon the column (Fig. 50).

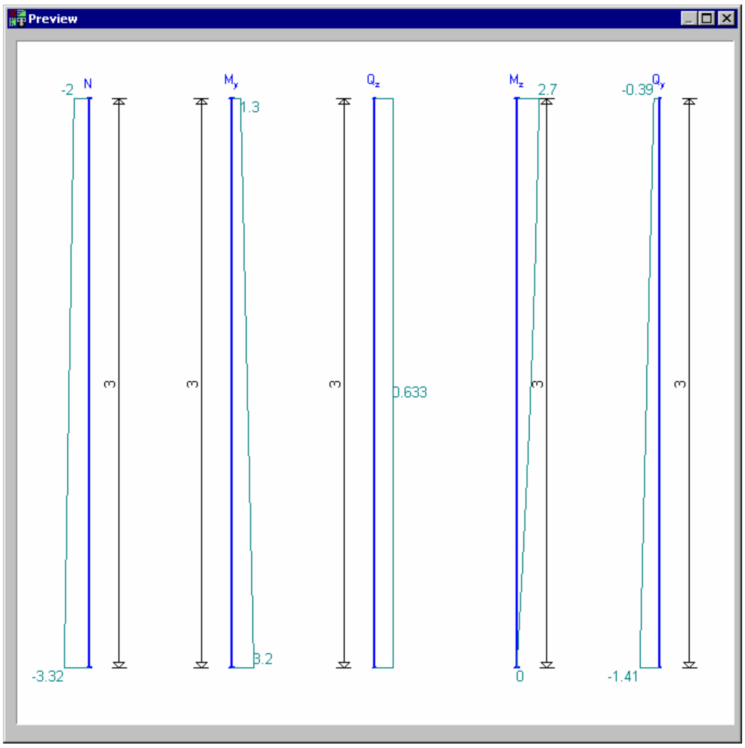


Fig. 50. *Curves of stresses in a column*

To delete the current loading, click the **Delete** button.

To edit a previously added loading (the editing mode allows only new loads to be added or the whole loading to be deleted), choose its No. from the drop-down list of Nos. of loadings. Having added new loads to the loading, click the **Apply** button.

Concrete

Properties of concrete are defined on the **Concrete** tab following the rules described in Section 3.2.

Segments of column

Segments of the column are defined on the **Segments** tab following the same rules as in the **Appraisal of Beam** mode (see Section 4.3). The only difference is that there is no list of spans, and the data are specified for all segments together, their number being selected from the **Number of segments** list. The numbering of segments in a column goes from the bottom to the top.

Drafting a column

The application has a mode in which it creates automatically a set or working drawings for a column (the **Draft column** button ). The drawings contain a specification, a bill of steel, a bill of parts, and a drawing of the reinforcement cage (if the cages are used) with its specification.

The following peculiarities and limitations should be taken into account for the column drafting process:

1. The drafting is available only for columns of a rectangular or annular section.
2. Longitudinal reinforcement in a column is always symmetric, i.e. $S_1=S_2$, and constant over its height. To provide the source data, you need to specify only the diameter and the spacing of reinforcement S_1, and S_2 will be taken care of automatically.
3. Reinforcement S_1 and S_2 can have only one row (if second row is specified, it will be ignored).
4. One row (S_1 and S_2) may not have less than two rebars.
5. A diameter and spacing must be specified for transverse reinforcement.

Note: As the transverse reinforcement is the same over the length of the column, it can be defined on the first segment only; it suffices to specify the diameter and the spacing of the transverse reinforcement for all the other segments.

After the drafting mode is launched, it checks the available source data, and if there are no errors the **Reinforcement Type** dialog box is called up where you use controls available there to choose a method of reinforcement: either with rebars or with cages. Making your choice and clicking **OK** opens the **Column Draft** dialog box (Fig. 51) which contains a toolbar of control buttons and a drawing field.

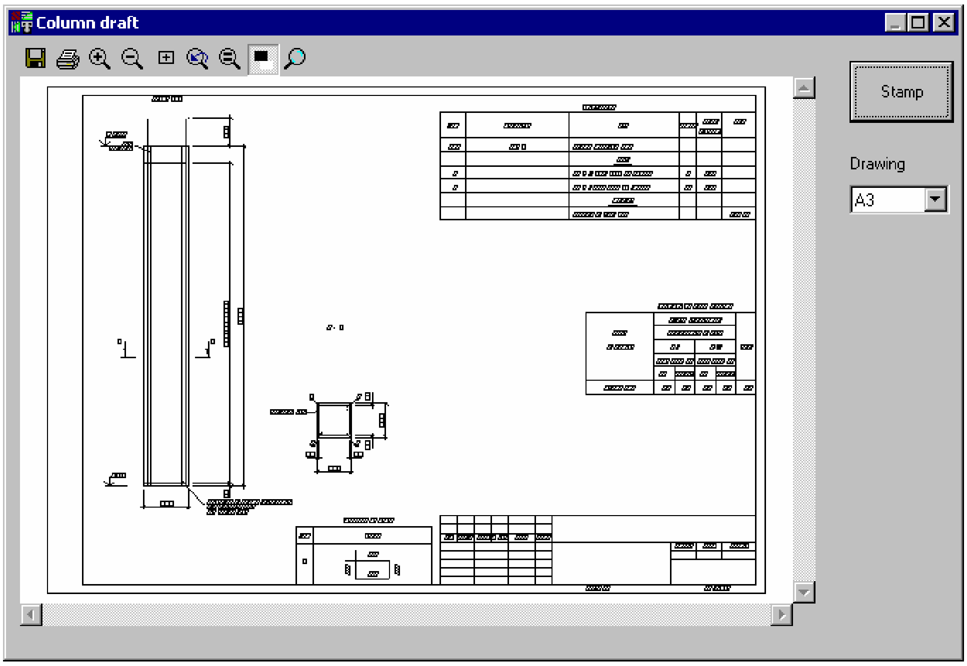
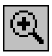
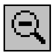
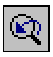
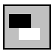


Fig. 51. *The* **Column Draft** *dialog box*

The toolbar contains buttons to perform the following actions:

-  — **Original view**,  — **Zoom in**, and  — **Zoom out** — these three are used to zoom in or out when viewing the document. When the picture is zoomed in, scrollbars appear on the edges of the display field to help you navigate across the document. If your mouse has a wheel, it can be rolled to duplicate the actions *Zoom in* (roll forward) and *Zoom out* (roll backward).
-  — **Return to previous scale** — set the scale which was used before invoking any of the above commands;

-  — **Invert image** — alter the view of the drawing (turn black-and-white into white-and-black and vice versa);
-  — **Magnifier** — clicking this button will open a pane (at the bottom of the drawing) where a magnified part of the drawing in the vicinity of the mouse pointer will be displayed (Fig. 52);

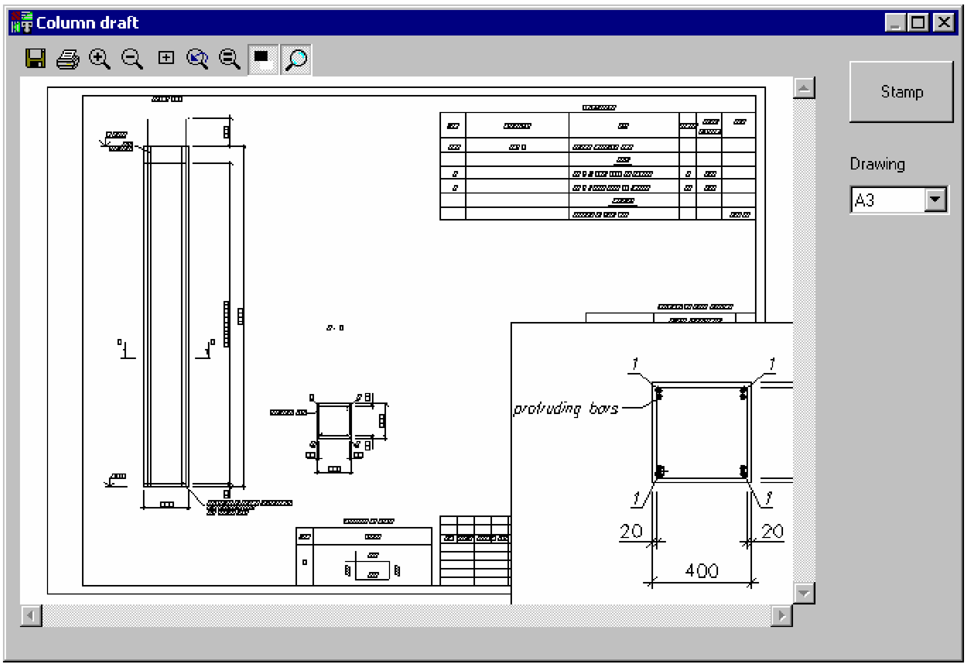
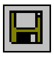


Fig. 52. *The* **Draft Column** *dialog box* — using the magnifier

-  — **Print** — sends the document to a printing device (to make a hard copy). A standard dialog box, **Print**, opens up where you should choose a device you want to send the document to and change its properties if necessary;
-  — **Save** — if you want to polish your document before making a hard copy of it, you can generate a DXF file, which is one of the AutoCAD file formats. Clicking this button will open a standard dialog box, **Save As**, where you specify a name for the drawing, a folder to save it in, and a format (either DWG or DXF);

The **Title block** button opens a dialog box under the same name where you fill in the fields of the title block, form 3, according to GOST 21.101-97. (When the drawing of the reinforcement cage is being generated, you are suggested to fill in the title block, form 4).

Output documents can be generated in the A3 or A4 format. The cage drawing is always output in the A4 format. Choose the desired format in the **Drawing** drop-down list that includes the following items:

- *Column (A3 format)* — the drawing of the column with its sections, a specification, a bill of steel, and a bill of parts is arranged on a single sheet of the A3 format;
- *Column, sheet 1 (A4 format)* — the A4 format is used; the first sheet contains the drawing of the column with its sections, the bill of parts, and the title block;
- *Column, sheet 2 (A4 format)* — the A4 format is used; the second sheet contains the specification, the bill of steel, and the title block;
- *Cage (A4 format)* — if reinforcement cages are used, the sheet will contain a drawing of the cage on a single A4 sheet, with the specification and the title block. If the reinforcement consists of separate rebars, this option will not be available.

Crack resistance

When SNiP 2.03.01-84* is used, the state of the **Crack resistance analysis** checkbox on the **General** tab defines whether a check of the column's reinforcement for crack resistance is needed. If the analysis is based on SP 52-101-03, this checkbox is always on.

Crack resistance data are specified on the **Crack Resistance** tab following the steps described in Section 3.4.

Appraisal results

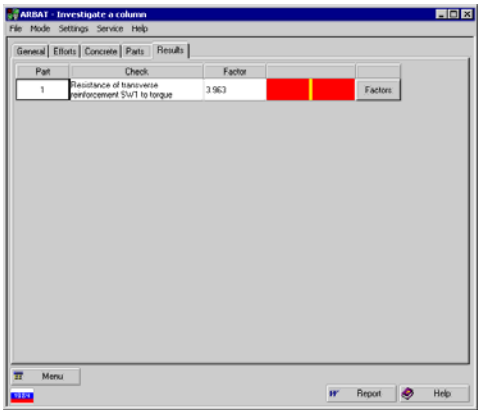


Fig. 53. *The* **Appraisal Results** *tab*

This tab (Fig. 53) opens automatically as soon as the appraisal mode is activated (with the **Calculate** button).

Results of the analysis are displayed as a table. The first column of the table lists Nos. of segments. For each segment, the **Check** column displays the name of the criterion that produces the greatest value of the load-bearing ability utilization factor, and the **Factor** column displays the value itself. The last column of the table presents a graphical representation of the factors where red bars denote values greater than one.

A report can be generated after the results of the analysis (the **Report** button).

Similarly to the **Appraisal of Beam** mode (see Section 4.6), the **Criteria** buttons in the table let you have a detailed information about all checks performed for each segment of the column.

4.8 Appraisal of a slab

This mode is used to perform an appraisal of a given structural scheme of a rectangular field of a solid slab. We can distinguish between slabs that can be bent in one direction and those that can be bent in two directions, depending on the ratio of the lengths of their sides.

The field of a slab can be either a whole structural member in a structure or a building (like a floor panel over a rectangular hole) or a part of a ribbed slab. The load-bearing ability of the slab is determined from the limit equilibrium conditions according to a procedure defined in the Guide to SNiP 2.08.01-85 [17] and the Instruction Manual for analysis of statically indeterminate ferroconcrete constructions, with redistribution of stresses taken into account [18].

To perform the appraisal of a slab, a limit value of the uniformly distributed load is compared with an overall load caused by the specified loadings.

The application performs the check of:

- the load-bearing ability of the slab with a bending moment caused by the overal uniformly distributed load as a criterion, counting or not counting the load-bearing ability of the anchors;
- the load-bearing ability of the slab with a shear force caused by the overal uniformly distributed load as a criterion;
- the load-bearing ability of the slab with cracks in the slab's span and along the lines of support as a criterion;
- the maximum crack opening width in the span and in the supported sections of the slab;
- the maximum deflection of the slab.

Features of the current version

- For slabs that can be bent in one direction only, the application implements regulations of the Guide to SNiP SNiP 2.08.01-85 [17] which allow the limits of the bending moments in the span and on the supports to be increased by 20%;
- For slabs that can be bent in two directions, the application implements the regulation of Instruction Manual [18] which

allows the limit of the uniformly distributed load to be increased by 10%;
- only sections in the vicinity of supports are checked for resistance to the shear force;
- the check of formation of cracks and their opening width is performed only for sections normal to the longitudinal axis of the slab.

Support conditions for the slabs:

(a) for slabs that can be bent in one direction, the support conditions are defined on two sides only. At least one side should be clamped. The other side of the slab can be clamped, simply supported, or free from support. This way of combining the possible support conditions permits to simulate extreme or middle spans of continuous “beam-like” slabs, where the second and the next spans of the continuous slab are not distinguished between one another. In all cases, the distribution of the stresses is assumed to be the same as that in the second span from the slab’s edge. These support conditions permit also to analyze the slab as a separate structural member in all practically important cases:

(b) for slabs that can be bent in two directions, the sides can be either clamped or simply supported. One of the shorter sides of the slab can be free from supports.

In all cases, the loads are assumed to be uniformly distributed over the area of the slab.

The slabs are checked for strength and crack resistance to comply with SNiP 2.03.01-84* (SP 52-101-03).

Source data are prepared on the **General**, **Loads**, **Concrete**, and **Crack Resistance tabs**, and results are reviewed on the **Appraisal Results** tab.

General

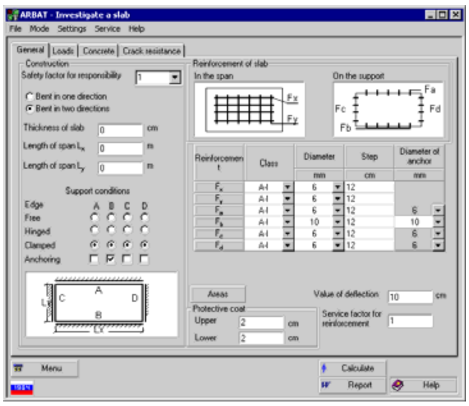


Fig. 54. *The* **General** *tab*

This tab (Fig. 54) is used to enter data that describe a structural scheme of the slab, the diameter, class, and spacing of rebars, and the diameter of anchor bars (for slabs in two-direction bending), the thickness of bottom and top covers, service factors for reinforcement, and the allowable deflection limit. Anchor bars can be installed on the clamped sides of the slab only. The reinforcement class for the anchor bars is assumed to be the same as that for the working reinforcement. The anchors are made of single rebars installed in one row.

To select a structural scheme for the slab, use an appropriate checkbox to indicate its type of behavior (bending in one or two directions), specify the length of the slab, and its sizes along axes X (L_x) and Y (L_y). If the ratio of the lengths of the sides is three or less (that is, $\frac{L_x}{L_y} \le 3$ or $\frac{L_y}{L_x} \le 3$), the slab should be treated as one capable of bending in two directions.

Support conditions for every face of the slab are assigned by selecting appropriate radio buttons: **Free** (edge), **Simply Supported** (edge), or **Clamped** (edge), which indicate the way the slab is supported by its lower structures or the way interior areas of the slab are supported by ribs. For clamped faces, the **Anchoring** group of checkboxes is used to indicate the presence of anchors.

Active columns and rows in the slab reinforcement table depend on a given design decision.

The **Areas** button provides reference information about the areas of reinforcement per meter of length of the slab.

Loads

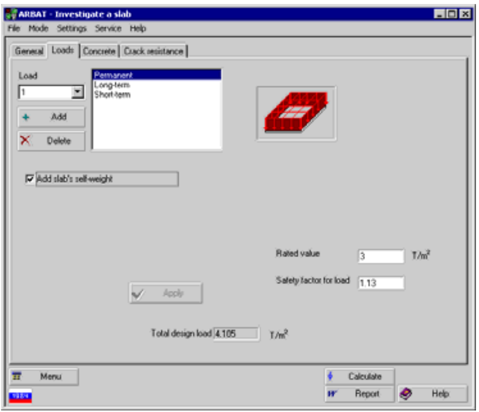


Fig. 55. *The* **Loads** *tab*

The slab is checked for action of only distributed loads over its whole area. It is the user who is responsible for making the uniformly distributed loads out of real ones. This tab (Fig. 55) is used to specify uniformly distributed loads. The analysis involves only one combination of loads. The loads are specified in the same way as described in Section 4.3.

The tab contains a checkbox, **Add self-weight of slab**. If the checkbox is turned on, the application will add the self-weight load automatically to the list of loads.

Concrete

Properties of concrete are specified on the **Concrete** tab according to the rules defined in Section 3.2.

Limitations of the current version

- the grade of concrete cannot be lower than V12,5;
- fine-grained concrete can be of either group A or B only;
- a natural humidity is assumed for concrete. Water saturation, or alternate water saturation and drying, are not taken into account.

Crack resistance

Crack resistance data are specified on the **Crack resistance** tab according to the rules defined in Section 3.4.

Appraisal results

The result of the appraisal is a value of K_{max} (a maximum value of the given criterion) which is displayed at the bottom of the dialog box. A complete list of all criteria (checks) can be obtained by clicking the **Criteria** button. This will open a standard dialog (see Section 1.1) containing a list of all checks.

A report can be generated after the results of the analysis (using the **Report** button).

5. Local strength

All modes of this group perform an appraisal (verification) of ferroconcrete structural members, including fixings in concrete, under a local action of loads in compliance with SNiP 2.03.01-84* [5] and taking into account requirements and recommendations of the Guide to SNiP 2.03.01-84* [7] (the Guide to SP 52-101-03), Manual of engineering of concrete and ferroconcrete constructions made of heavy-weight concrete [15], another Manual of engineering of concrete and ferroconcrete constructions made of heavy-weight concrete [16], Recommendations on engineering of steel fixings in ferroconcrete constructions [14].

The structural scheme is assumed to be known, in the sense that we know the load application area, methods of junction between members etc., and we also know what additional transverse reinforcement is needed to ensure the local strength.

Working with these modes is in no way different from working with the appraisal modes.

5.1 Local compression (SNiP 2.03.01-84)*

This mode implements checks of ferroconcrete structural members for local compression in compliance with Sec. 3.39 through 3.41 of SNiP 2.03.01-84* [5]. All structural schemes of ferroconcrete members presented in Drawing 15 of SNiP 2.03.01-84* are implemented.

The appraisal procedures include checks for a longitudinal force of members both with an additional lateral reinforcement by transverse mats and without the lateral reinforcement.

Depending on the presence of the lateral reinforcement, one of the following criteria is analyzed:

- strength based on local compression conditions for a non-reinforced cross-section;
- strength based on local compression conditions for a cross-section reinforced by mats.

General

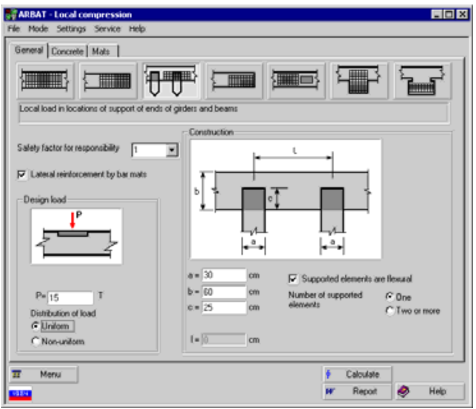


Fig. 56. *The* **General** *tab* (SNiP 2.03.01-84*)

This tab (Fig. 56) is used to specify:

- a scheme for the local compression analysis and sizes of the load application area (shown as a darkened rectangle in the schematic);
- a design load with its distribution over its application area indicated (whether uniform or non-uniform);
- the presence of lateral reinforcement.

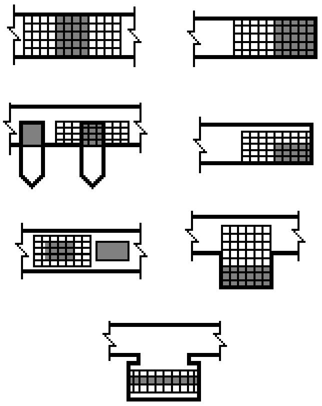


Fig. 57. *Loading schemes* (*as per* SNiP 2.03.01-84*)

The scheme of loading (Fig. 57) is selected by pointing at an appropriate icon; it conforms to one of schemes presented in Drawing 15 of SNiP 2.03.01-84*. Modifications of the loading schemes defined by Drawing 15 are described in the **Features of the current version** section.

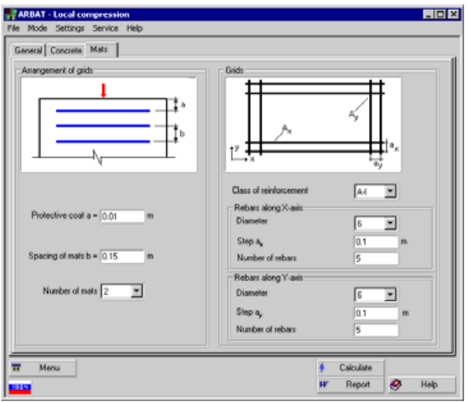


Fig. 58. *The* **Mats** *tab*

When the **Lateral reinforcement by bar mats** checkbox is on, the **Mats** tab becomes accessible for entering source data (Fig. 58).

This tab is used to define:

- the cover of concrete;
- the vertical spacing of the mats;
- the number of the mats;
- the class of reinforcement, which is assumed to be the same for rebars in both directions;
- the diameter, the spacing, and the number of rebars parallel to the X-axis;
- the diameter, the spacing, and the number of rebars parallel to the Y-axis.

Concrete

Properties of concrete are specified on the **Concrete** tab in the same way as defined in Section 3.2.

Features of the current version

Loading schemes (*c*) and (*d*) of Drawing 15 of SNiP 2.03.01-84* are merged into one image.

Loading scheme (*f*) of Drawing 15 of SNiP 2.03.01-84* has been modified. The application suggests the capability of specifying multiple application points for the same loads (see Fig. 57) similarly to schemes (*c*) and (*d*). The method of determining the local area of bearing is the same as in schemes (*c*) and (*d*).

The vertical spacing of the mats and the number of the mats are verified using formulas (198) and (199) from Section 3.94 of the Guide to SNiP 2.03.01-84* [7].

Limitations of the current version

The limitations of the current version depend on the following circumstances:

- whether it is legitimate to do the check for local compression (it depends on the ratio between the lateral sizes of the member of interest and the sizes of the load application area);
- how the lateral reinforcement by flat mats is done;
- constructional limitations of the classes and diameters of rebars used, the spacing between rebars in mats and between the mats, and other limitations defined in documents [7], [15], [16].

Limitations related to the legitimacy of a check are terminal: once they are violated, the respective analysis is prohibited.

Limitations related to the constructional requirements are not terminal, and they allow to perform the analysis, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of the limitations are implemented “rigidly”: you are permitted to choose certain values from a limited list only — for example, the number of mats, the reinforcement class etc. Other kinds of limitations do no more than produce warnings in the reports and on the screen.

These are constructional limitations implemented in the current version of the software:

- the number of lateral reinforcement mats is 2 to 4;
- the concrete cover to reinforcement is 10 to 20 mm;
- the vertical distance between the mats (the mat spacing) is 60 to 150 mm;
- the reinforcement classes are Vr-I, A-I, A-II, A-III as per SNiP 2.03.01-84* and A400S as per TSN 102-00 [5];
- the diameters of rebars, depending on their class, are 3 to 14 mm;
- the distance between the bars of the mats (the bar spacing) in either direction is 50 to 100 mm.

5.2 Local compression (SP 52-101-03)

This mode implements checks of ferroconcrete structural members for local compression according to requirements of Sec. 6.2.43 through 6.2.45 of SP 52-101-03. All designs of ferroconcrete members shown in Fig. 6.11 of SP 52-101-03 are implemented.

Both members with additional lateral reinforcement by transverse mats and members without such are checked for the action of a longitudinal force. Depending on the presence of lateral reinforcement, one of the following criteria is investigated:

- strength based on local compression conditions for a non-reinforced section;
- strength based on local compression conditions for a section reinforced by mats.

General

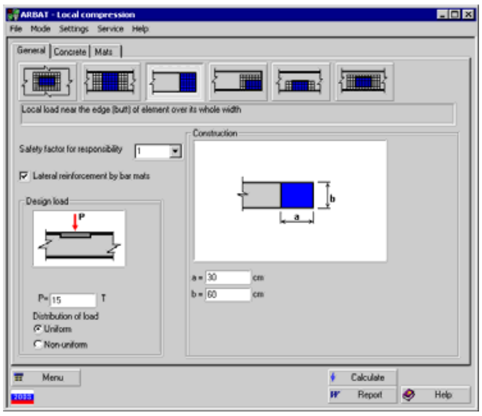


Fig. 59. *The* **General** *tab*

This tab (Fig. 59) is used to choose the following:

- a loading scheme for the local compression analysis, and sizes of the load application area (it is shown on the schematic as a darkened rectangle);
- a design load and the way it is distributed over the area of application (uniformly or non-uniformly);
- the presence of lateral reinforcement.

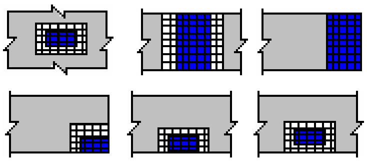


Fig. 60. Loading schemes as per SP 52-101-03

The scheme of loading (Fig. 60) is selected by pointing at an appropriate icon; it conforms to one of schemes presented in Fig. 6.11 of SP 52-101-03.

If the **Lateral reinforcement by flat mats** checkbox is enabled, the **Mats** tab becomes accessible for entering source data (Fig. 58); its contents are similar to those of the respective tab in the analysis complying with SNiP 2.03.01-84*.

Concrete

Properties of concrete are defined on the **Concrete** tab according to rules described in Section 3.2.

Features of the current version

The vertical spacing between the mats and the number of the mats is checked using formulas (198) and (199) of Section 3.94 of the Guide to SNiP 2.03.01-84* [7].

Limitations of the current version

The limitations of the current version depend on the following circumstances:

- whether it is legitimate to do the check for local compression (it depends on the ratio between the lateral sizes of the member of interest and the sizes of the load application area);
- how the lateral reinforcement by flat mats is done;
- constructional limitations of the classes and diameters of rebars used, the spacing between rebars in mats and between the mats, and other limitations defined in documents [7], [15], [16].

Limitations related to the legitimacy of a check are terminal: once they are violated, the respective analysis is prohibited.

Limitations related to the constructional requirements are not terminal, and they allow to perform the analysis, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of the limitations are implemented "rigidly": you are permitted to choose certain values from a limited list only — for example, the number of mats, the reinforcement class etc. Other kinds of limitations do no more than produce warnings in the reports and on the screen.

These are constructional limitations implemented in the current version of the software:

- the number of lateral reinforcement mats is 2 to 4;
- the concrete cover to reinforcement is 10 to 20 mm;
- the vertical distance between the mats (the mat spacing) is 60 to 150 mm;
- the reinforcement classes are A240, A300, A400, A500, V500 according to [6];
- the diameters of rebars, depending on their class, are 10 to 14 mm;
- the distance between the bars of the mats (the bar spacing) in either direction is 50 to 100 mm.

5.2 Punching (SNiP 2.03.01-84*)

This mode implements a check of slab-like structures for punching (there is no transverse reinforcement) in compliance with Sec. 3.42 of SNiP 2.03.01-84*and Sec. 3.98 of the Manual of engineering of heavy-weight concrete [16]. The subject of consideration includes both constructions with an additional lateral reinforcement of vertical bars located within the punching pyramid and constructions without any lateral reinforcement.

Depending on the presence of lateral reinforcement, one of the following criteria is analyzed:

- strength based on punching conditions, without additional reinforcement;
- strength based on punching conditions, taking additional reinforcement into account.

General

The **General** tab (Fig. 61) is used to specify:

- the sizes of the slab's base;
- the sizes of the load application area;
- the height of the slab (*H*), or the working height of the section (*h*) where:
- *h* is the distance from the slab's top to the center of gravity of bottom reinforcement (selection is made by enabling an appropriate radio button). If the slab height parameter *H* is used, the value of *h* is taken equal to 0.9*H*;
- the resultant of the punching load brought to the center of the load application area;
- a uniform load (in particular, equal to zero) applied to the base of the slab, which resists to punching.

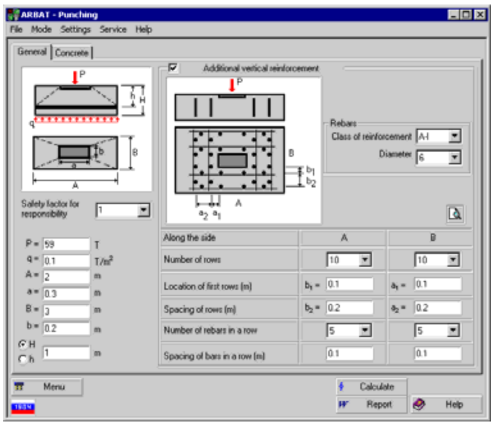
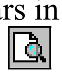


Fig. 61. *The* **General** *tab*

When the **Additional vertical reinforcement** checkbox is enabled, the following data need to be specified:

- the class and the diameter of vertical rebars that make up the mats and are parallel to the sides of the slab;
- the number of rows of vertical bars parallel to the sides of the slab;
- the position of first rows with respect to the load application area;
- the number of rows of vertical rebars along each side of the base;
- the spacing (the distance between the rebars in a row);
- the number of rebars in rows.

Clicking the button  displays a picture of the arrangement of rebars according to the specified parameters of the additional vertical reinforcement.

Concrete

Properties of concrete are specified on the **Concrete** tab following the steps defined in Section 3.2.

Features of the current version

The requirement of Sec. 3.42 of SNiP 2.03.01-84* that the punching force should act on a “limited area” is implemented in the following way: the area is recognized as “limited” if the inclination of the side faces of the punching pyramid to the horizon does not exceed 68°. This limitation complies with fifth paragraph of Sec. 3.42 of SNiP 2.03.01-84*.

The punching is analyzed for slab-like constructions only. Steps in the slabs (such as foundation jumps) are not taken into consideration.

The vertical punching load and the load applied to the base of the slab is assumed to spread uniformly over their aplication areas.

There is supposed to be no eccentricity in the application of the loads.

The next-to-last paragraph of Sec. 3.42 of SNiP 2.03.01-84* is not implemented.

Additional lateral reinforcement is implemented only as vertical mats with working vertical rebars. The mats are parallel to the sides of the slab, are symmetric, and do not fall within the load application area. The location of mats nearest to the load application area is such that they always fall within the punching prism.

The requirement of Sec. 5.29 of SNiP 2.03.01-84* is implemented; it concerns the installation of additional reinforcement against possible punching.

Limitations of the current version

The limitations of the current version depend on the following circumstances:

- whether it is legitimate to do the check for punching, which depends on the ratio between the lateral sizes of the slab and the sizes of the load application area;
- the ratio of the punching load and the pressure under the base of the slab;
- constructional limitations of the used classes and diameters of rebars, of the spacing between rebars in the mats and between the mats, other limitations stated in documents [7], [15], [16].

Limitations related to the legitimacy of checks and to the ratios of loads are terminal: once they are violated, the analysis is prohibited.

Limitations related to the constructional requirements are not terminal, and they allow to perform the analysis, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of the limitations are implemented "rigidly": you are permitted to choose certain values from a limited list only — for example, the number of rebar rows, the reinforcement class etc. Other kinds of limitations do no more than produce warnings in the reports and on the screen.

Below are constructional limitations implemented in the current version of the software:

- the allowed reinforcement classes are Vr-I, A-I, A-II, A-III as per SNiP 2.03.01-84*, and A400S as per [6];
- the allowed diameters of rebars depending on their class are 3 to 14 mm;
- the minimum spacing between rebar rows is defined by Sec. 5.29 of SNiP 2.03.01-84*;
- the distance of first row of rebars from the load application area may not be less than 30 mm or greater than the spacing between the rows;
- the total number of rebar rows along any of the sides is 2 to 10;
- the number of rebars in each row must be the same in any row along the same side; it may not be less than 2 or greater than 10.

The appraisal check analyzes the number of rows and the number of rebars in the rows which fall within the punching prism. In any of the rebar rows or particular rebars in the rows are out of the punching prism, a message is generated for the report and displayed on the screen.

5.3 Punching (SP 52-101-03)

This mode implements a check of slab-like structures for punching in compliance with Sec. 6.2.46-6.2.52 SP 52-101-03 [9]. The subject of consideration includes both constructions without any lateral reinforcement and constructions with an additional lateral reinforcement of vertical bars arranged in the cross-section of a slab's fragment uniformly or in a crosswise manner. In the general case, the analysis involves a combined action of a longitudinal compressive force and a bending moment.

All checks defined by Sec. 6.2.46-6.2.52 of SP 52-101-03 are implemented.

General

This tab (Fig. 62) is used to specify:

- the footprint sizes of the slab;
- the sizes of the load application area and, if necessary, its binding to the slab's edges;
- the height of the slab (*H*), or working heights of the sections for longitudinal reinforcement along the X and Y axes (to make a choice, use an appropriate radio button). If the source data consists of the slab's height, *H*, then the working heights of the sections are averagely equal to 0.9H;
- the resultants of the punching load (the longitudinal force and the bending moments) brought to the center of the load application area;
- the location of the load application area — near a free edge of the slab (Fig. 63) or in its middle part (Fig. 64).

If the **Reinforcement** checkbox is enabled, you specify the arrangement of reinforcement — either uniform (Fig. 65-66) or crosswise (Fig. 67-68);

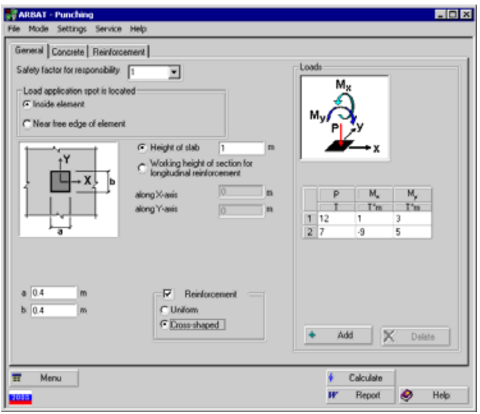


Fig. 62. *The* **General** *tab*

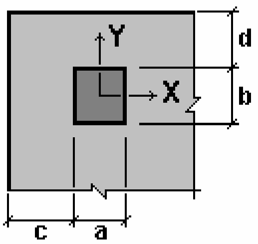


Fig. 63. *A load application area near a free edge of a slab*

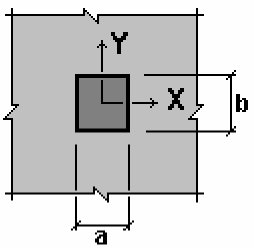


Fig. 64. *A load application area in the middle part of a slab*

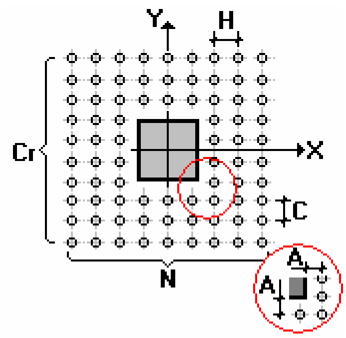


Fig. 65. *Reinforcement arranged uniformly*

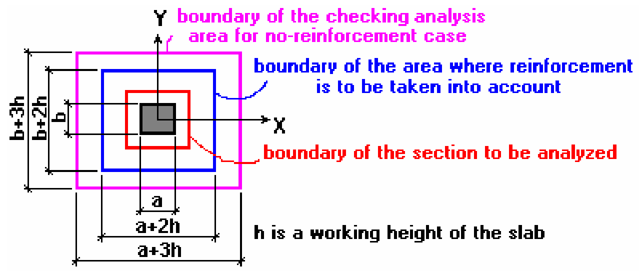


Fig. 66. *Zones on the slab for the checking analysis, reinforcement arranged uniformly*

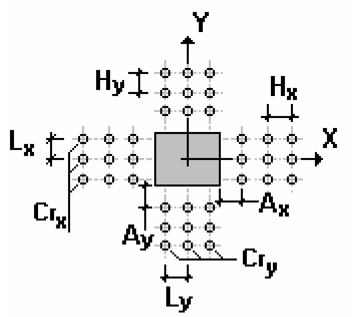


Fig. 67. *Crosswise arrangement of reinforcement*

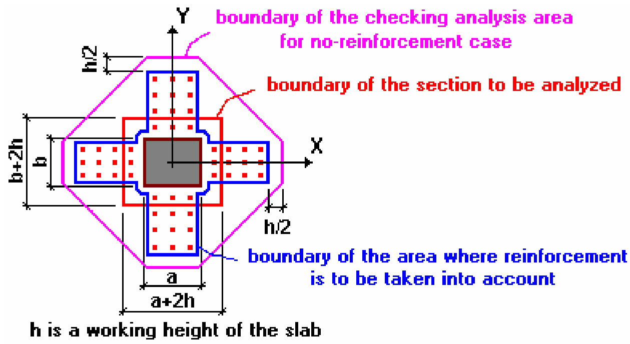


Fig. 68. *Zones on the slab for the checking analysis, crosswise arrangement of reinforcement*

Concrete

Properties of concrete are specified on the **Concrete** tab following the steps defined in Section 3.2.

Reinforcement

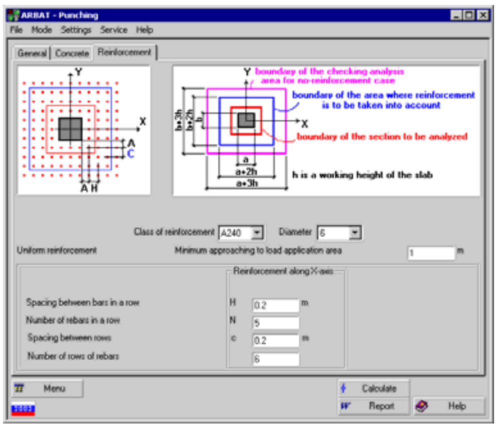


Fig. 69. *The* **Reinforcement** *tab*

The **Reinforcement** tab (Fig. 69) is used to specify:

- the class and the diameter for vertical rebars that make up mats parallel to the sides of the slab;
- the minimum distance from the reinforcement to the load application area;
- the number of rows of vertical rebars parallel to the sides of the slab;
- the spacing (the distance between rebars in a row);
- the spacing between the rebar rows.

Features of the current version

The punching is implemented for slab-like constructions only.

An approach to the free edges of a slab (for various schemes of punching) is taken into account only when the distance from the load application area to the free edge exceeds 3 working heights (h_0) of longitudinal reinforcement in the slab along one of the axes, X, Y (free plate's edge), or both axis X, Y (free plate's corner). The punching analysis cannot be done when the free edge of the slab falls within an area which is nearer to the load application area than half the working height of the slab.

Lateral reinforcement is symmetric in both cases (uniform or crosswise) with respect to axes X, Y.

Transverse reinforcement is taken into account by the analysis within the full pyramid of punching, with an offset of h_0 from the load application area to either side.

Limitations of the current version

Here follow the limitations of the current version of the software imposed by design codes:

- the punching analysis cannot be done when the ratio of the bending moments used in the analysis to the ultimate bending moments exceeds the ratio of the longitudinal forces used in the analysis to the ultimate ones that concrete can resist (see Sec. 6.2.46 of SP 52-101-03);
- lateral vertical reinforcement is implemented only in the case when the load application area is located in the middle of the slab;
- if the load application area is located near the edge of the slab, the requirement of the last paragraph of Sec. 6.2.49 of SP 52-101-03 is to be complied with;
- in the implementation of the last paragraph of Sec. 6.2.52 of SP 52-101-03, the moments of section of transverse reinforcement are equal to those of concrete;
- there are other constructional limitations of classes and diameters of rebars that can be used, of the spacing between rebars in mats and between mats, and other limitations stated in documents [7], [15], [16].

An additional lateral reinforcement is implemented only as vertical mats with vertical working rebars. The bar mats are parallel to the sides of the slab and do not fall within the load application area. The location of the mats nearest to the load application area is such that they always fall within the punching prism.

Limitations related to the legitimacy of checks and to the ratios of loads are terminal: once they are violated, the respective analysis is prohibited.

Limitations related to the constructional requirements are not terminal, and they allow to perform the analysis, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of the limitations are implemented “rigidly”: you are permitted to choose certain values from a limited list only — for example, the number of rebar rows, the reinforcement class etc. Other kinds of limitations do no more than produce warnings in the reports and on the screen.

These are the constructional limitations which are used:

- the allowed reinforcement classes are A240, A300, A400, A500, V500 according to the document;
- the allowed rebar diameters, depending on the reinforcement class, are 6 to 40 mm;
- the arrangement of rebars complies with Sec. 8.3.15 of SP 52-101-03.

5.4 Tearing

This mode implements checks of joints of ferroconcrete structural members for tearing in compliance with Sec. 3.43 of SNiP 2.03.01-84*, Sec. 3.97 of the Guide to SNiP 2.03.01-84*, and Sec. 3.121 of the Manual of engineering of concrete and ferroconcrete constructions made of heavy-weight concrete [15].

Additional reinforcement is always present in the area where the joint to be checked is located.

The check analyzes the criterion of strength based on conditions of local tearing.

General

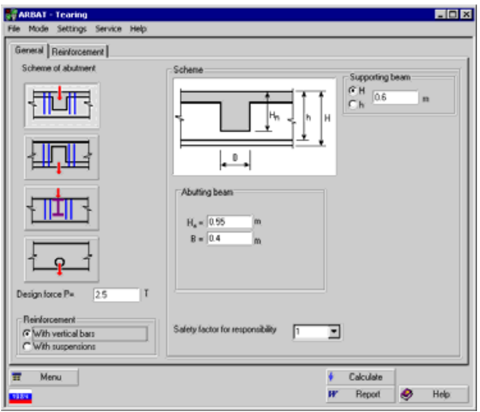


Fig. 70. *The* **General** *tab*

The **General** tab (Fig. 70) is used to select one of four schemes subject to the tearing check (Fig. 71).

First scheme is a monolithic ferroconcrete floor where beams join in the top area.

First scheme is a monolithic ferroconcrete floor where beams join in the bottom area.

Third scheme is an abutting steel beam (a double tee is shown in the figure). The abutting beam is defined by its height and its cross-section's width.

Fourth scheme is a concentrated force applied by hanging a weight through a hole in the beam.

To choose reinforcement for the area of tearing by either vertical bars or stirrups, use an appropriate radio button in the **Reinforcement** group.

Having selected a joint scheme, you need to specify:

- the design load;
- the sizes of the supporting and abutting beams (for schemes 1 through 3);
- the sizes of the supporting beam, the location and the sizes of the hole (for scheme 4).

For any of the schemes, you need to specify for the supporting beam its height (*H*), or the working height of the section (*h*) where *h* is the distance from the top of the beam to the center of gravity of bottom reinforcement (to choose, use an appropriate radio button). If the height parameter, *H*, is used, the value of *h* is taken as 0.9*H*;

Bottom reinforcement is assumed to be arranged in one row over the height of the beam.

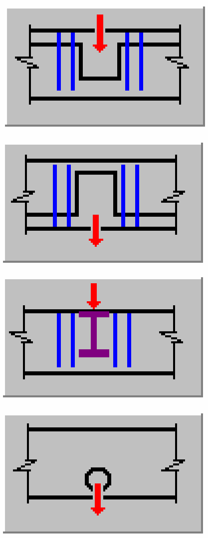
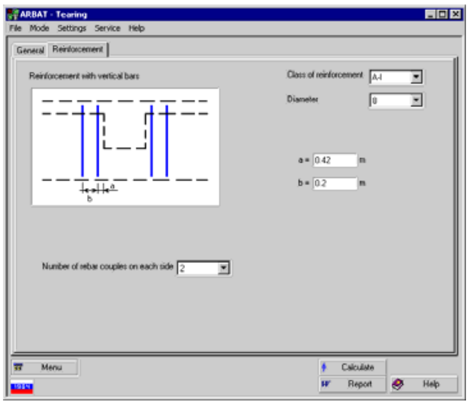
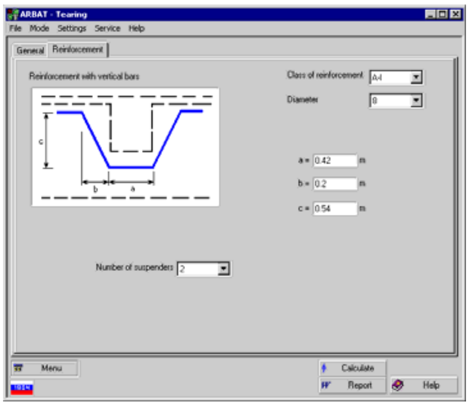


Fig. 71. *Schemes checked for tearing*

Reinforcement



(a)



(b)

Fig. 72. *The* **Reinforcement** *tab*

Depending on a reinforcement scheme, this tab is used to specify data about vertical rebars (Fig. 72, *a*) or stirrups (Fig. 72, *b*).

For the vertical bar reinforcement, the following should be specified:

- the class and the diameter of reinforcement;
- the number of rebar couples on each side of the abutting beam;
- the distance from the abutting beam to the first of the rebars;
- the distance between the rebars.

For the stirrup reinforcement, the following should be specified:

- the class and the diameter of reinforcement;
- the number of stirrups;
- the sizes of the stirrups (in the check of fourth scheme, the size of the horizontal part of the stirrup is not required).

Features of the current version

There is an important issue, how to transfer the load from the abutting beam to the supporting one. The following approach is used:

- in the check of schemes 1, 2 (joints between ferroconcrete beams), the load is transferred at the level of the center of gravity of the compressed concrete area in the abutting beam. The height of the compressed area of concrete is assumed by default to be equal to 40% of the abutting beam's height (it is possible to change the height of the compressed area);
- in the check of scheme 3 (a steel beam abutting), the load is transferred uniformly over the height of the abutting beam;
- in the check of scheme 4 (suspension of a weight through a hole in the beam), the load is transferred at the level of the hole's bottom, and the width of the load transfer area is assumed to be equal to half of the hole's diameter.

Additional reinforcement (vertical rebars, stirrups) is installed symmetrically with respect to the joint.

The last paragraph of Sec. 3.97 of the Guide to SNiP [7] is not implemented.

Limitations of the current version

The following reinforcement methods can be used with various structural schemes of the joints:

- reinforcement by vertical rebars can be used in schemes 1, 2, 3;
- reinforcement by stirrups can be used in schemes 1, 3, 4.

Limitations related to the design arrangement of a joint are terminal; once they are violated, the analysis is prohibited.

Limitations related to the constructional requirements are not terminal, and they allow to perform the analysis, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of the limitations are implemented "rigidly": you are permitted to choose certain values from a limited list only — for example, the number of rebars or the reinforcement class. Other kinds of limitations do no more than produce warnings in the reports and on the screen.

These are the constructional limitations that are used:

- the number of couples of rebars on each side in the reinforcement by vertical bars is one to three;
- if stirrups are used for reinforcement, then the number of those is 1 to 3;
- the distance between vertical bars is at least 50 mm;
- the classes of reinforcement are Vr-I, A-I, A-II, A-III as per SNiP 2.03.01-84* [5] and A400S as per TSN 102-00 [6]**;**
- the diameter of the rebars (depending on the class) is 3 to 14 mm;

The check determines the number of stirrups and bars that fall withint the area of tearing. If any of the bar do not fall within the area of tearing, a message is displayed on the screen and included in the report.

5.5 Fixings

This mode implements checks of anchors of welded steel fixings which consist of a flat plate and a normal or/and oblique anchor welded to it.

The exterior side of the plate of a fixing belongs to one plane with the exterior surface of its respective ferroconcrete member.

The checks comply with Sec. 3.44 through 3.46 of SNiP 2.03.01-84* and Recommendations on engineering of steel fixings [14].

Three types of anchors are subject to checking:

- normal anchors butt-welded to the plate;
- normal anchors butt-welded to the plate in combination with oblique anchors lap-welded to the plate;
- oblique anchors welded to the plate by submerged arc welding.

Both oblique and normal anchors can have reinforcing parts on their ends.

The strength of the fixing's plate is not subject to checking because there is no information about the design of a bearing part supported by it (a table, a rib etc.). The only criterion being checked is the compliance of the thickness of the plate and the diameters of the anchors with welding specifications.

The plate of a fixing is not specified to include additional retaining parts or devices that transfer some part of the lateral load onto concrete.

The following criteria are subject to checking for each of the types of anchors taken into consideration:

- the strength of a most stressed anchor;
- the immured length of an anchor in tension;
- bearing of concrete under the reinforcement of a most compressed anchor;
- spalling of concrete under an anchor in tension on the edge of a ferroconcrete member (in case when the distance from the edge of the fixing to the edge of the member is specified).

General

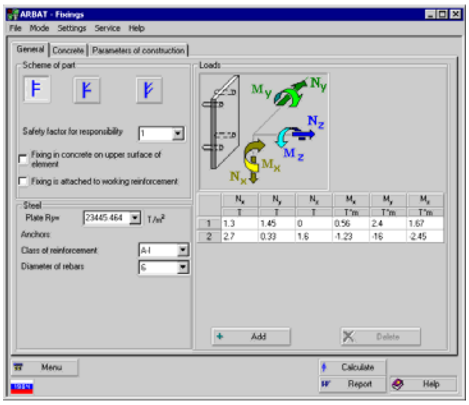


Fig. 73. *The* **General** *tab*

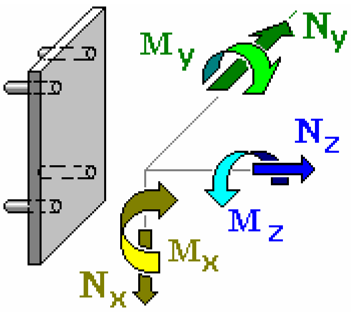


Fig. 74. *Loads upon fixings*

The **General** tab (Fig. 73) is used to specify:

- the type of a fixing;
- attachment of the anchors of the fixing to the working reinforcement of the member (if there is an attachment, only the strength of the anchors will be subject to checking);
- the class of steel for the plate of the fixing, the class and the diameter of the anchor's reinforcement (anchors of the same type are assumed to have the same diameter and the same class of reinforcement);
- properties of concrete (follow the same conventions as in the **Section Resistance** mode);
- loads on the fixing, brought to the center of the exterior surface of the plate.

A fixing of first type can transfer 6 force components to a ferroconcrete member: 3 forces along the coordinate axes and 3 moments with their vectors parallel to the respective axes. Positive directions of the force and moment vectors are shown in Fig. 74.

A fixing of second or third type can transfer only three force components belonging to the XoZ plane to a ferroconcrete member (Fig. 74).

Design parameters

This tab (Fig. 75–77) is used to specify the following for each type of fixings:

- distances from the edges of a fixing to the faces of the ferroconcrete member along the coordinate axes (in cases when the **General** tab indicates the presence of these limitations);
- reinforcing of the anchor bars;
- the thickness of the fixing plate;
- lengths of the anchors;
- the number and arrangement of anchors on the fixing;
- the angle of slope of the anchors (only for 2nd and 3rd type fixings).

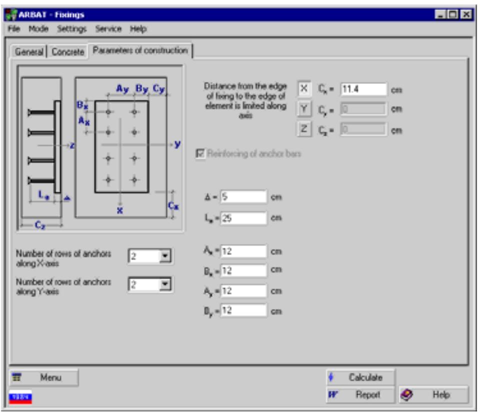


Fig. 75. *Design parameters for a 1st-type fixing*

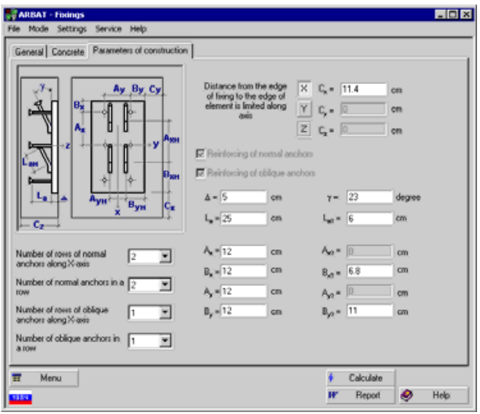


Fig. 76. *Design parameters for a 2nd-type fixing*

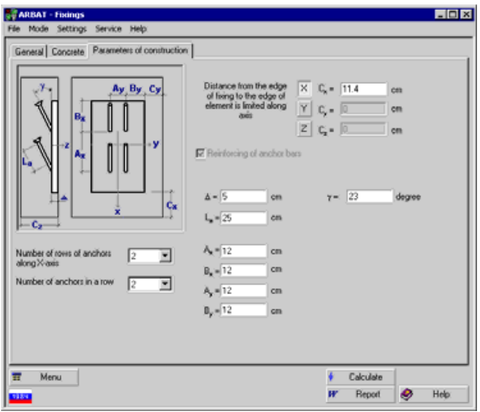


Fig. 77. *Design parameters for a 3rd-type fixing*

Limitations of the current version

The limitations of the current version are based on constructional limitations of the classes and diameters of rebars used for anchors, distances between the anchors, ratios of the plate thickness and the anchor bar diameters, the minimum number of anchors, and other limitations presented in SNiP 2.03.01-84* and Recommendations on engineering of steel fixings [14].

Limitations related to the legitimacy of checks are terminal: once they are violated, the analysis is prohibited.

Limitations related to the constructional requirements are not terminal, and they allow to perform the analysis, but the user gets a warning in the report and on the screen that the limitations have been violated. Some of the limitations are implemented “rigidly”: you are permitted to choose certain values from a limited list only — for example, the reinforcement class. Other kinds of limitations do no more than produce warnings in the reports and on the screen.

Here are the constructional limitations:

- the anchor bars can be made of reinforcement of Class A-I, A-II, A-III, A-IV, and A-400;
- the diameter of reinforcement can be 8 to 25 mm depending on the reinforcement class;
- limitations on the arrangement of reinforcement comply with Sec. 5.5 of the Recommendations [14];
- the number of rows of anchors bars can be 1 to 4 depending on the fixing type. The number of anchor bars in each of the rows is also 1 to 4;
- the plane of the bending moment (M_x, M_y) must have at least two normal or oblique anchors installed in it. To resist the torque (M_z), at least two normal anchors must be installed;
- the size of a reinforcing part at the end of the anchor is not particularized. The diameter of the reinforcing part is assumed, conditionally, to be equal to three diameters of the anchor. If the anchors are made of Class A-I reinforcement, then the reinforcing parts are always present.

Features of the current version

Special features of this implementation are based on assumptions and limitations for checks defined in SNiP 2.03.01-84* and the Recommendations [14]. They are as follows:

- when checking concrete for bearing under the anchor's reinforcement, the effective area of concrete is always assumed to be greater than the area of the anchor reinforcement. The φ_6 factor according to Sec. 3.39 of SNiP 2.03.01-84* is always 2.5;
- when checking concrete for bearing under the anchor's reinforcement, it is assumed that there can be a crack in concrete along the anchor;
- it is assumed in the check of the immuring length of a tensioned anchor that the whole anchor is in the tensioned area of concrete, to have a reserve of strength. Formula (62) of the Recommendations [14] assumes ω_{an} = 0,7 and φ_c = 1;
- there is no check of punching of concrete under anchors in tension.

5.6 Short cantilevers

This mode is used to check short cantilevers for the action of a lateral force, to ensure the strength of a compressed oblique strip between the load and the support in compliance with Sec. 3.34 of SNiP 2.03.01-84* and Sec. 3.99 of the Guide to SNiP [7].

The check is performed after all necessary conditions and requirements to the short cantilevers are fulfilled. The principal condition is $S_1 \leq 0{,}9h_0$ where $h_0=h-a_1$ is the working height of the cantilever's section at the support, S_1 is the distance from the interior face of the bottom part of the column to the end of the beam's area of bearing. Only one-sided cantilevers of three types are under consideration: a rectangular one, one with a slope to support a beam, and one with a slope to support a crane runway beam. To choose the cantilever type, click on the respective icon.

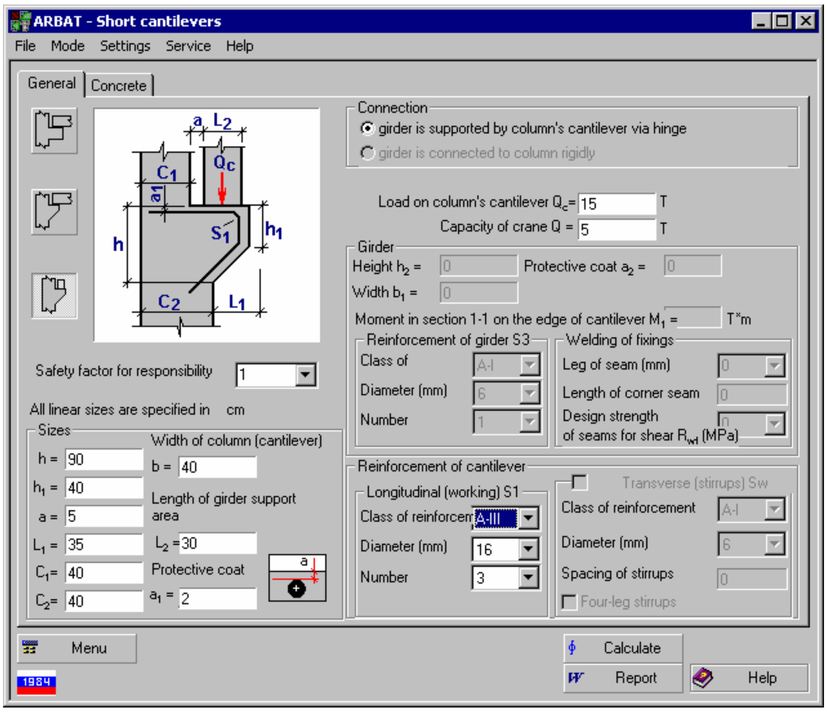


Fig. 78. *The* **General** *tab*

The **General** tab (Fig. 78) is used to specify:

- a joint type (Fig. 79):
  (a) the cantilever is a support for a crane runway beam;
  (b) the cantilever is a support for a beam;

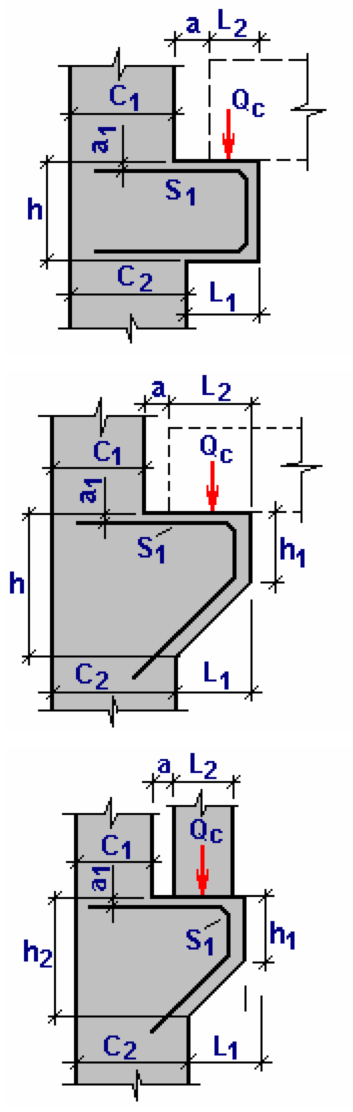


Fig. 79. *Short cantilever types*

- the sizes of the cantilever and the column: the height of the cantilever in the section at the support, **h**; the height of the free end of the cantilever, $\mathbf{h}_1$; the outreach of the cantilever, $\mathbf{L}_1$; the actual length of the beam's area of bearing along the outreach of the cantilever, $\mathbf{L}_2$; the distance from the interior face of the column to the side face of the beam, **a**; the height (of the section) of the top part of the column, $\mathbf{c}_1$; the height (of the section) of the bottom part of the column, $\mathbf{c}_2$; the width of the column or the cantilever, **b**; the distance from the top face of the cantilever to the center of gravity of the longitudinal reinforcement, $\mathbf{a}_1$;
- the method of joint between the beam and the column's cantilever;
- the capacity of the crane, **Q**, that works in the span between the columns, if the cantilever is a support for a crane runway beam;
- the load, $\mathbf{Q}_c$, on the cantilever caused by the imposed weight;
- the moment in the beam's section on the edge of the cantilever, in case the beam and the cantilever are joined rigidly;
- the class and the diameter of longitudinal (working) reinforcement;
- the class, the diameter, and the spacing of transverse reinforcement (stirrups).

The **Concrete** tab is generally similar to those described in the previous modes; the one here does not need hardening conditions to be specified but requires a service factor for concrete, $\gamma_{\beta 2}$.

Features of the current version

Admissible cantilevers can have haunches with any angle of slope. When a cantilever is 100 mm high or lower, or it has the outreach of $\mathbf{L_1}$ = 100…150 mm, it can be designed to be rectangular.

The overall dimensions of the cantilevers are set on the following basis: the height of the cantilever, h, in its section at the suppot may not be less than 250 mm; the height of the free edge, $\mathbf{h_1}$, of cantilevers that bear precast crane runway beams is taken as follows, depending on the capacity of the crane, **Q**:

$Q \le 5$ tons — $h_1 \ge 300$ mm,

5 tons $< Q <$ 15 tons — $h_1 \ge 400$ mm,

$Q \ge 15$ tons — $h_1 \ge 500$ mm;

and the condition $h_1 \ge (1/3)h$ must hold.

The strength of the oblique compressed strip between the load and the support is checked no matter there is or there is no transverse reinforcement.

When a beam is supported by a cantilever, the $\mathbf{Q_c}$ force is applied to the center of gravity of a punching triangle, i.e at the distance of 1/3 $\mathbf{L_2}$ from the edge of the cantilever. When the **Fixed area of bearing** checkbox is enabled, the $\mathbf{Q_c}$ force is applied at a given distance, $\mathbf{L_q}$, from the interior face of the top part of the column. Also, in this case the user must specify the length of the load transfer area, $\mathbf{L_{sup}}$. This area is assumed to be symmetric with respect to the application point of the $\mathbf{Q_c}$ load.

The width of the cantilever, **b**, is set equal to the width of the column. The width of the horizontal beam, $\mathbf{b_1}$, is set to be less than or equal to the width of the cantilever.

Two junctions between the beam and the column are taken into consideration: a hinged junction, and a rigid joint — it is immured and the beam's lower reinforcement is welded to the cantilever's reinforcement via fixings. If, in the rigid joint, the $\mathbf{M_1}$ moment puts the lower face of the beam in tension, then the moment should be used with the "minus" sign.

Transverse reinforcement for the cantilevers consists of:

- at **h** ≤ 2.5C — oblique stirrups over the whole height of the the console;
- at **h** > 2.5**C** — bent rebars and horizontal stirrups over the whole height of the the console;

here **C** is the distance from the interior face of the bottom of the column to the application point of force $\mathbf{Q_c}$.

The spacing of the stirrups is (in all cases) no less than **h**/4 and no greater than 150 mm.

By default, the transverse reinforcement of the cantilever is implemented as two-branch stirrups (thus the design formulas will include the double area of the transverse rebar specified in the dialog box).

In case when the transverse reinforcement consists of four-branch stirrups (the respective option should be selected), the design formulas will contain the quadruple area of transverse reinforcement.

In all cases when the outreach of the cantilever, $\mathbf{L_1}$, is less than the length of the area of bearing, $\mathbf{L_2}$, the analysis should take into account only the load on the cantilever located within the outreach.

Concrete of the cantilever under the area of bearing is subject to checking; the bearing stress in places where the load is transferred onto the cantilever may not exceed $\mathbf{R_{b,\,loc}}$ (Sec. 3.39 of SNiP II-23-81*), otherwise the class of concrete should be upgraded or the area of load bearing should be increased. In the check for local compression, the bearing area is taken equal to its effective design value.

In the check of longitudinal reinforcement, when the beam joins stiffly to the column, one of limitations of the horizontal force, $\mathbf{N_s}$, depends on the height and length of a corner weld that connects fixings of the beam with the cantilever. The least leg of the seam, $\mathbf{K_f}$, is assigned in such way so as to comply with Table 38* of SNiP II-23-81*; depending on the type of weld, the yield point of steel, the thickness of the thicker of the members to be welded together, it varies between 3 and 12 mm. The design strength of the corner welds for shear in the metal, $\mathbf{R_{wf}}$, is taken from Table 56 of SNiP II-23-81* depending on the electrode type (for E42, E42A it is 180 MPa; for E46, E46A it is 200 MPa; for E50, E50A it is 215 MPa; for E60 it is 240 MPa; for E70 it is 280 MPa; for E85 it is 240 MPa). The length of the corner weld, $\mathbf{l_w}$, that connects the fixings in the beam and in the column is determined by the sizes of the fixings, but it cannot exceed the double length of the area of load bearing along the outreach of the cantilever, $2\mathbf{L_2}$, and it cannot be less than $4\mathbf{K_f}$.

Limitations of the current version

Limitations related to the design arrangement of the joint are terminal: once they are violated, the respective analysis is prohibited.

The application operates only the class and the diameter of longitudinal and transverse reinforcement, therefore it is the user who accepts full responsibility for arrangement of reinforcement in the bulk of the cantilever, installation of proper anchoring (Sec. 5.14 of SNiP 2.03.01-84* and Sec. 5.45 of the Manual [7]) for longitudinal (working) reinforcement, and the form of transverse reinforcement (see recommendations above).

The use of stiff reinforcement to reinforce a cantilever of a limited height is out of consideration.

Limitations related to the constructional requirements are not terminal, and they allow to perform the analysis, but the user gets a warning. Some of the limitations are implemented “rigidly”: you are permitted to choose certain values from a limited list only — for example, the least leg of a seam, the reinforcement class etc.

Reinforcement

The longitudinal reinforcement and bent rebars of the cantilevers should be of Class A-III; Class A-II is also acceptable.

The stirrups and lateral rebars should be of Class A-I.

Concrete

Properties of concrete are specified on the **Concrete** tab following the steps defined in Section 3.2.

6. Reinforcement proportioning

6.1 Reinforcement proportioning for a beam

This mode is used to proportion reinforcement in multiple-span continuous beams of constant cross-sections using the criteria of strength and crack resistance which comply with SNiP 2.03.01-84* (SNiP 52-01-2003, SP 52-101-03). The subject of consideration is a planar bending of a beam under distributed and concentrated loads combined into loading patterns (loadings). The latter can be classified by their physical origin and properties into permanent, temporary but sustained, short-term, wind, and snow. The proportioning is performed for design stress combinations (DSC) created automatically. The DSC factors that take account of the nature of a particular loading are assigned by the application in compliance with SNiP 2.01.07-85*.

The beam is supposed to be free from longitudinal forces; only the following force actions are taken into account:

M, a bending moment;

Q, a shear force.

The analysis can be done for a beam of a rectangular, tee, or double tee section. The results of the analysis consist of the areas of top and bottom reinforcement on the segments, and the area and spacing of rebars of transverse reinforcement. The reinforcement thus proportioned is assumed to be the same along a particular segment, and the user should set the number and lengths of segments that the beam is divided into.

Source data for the proportioning are entered on the **General**, **Loads**, **Concrete**, **Crack Resistance**, and **Segments** tabs, and results are reviewed on the **Results** tab.

General

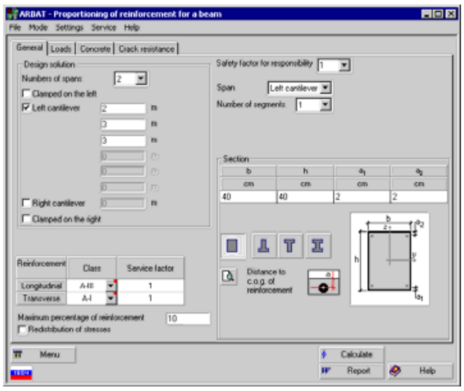


Fig. 80. *The* **General** *tab*

The **General** tab (Fig. 80) is used to assign geometrical characteristics to a multiple-span beam, specify its section's type, enter the sizes of the section and the distance to the center of gravity of the rebars, define the number and lengths of segments in the span, indicate the type and service factors for longitudinal and transverse reinforcement. Entering all this information is similar to working with the **Appraisal of Beam** mode (see section 4.3).

The number of segments for each span (cantilever) is selected from the **Number of segments** drop-down list. No. of the span is selected from the **Span** list.

The **Lengths of segments specified as** group is used to assign the method of length specification using appropriate radio buttons:

- **Absolute** — to specify the lengths of the segments in units of length;
- **Relative** — to specify the lengths of the segments as percentage of the total span length.

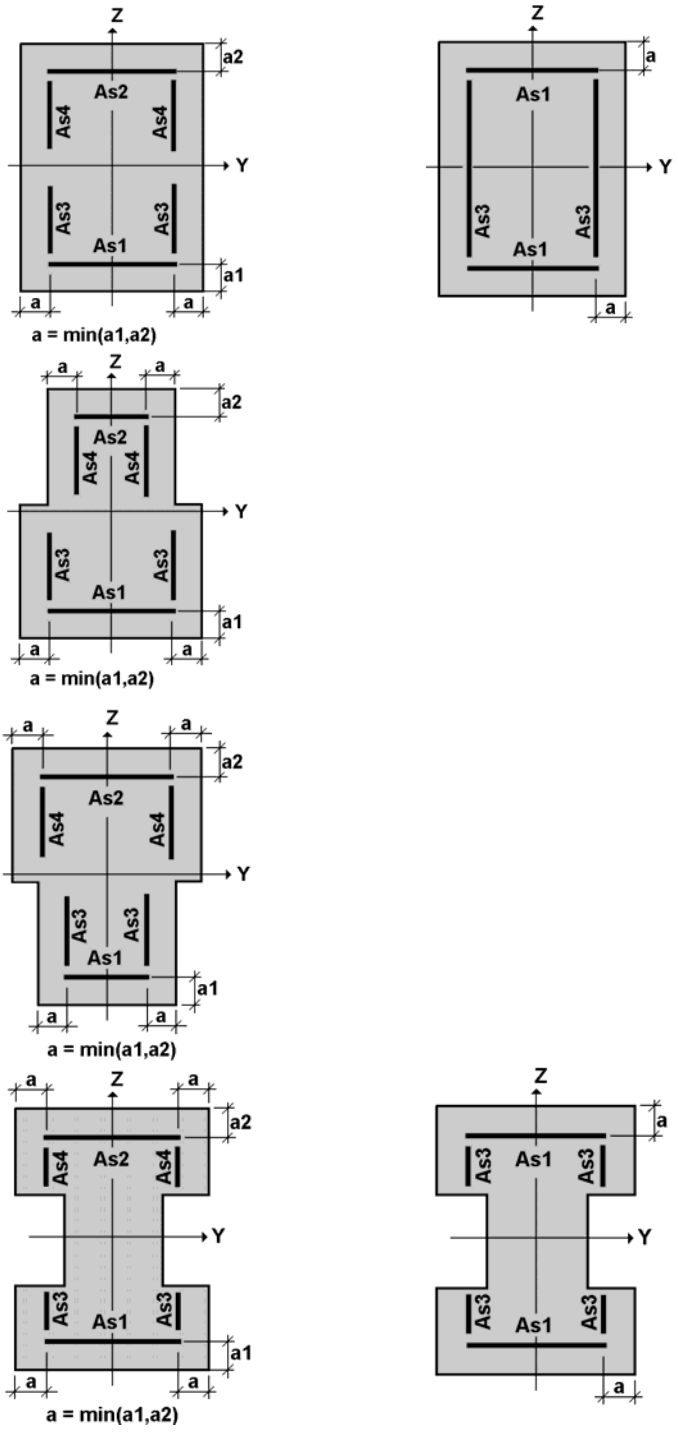
Depending on the way the lengths are specified, you should fill the table with either the lengths of the segments or the respective percent fractions for each span.

The shape is assigned to the section in the standard way (see Section 3.1). Just note that the distances to the center of gravity of reinforcement, $\mathbf{a_1}$ and $\mathbf{a_2}$, should be specified.

The arrangement of longitudinal and transverse reinforcement in the sections is shown in the respective Figs. 81 and 82.

This tab is also used to specify a maximum allowable percentage of reinforcement; when this value is exceeded, it means the proportioning of reinforcement has failed.

The way to take account of redistributed stresses is described in the **Appraisal of Beam** section; the respective checkbox should be enabled.



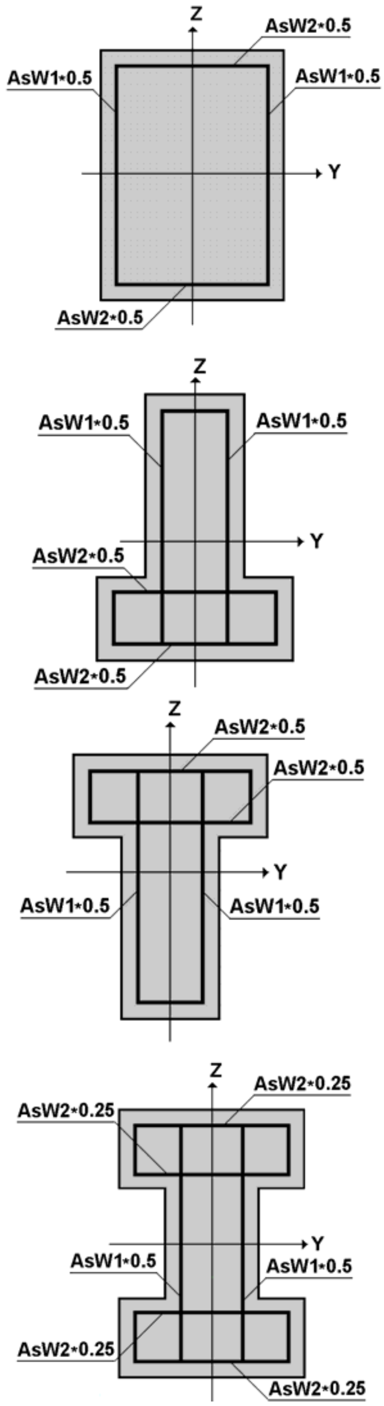


Fig. 81. *Arrangement of "areas" of longitudinal reinforcement*

Fig. 82. *Arrangement of transverse reinforcement*

`Loads`

Following the same procedure to specify the loads as described in the **Appraisal of Beam** mode (see Section 4.3).

`Concrete`

Enter the characteristics of concrete on the **Concrete** tab following the same steps as in Section 3.2.

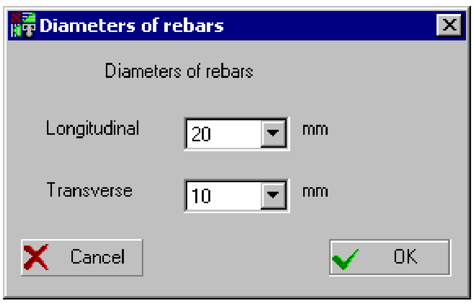
`Crack resistance`

Specify crack resistance data on the **Crack Resistance** tab following steps defined in Section 3.4.

In addition, if the third category of crack resistance has been selected, you have to specify the diameters of rebars of longitudinal and transverse reinforcement.

If the distance to the extreme row of longitudinal reinforcement is greater than that to the center or gravity of reinforcement defined on the **General** tab, it means a two-row reinforcement is required and, respectively, the reinforcement will be proportioned for the two-row arrangement.

By using the results of the reinforcement area proportioning and the **Discrete Reinforcement** service tool (see Section 8.5), you can determine the needed diameter and number of rebars. If the diameter of rebars turns out to be different from one specified in the **Crack Resistance** window, you will have to vary the diameter of longitudinal rebars and do a new calculation.



Results

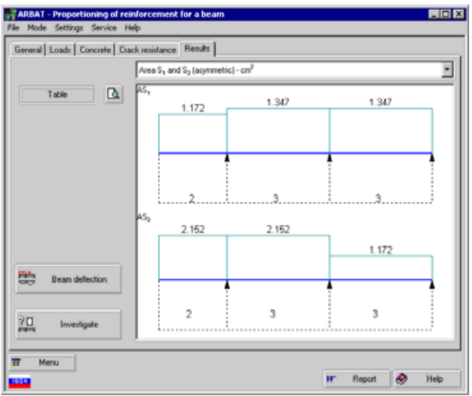


Fig. 83. *The* **Results** *tab*

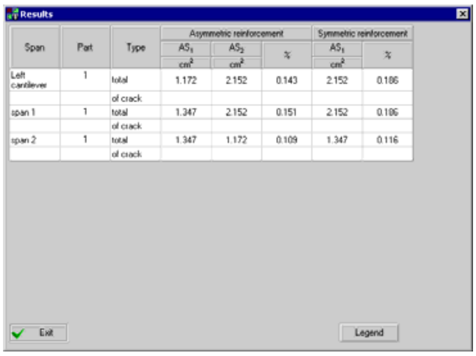
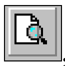


Fig. 84. *The* **Reinforcement Results** *dialog box*

Having prepared the source data, launch the proportioning process by clicking the **Calculate** button. After that, the **Results** tab will open (Fig. 83) and display distribution curves for the results of the proportioning. The form of representation for the results, the reinforcement percentage, and the crack opening width are selected from the drop-down list located in the top left corner of the window. For segments shown in red, the proportioning of reinforcement has failed. Information about the reason for this can be obtained from the table of results (the reason can be a maximum allowed percentage set by the user).

Depending on what is set in the list, the distribution curves can be displayed separately for each type of reinforcement or in couples. For example, you can have the curves AS_1 and AS_2 displayed together. Clicking the button , **Preview Reinforcement**, will open the **Reinforcement Layout** dialog box that shows areas of the proportioned longitudinal reinforcement in sections of each span of the beam. If the reinforcement proportioning has failed, the respective section is displayed in red.

Tabular data are displayed in a separate dialog box, **Reinforcement Results** (Fig. 84) which opens by clicking the **Table** button. Results of the proportioning for each segment are displayed on one line, if the crack resistance criterion does not require any additional reinforcement, or on two lines if such a reinforcement is required. The first line shows the overall reinforcement (for strength and crack resistance) and the second line shows the area of reinforcement added to ensure crack resistance.

If no reinforcement has been proportioned for a particular segment, the respective line of the **Type** column will display information about reasons why the error has occured.

Depending on the choice in the **Output transverse**

reinforcement group, the area of transverse reinforcement (stirrups) can be shown for a design value of the spacing (the **Default spacing** option) calculated during the proportioning or for a value specified by the user. In the latter case, click the **Apply** button after you enter the spacing.

Fig. 85. *The* **Rebar Diameters** *dialog box*

A report can be generated on the basis of reinforcement proportioning results (the **Report** button), which includes a schematic of the beam and schemes of loadings upon it, sizes of the section, properties of concrete and reinforcement, distribution curves for forces included in the loadings, and curves and a table of the proportioning results.

The application suggests the capability of passing the reinforcement proportioning results to the deflection calculation mode (the **Deflections** button) or to the appraisal mode (the **Appraisal** button). These two modes work with a particular reinforcement scheme for which the rebars are defined automatically on the basis of diameters specified on the **Crack Resistance** tab (the rebars are arranged in one row, and if the number of the rebars is greater than 40, an error message is generated). If the analysis has been done for the first category of crack resistance, then the diameter of the rebars should be specified by the user in the **Rebar Diameters** dialog box (Fig. 85) that opens after the above-said modes are called up.

6.2 Reinforcement proportioning for a single-span beam

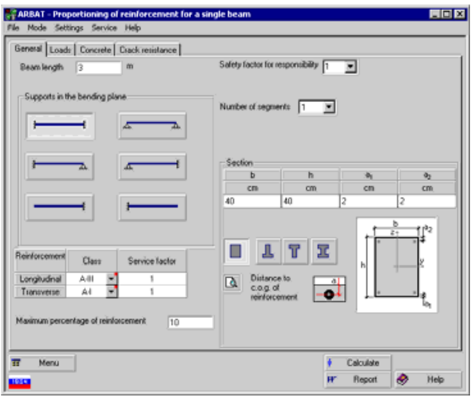


Fig. 86. *The* **General** *tab in the mode of reinforcement porportioning for a single-span beam*

This mode is similar to **Reinforcement Proportioning for Beam** (see the previous section). The differences are as follows:

- only the length of the beam needs to be specified (Fig. 86);
- a system of fixations should be chosen for the beam in the plane of its bending (to choose a fixation method, depress its respective button);
- no stress redistribution is taken into account;
- the **Loads** tab does not need to know the span to which the load is applied.

6.3 Reinforcement proportioning for a column

This mode is used to proportion the reinforcement area for a column of a constant section to comply with the criteria of strength and crack resistance defined by SNiP 2.03.01-84* (SNiP 52-01-2003, SP 52-101-03). An eccentric compression/tension with a biaxial eccentricity is under consideration. All checks of the column's sections involve automatically created design stress combinations (DSCs). DSC factors which take into account the nature of loading are assigned by the application on the basis of regulations from SNiP 2.01.07-85*.

When the proportioning complies with SNiP 2.03.01-84*, it is performed only for the first limit state with the following set of force actions:

N, a longitudinal force;

M_y, a moment that bends the member in the XoZ-plane with its vector along the Y-axis;

M_z, a moment that bends the member in the XoY-plane with its vector along the Z-axis;

Q_z, a shear force along the Z-axis;

Q_y, a shear force along the Y-axis;

M_t, a torque with its vector along the X-axis;

Otherwise, the following force actions are under consideration:

N, a longitudinal force;

M_y, a moment that bends the member in the XoZ-plane with its vector along the Y-axis;

Q_z, a shear force along the Z-axis;

When the analysis is to comply with SNiP 52-01-2003, the second limit state can be used with an arbitrary set of N, M_y, and M_z. The analysis can be done for a column of a rectangular, tee, double-tee, or annular section. The results of the analysis include the areas of symmetric or asymmetric longitudinal reinforcement, and the area and spacing of transverse reinforcement on the segments of the column. The reinforcement thus proportioned is assumed to remain constant throughout a particular segment. It is the user who choses the number and length of the segments the column is divided into.

Source data are prepared on the **General**, **Loads**, **Concrete**, **Segments**, and **Crack Resistance** tabs, and the results are reviewed on the **Results** tab.

General

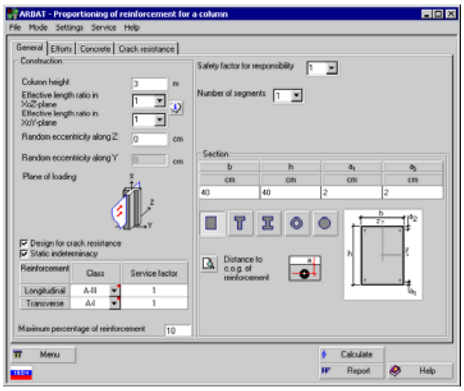


Fig. 87. *The* **General** *tab*

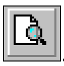
The **General** tab (Fig. 87) is used to specify the height of the column, its section's type and sizes, the distance to the center of gravity of reinforcement, effective length factors and values of the random eccentricity. You also choose the number of segments to divide the column into, assign classes and service factors for longitudinal and transverse reinforcement. Also, the state of the **Crack resistance analysis** checkbox defines whether it is necessary to proportion the reinforcement for the second limit state, and the **Statically indeterminant** checkbox specifies whether the column belongs to a statically indeterminate structure or to a statically determinate one.

The number of segments in the column is set in the **Number of segments** drop-down list.

The **Lengths of segments specified as** group lets you use radio buttons to define the way the lengths are specified:

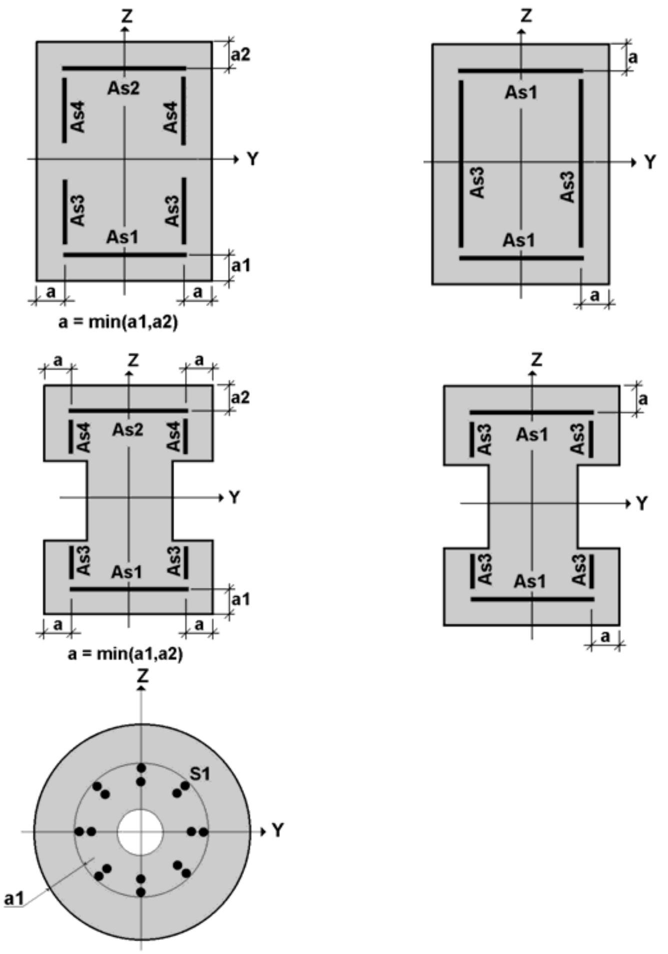
- **Absolute** — to specify the lengths of the segments in units of length;
- **Relative** — to specify the lengths of the segments as percentage of the total span length.

Depending on the way the lengths are specified, you should fill the table with either the lengths of the segments or the respective percent fractions for each segment. The numbering of the segments goes from the bottom to the top.

To choose a section shape, click the button that depicts the desired shape and enter the sizes and the distances to the center of gravity of reinforcement, $\mathbf{a_1}$ and $\mathbf{a_2}$, in the respective fields. The selected shape goes through verification in the **Section** dialog box which opens by clicking the button .

The arrangement of longitudinal and transverse reinforcement in the sections is presented in the respective Figs. 88 and 89.

The effective length factors and the random eccentricities are specified in the same way as in the **Resistance of ferroconcrete sections** mode (see Section 4.1).



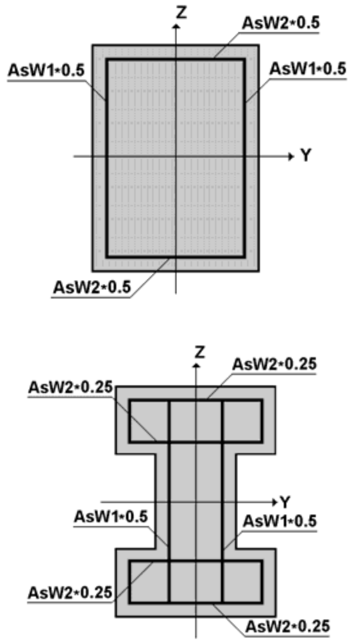


Fig. 88. *Arrangement of "areas" of longitudinal reinforcement*

Fig. 89. *Arrangement of transverse reinforcement*

Loads

Properties of loadings are specified on the **Loads** tab following the same steps as described in the **Appraisal of Column** mode (see Section 4.3).

Concrete

Properties of concrete are specified on the **Concrete** tab following the same steps as described in Section 3.2.

Crack resistance

Crack resistance data are specified on the **Crack Resistance** tab following steps described in Section 3.4.

The tab is always accessible for analyses based on SNiP 52-01-2003 and when the **Crack resistance analysis** (SNiP 2.03.01-84*) checkbox is enabled on the **General** tab.

Results

Having prepared the source data, launch the proportioning process by clicking the **Calculate** button. After that, the **Results** tab will open (Fig. 90) and display distribution curves for the results (the area of reinforcement, the percentage of reinforcement, the crack opening width). The form of representation for the results is selected from the drop-down list in the top left corner of the window. For segments shown in red, the proportioning of reinforcement has failed (this depends on the maximum

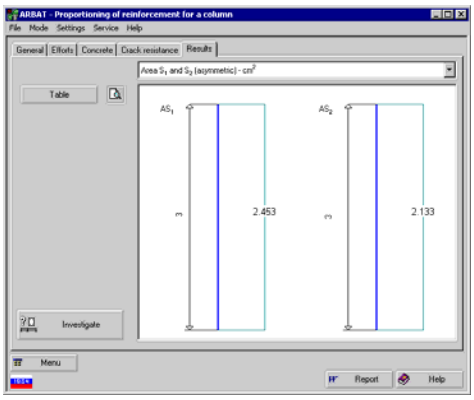


Fig. 90. *The* **Results** *tab*

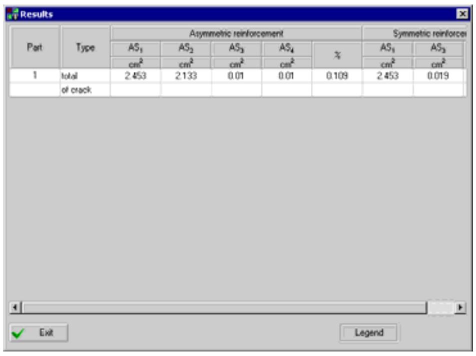


Fig. 91. *The* **Reinforcement Results** *dialog box*

reinforcement percentage defined by the user). Information about the reason for this can be obtained from the tables of results.

Depending on what is set in the list, the distribution curves can be displayed separately for each type of reinforcement or in couples. For example, you can have the curves AS_1 and AS_2 or AS_3 and AS_4 displayed together. Clicking the button , **Reinforcement Preview**, will open the **Reinforcement Layout** dialog box that shows areas of the proportioned longitudinal reinforcement in sections of each segment of the column. If the reinforcement proportioning has failed, the respective section is displayed in red.

Tabular data are displayed in a separate dialog box, **Reinforcement Results** (Fig. 91) which opens by clicking the **Table** button. Results of the proportioning for each segment are displayed on one line, if the crack resistance criterion does not require any additional reinforcement, or on two lines if such a reinforcement is required. The first line shows the overall reinforcement (for strength and crack resistance) and the second line shows the area of reinforcement added to ensure crack resistance.

If no reinforcement has been proportioned for a segment, the respective line of the **Type** column will display information about reasons why the error has occured.

Depending on the choice in the **Output transverse reinforcement** group, the area of transverse reinforcement (stirrups) can be shown for the design value of the spacing (the **Default spacing** option) calculated during the proportioning or for one specified by the user. In the latter case, click the **Apply** button after you enter the spacing.

A report can be generated after the results of reinforcement proportioning (the **Report** button), which includes a schematic of the column and parameters of its loadings, sizes of the section, properties of concrete and reinforcement, a table of the proportioning results.

The application suggests the capability of passing the proportioning results to the reinforcement appraisal mode (the **Appraisal** button). The number of rebars is determined by the same rules as in the **Reinforcement Proportioning for Beam** mode (see Section 6.1).

Features of the current version

Analysis of eccentrically compressed members

According to Section 3.24 of SNiP 2.03.01-84*, the structural analysis of such members is based on the non-deformed shape of a structure, where at the flexibility of l_0/i > 14 the effect of the member's deflection on its strength should be taken into account (l_0 is an effective length of the member, i is a radius of inertia in the plane of loading of the member's cross-section).

Analysis of weakly reinforced cross-sections

When calculating the crack opening width for a weakly reinforced section, at $M = M_{crc}$ the cracks get too widely open. The application reduces the value of a_{crc} in the range $M_{crc} \le M \le M_0$ by multiplying it by a factor that takes account of the work of tensioned concrete above the crack [18].

Limitations of the current version

When the design code to comply with is SNiP 2.03.01-84*:

There is no analysis of ferroconcrete members made of cellular, porous, or tensioning concrete.

There is no analysis of prestressed ferroconcrete members.

There is no analysis for endurance.

When doing the second limit state analysis, there is no analysis of crack closing.

7. Geometrical characteristics

This mode is used to calculate numerical geometrical characteristics of a section. The section itself and data concerning its concrete and reinforcement are specified according to rules defined earlier (see Section 3). Results are displayed on the **Geometrical Characteristics** tab in the form shown in Fig. 88, they consist of an area, moments of inertia of the concrete and transformed section, a percentage of reinforcement. These pieces of data can be used in static or dynamic structural analysis. For example, if you specify the stiffness of a bar element in the **SCAD** environment in a numerical-parametrical form and use data obtained from the **Geometrical Characteristics** mode, you will have the effect of reinforcement taken into account.

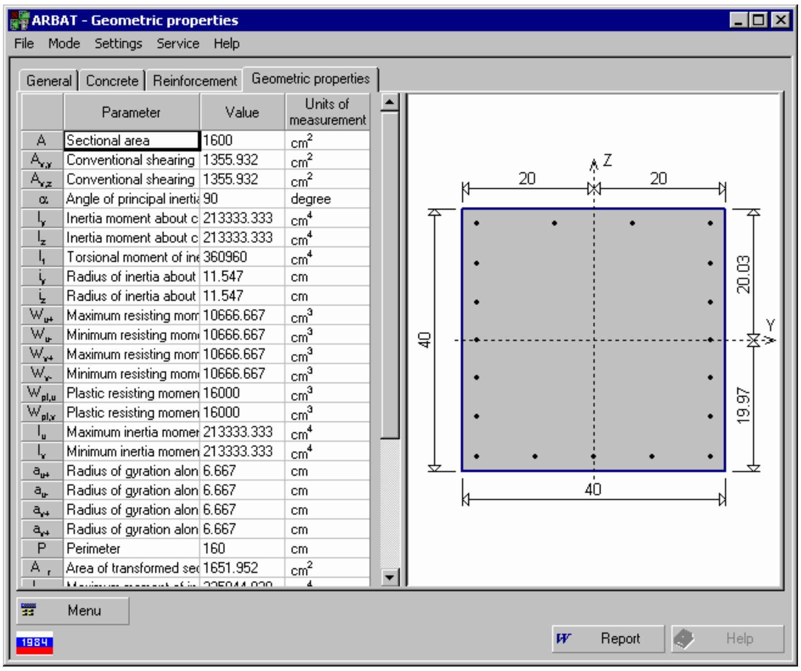


Fig. 88. *The* **Geometrical Characteristics** *dialog box*

8. APPENDIX

8.1 On seismic actions

The **ARBAT** application does not know the notion of "seismic load". The reason is that Sec. 2.14 of SNiP II-7-81* "Construction in seismic regions" [3] demands to use an additional service factor greater than 1 (see Table 7 of SNiP II-7-81*). When a structure is analyzed for combined action of multiple loads (structural constructions are nearly always subjected to more than one load at a time), one of them being seismic, SNiP formally requires than this coefficient should be used even in cases when the fraction of the seismic action is small comparing to other (sustained) loads. This requirement may produce "dangerous results".

The user can always take seismic effects into account by setting appropriate service factors for concrete and reinforcement.

8.2 Design code documents implemented in the ARBAT environment

Mode	**References to clauses of regulations**	
	SNiP 2.03.01-84*	**SNiP 52-01-03** **SP 52-101-03**
Concrete class	Table 12, 13 of SNiP 2.03.01-84*	Sec. 2.1.1, Table 2.1-1, 2.1-2, 2.1-3 of SP 51-101-03 Sec. 5.1.1–5.1.3 of SNiP 52-01-03
Concrete grade	Table 11, 13 of SNiP II-21-75	Table 11, 13 of SNiP II-21-75
Reinforcement	Table 19, 22 of SNiP 2.03.01-84*; GOST 5781-82	Sec. 5.3.1-5.3.3 of SNiP 52-01-03 Sec. 2.2.1, Table 2.2-1, 2.2-2 of SP 51-101-03
Service factors	Table 15–17 of SNiP 2.03.01-84*	Sec. 5.2.3, 5.4.3 of SNiP 52-01-03 Sec. 2.1.2.3, 2.2.2.2, 2.2.2.3 of SP 51-101-03
Deflection limits	Table 19, 21, 22 of SNiP 2.01.07-85*	Table 19, 21, 22 of SNiP 2.01.07-85*
Resistance of f/c sections	Sec. 3.10–3.12, 3.15–3.20, 3.24, 3.26, 3.28, 3.30, 3.32 of SNiP; Sec. 3.1, 3.11–3.23, 3.50–3.54, 3.61–3.62, 3.64–3.68, 3.30–3.33, 3.40, 4.2–4.4, 4.7–4.9, 4.11 of the Guide to SNiP 2.03.01-84* [7]	Sec. 4.2.6, 5.1.3, 5.1.8–5.13, 5.1.17–5.1.22, 5.1.24, 5.2.3, 5.2.6–5.2.12, 6.2.1–6.2.3, 6.2.16, 6.2.21–6.2.34, 6.2.37, 7.1.2–7.1.3, 7.2.2–7.2.5, 7.2.11–7.2.12, 7.2.14, 7.2.15 of SP 52-101-03; Sec. 4.3, 5.2.1–5.2.4, 5.4.1–5.4.4, 6.1.6, 6.2.1, 6.2.8–6.2.11, 6.2.13, 6.3.1, 6.3.4, 6.4.1–6.4.5 of SNiP 52-01-2003; Sec. 3.29–3.35, 3.52, 3.71, 4.28 of the Guide to SP 52-101-2003
Resistance of concrete sections	Sec. 1.21, 3.1–3.5, 3.6, 3.8, 3.30, 3.32 SNiP 2.03.01-84*; Sec. 3.54, 4.4 of the Guide to SNiP 2.03.01-84*.	Sec. 4.1.2, 4.2.6, 5.1.3, 5.1.8–5.1.13, 5.1.17–5.1.22, 5.1.24, 6.1.2, 6.1.7, 6.1.8, 6.1.11, 6.2.16, 6.2.21–6.2.31 SP 52-101-03; Sec. 5.2.1–5.2.4, 6.1.6, 6.2.1, 6.2.5, 6.2.6 of SNiP 52-01-2003; Sec. 3.29–3.35, 3.52, 3.71, 4.28 of the Guide to SP 52-101-2003

Mode	References to clauses of regulations	
	SNiP 2.03.01-84*	**SNiP 52-01-03 SP 52-101-03**
Beam deflection	Sec. 4.24, 4.27–4.31 of SNiP 2.03.01-84*; Sec. 4.12–4.19, 4.21, 4.22 of the Guide to SNiP 2.03.01-84* [7]	Sec. 5.1.3, 5.1.8–5.13, 5.1.17–5.1.23, 5.2.3, 5.2.6–5.2.12, 6.2.1, 6.2.2, 6.2.21–6.2.34, 7.1.2–7.1.3, 7.2.5, 7.2.11, 7.3.1, 7.3.3, 7.3.4, 7.3.7, 7.3.8, 7.3.16 of SP 52-101-03; Sec. 5.2.1–5.2.4, 5.4.1–5.4.4, 6.1.6, 6.2.1, 6.2.8, 6.2.10, 6.2.11, 6.2.13, 6.3.1, 6.3.4, 6.5.1, 6.5.3, 6.5.4 of SNiP 52-01-2003; Sec. 4.17, 4.18, 4.20–4.22, 4.27, 4.28 of the Guide to SP 52-101-2003
Appraisal of beam	Sec. 3.10–3.12, 3.15–3.18, 3.30, 3.32 of SNiP 2.03.01-84*; Sec. 3.1, 3.11–3.23, 3.30–3.33, 3.40, 4.2–4.4, 4.7–4.9, 4.11 of the Guide to SNiP 2.03.01-84* [7]	Sec. 5.1.3, 5.1.8 – 5.13, 5.1.17-5.1.22, 5.1.24, 5.2.3, 5.2.6–5.2.12, 6.2.1, 6.2.2, 6.2.21–6.2.34, 7.1.2–7.1.3, 7.2.2–7.2.5, 7.2.11–7.2.12, 7.2.14, 7.2.15 of SP 52-101-03; Sec. 4.3, 5.2.1–5.2.4, 5.4.1–5.4.4, 6.1.6, 6.2.1, 6.2.8, 6.2.10, 6.2.11, 6.2.13, 6.3.1, 6.3.4, 6.4.1 – 6.4.5 of SNiP 52-01-2003; Sec. 4.28 of the Guide to SP 52-101-2003
Appraisal of column	Sec. 3.10–3.12, 3.19–3.20, 3.24, 3.26, 3.28, 3.30, 3.32 SNiP 2.03.01-84*; Sec. 3.1, 3.50–3.54, 3.61–3.62, 3.64–3.68, 3.30–3.33, 3.40, 4.2–4.4, 4.7–4.9, 4.11 of the Guide to SNiP 2.03.01-84 [7]	Sec. 4.2.6, 5.1.3, 5.1.8–5.13, 5.1.17–5.1.22, 5.1.24, 5.2.3, 5.2.6–5.2.12, 6.2.1–6.2.3, 6.2.16, 6.2.21–6.2.34, 6.2.37, 7.1.2–7.1.3, 7.2.2–7.2.5, 7.2.11–7.2.12, 7.2.14, 7.2.15 of SP 52-101-03; Sec. 4.3, 5.2.1–5.2.4, 5.4.1–5.4.4, 6.1.6, 6.2.1, 6.2.8–6.2.11, 6.2.13, 6.3.1, 6.3.4, 6.4.1–6.4.5 of SNiP 52-01-2003; Sec. 3.29–3.35, 3.52, 3.71, 4.28 of the Guide to SP 52-101-2003
Appraisal of slab	Sec. 2.3, 2.12–2.14, 2.17, 2.26, 2.30, 4.1, 4.2, 4.5, 4.13, 4.14, 5.3, 5.5, 5.16 of SNiP 2.03.01-84*; Sec. 18, 19; Sec. 6.19, 6.23, 6.29, 6.37, 6.40, 6.42–6.44, 6.46, 6.49, 6.50 of the Guide to SNiP 2.03.01-84*[7]	Sec.5.1.3, 5.1.8, 5.1.9, 5.1.13, 5.2.1, 5.2.2, 5.2.3, 5.2.6, 5.2.10, 8.3.1, 8.3.2, 8.3.3, 8.3.4 of SP 52-101-03; Sec. 18, 19 of the Instruction Manual [18]

Mode	References to clauses of regulations	
	SNiP 2.03.01-84*	SNiP 52-01-03 SP 52-101-03
Reinforcement proportioning for beam	Sec. 3.10–3.12, 3.15–3.18, 3.30, 3.32 of SNiP 2.03.01-84*; Sec. 3.1, 3.11–3.23, 3.30–3.33, 3.40, 4.2–4.4, 4.7–4.9, 4.11 of the Guide to SNiP 2.03.01-84* [7]	Sec. 5.1.3, 5.1.8 – 5.13, 5.1.17-5.1.22, 5.1.24, 5.2.3, 5.2.6 – 5.2.12, 6.2.1, 6.2.2, 6.2.21 – 6.2.34, 7.1.2 – 7.1.3, 7.2.2 – 7.2.5, 7.2.11-7.2.12, 7.2.14, 7.2.15 of SP 52-101-03; Sec. 4.3, 5.2.1 – 5.2.4, 5.4.1 – 5.4.4, 6.1.6, 6.2.1, 6.2.8, 6.2.10, 6.2.11, 6.2.13, 6.3.1, 6.3.4, 6.4.1 – 6.4.5 of SNiP 52-01-2003; Sec. 4.28 of the Guide to SP 52-101-2003
Reinforcement proportioning for column	Sec. 3.10–3.12, 3.19–3.20, 3.24, 3.6, 3.28, 3.30, 3.32 of SNiP 2.03.01-84*; Sec. 3.1, 3.30–3.33, 3.40, 3.50–3.54, 3.61–3.62, 3.64–3.68, 4.2–4.4, 4.7–4.9, 4.11 of the Guide to SNiP 2.03.01-84*	Sec. 4.2.6, 5.1.3, 5.1.8–5.13, 5.1.17–5.1.22, 5.1.24, 5.2.3, 5.2.6 – 5.2.12, 6.2.1–6.2.3, 6.2.16, 6.2.21–6.2.34, 6.2.37, 7.1.2–7.1.3, 7.2.2–7.2.5, 7.2.11–7.2.12, 7.2.14, 7.2.15 of SP 52-101-03; Sec. 4.3, 5.2.1–5.2.4, 5.4.1–5.4.4, 6.1.6, 6.2.1, 6.2.8–6.2.11, 6.2.13, 6.3.1, 6.3.4, 6.4.1–6.4.5 of SNiP 52-01-2003; Sec. 3.29-3.35, 3.52, 3.71, 4.28 of the Guide to SP 52-101-2003
Local compression	Sec. 3.39–3.41 of SNiP 2.03.01-84*; Sec. 3.94 of the Guide to SNiP 2.03.01-84*	Sec. 6.2.43–6.2.45 of SP 52-101-03
Punching	Sec. 3.42, 5.29 of SNiP 2.03.01-84*; Sec. 3.98 of the Manual [16]	Sec. 6.2.46–6.2.52 of SP 52-101-03
Tearing	Sec. 3.43 SNiP 2.03.01-84*; Sec. 3.97 of the Guide to SNiP 2.03.01-84* [7]; Sec. 3.121 of the Guide to SNiP 2.03.01-84* [15]	
Fixings	Sec. 3.44–3.46 SNiP 2.03.01-84*	
Short cantilevers	Sec. 3.34 SNiP 2.03.01-84*; Sec. 3.99 of the Guide to SNiP 2.03.01-84* [7]	

8.3 Formula calculator

The formula calculator can be launched from the **SCAD Office** program group by clicking the icon . The **Tools** menu can be used to start either the standard MS Windows calculator (provided it has been installed with the system) or a special kind of calculator (Fig. 89) that performs calculations of formulas.

The calculator takes a formula specified in its input field and does the calculation of it. The following rules should be observed when entering a formula:

- names of functions must be entered in lowercase Roman letters;

- the fractional and the integral parts of a number are separated by a period;
- arithmetic operations are specified by symbols +, –, *, /, ^ (raising to a power), for example, 2.5*2.5*2.5 can be written also as 2.5^3.

The following mathematical functions can be used in the formulas:

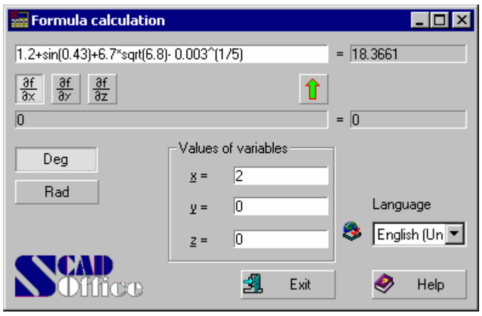


Fig. 89. *The dialog box of the calculator*

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;
log — natural logarithm;
log10 — 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.

The following formula,

$1,2+\sin(0,43)+6,7\sqrt{6,8}-\sqrt[5]{0,003}$

must be written as follows:

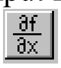
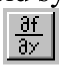
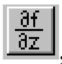
1.2+sin(0.43)+6.7*sqrt(6.8)–0.003^(1/5).

There is an additional option of using three independent variables **x, y, z** in formulas. Values for the variables should be specified in respective edit fields. This makes it possible to perform a series of similar calculations with different parameters. For example, to use this mode with the following formula,

$1,2+\sin(x)+6,7\sqrt{6,8}-\sqrt[5]{y}$

write it as

1.2+sin(x)+6.7*sqrt(6.8)–y^(1/5).

he application accepts into its main input field symbolic expressions that depend on variables *x, y, z*; enable one of the switch buttons,  ,  , , to get a symbolic expression of the respective partial derivative.

8.4 Converter of measurement units

The calculator can be invoked either from the **SCAD Office** program group — with the icon  — or from the **Tools** menu. This application converts data between different systems of measurement units (Fig. 90). To do the action, select a tab of respective measures (**Length**, **Area** etc.).

The procedure of conversion depends on whether the units of measurement are simple (like length, area, or mass) or compound (like pressure or velocity).

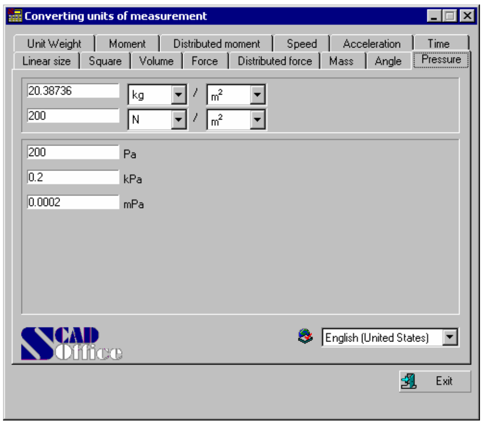


Fig. 90. *The* **Convert Units of Measurement** *dialog box*

To convert simple units, just enter a number in one of the edit fields. The other fields will display values of the same quantity in other units of measurement. If the units are compound, you choose the name of units to convert from in the drop-down lists of one line and then choose the name of units to convert into in the lists of the second line. Enter a number in the edit field of the first line, and you will see results of this conversion in the edit field of the other line.

8.5 Discrete reinforcement

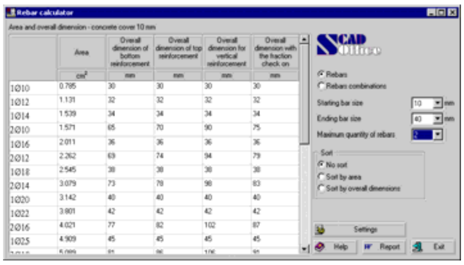


Fig. 91. *The* **Discrete Reinforcement** *dialog box*

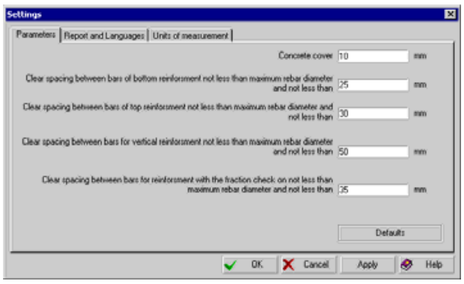


Fig. 92. *The* **Parameters** *dialog box*

The **Discrete Reinforcement** calculator (Fig. 91) generates all possible combinations of rebars for given limitations of the diameter and number of the rebars.

The following types of limitations can be specified:

- when the **Rebars** checkbox is enabled, it means that rebars of different diameters are never combined, and the application will generate all schemes of reinforcement which contain no more than a given maximum nuber of rebars from the range set in the **Start diameter** and **End diameter** lists;
- when the **Combinations of rebars** checkbox is enabled, it means the application will find all combinations that include two rebars of different diameters from the range set in the **Start diameter** and **End diameter** lists.

The table of results contains a list of combinations and the following data for each one of those: an area of reinforcement, minimum overall dimensions of the bottom/top reinforcement and vertical reinforcement, and the dimensions with a check of fractions.

All dimensions include the thickness of the cover which is set in the **Settings** dialog box, the **Parameters** tab (Fig. 92). The same tab can be used to change the minimum spacing between rebars recommended by SNiP or, by clicking the **Default** button, return to the default values. The minimum overall dimensions are calculated in accordance with limitations that SNiP defines for the arrangement of reinforcement in concrete sections.

Use appropriate radio buttons to sort the list by the area of reinforcement or by the overall dimensions.

9. References

1. GOST 27751-88. Safety of structural constructions and foundations. Guidelines for analysis. (ST SEV 384-87). Moscow, USSR State Committee for Standards, 1989.
2. SNiP 2.01.07-85*. Loads and actions / Russia State Committee for Construction and Architecture. Moscow, 2001. 44 p.
3. SNiP II-7-81* Construction in seismic regions. Building regulations / Ministry of Construction of Russia. Moscow, 1996. 52 p.
4. SNiP II-21-75. Concrete and ferroconcrete constructions. Moscow, "Stroyizdat", 1976, 89 p.
5. SNiP 2.03.01-84*. Concrete and ferroconcrete constructions. / Ministry of Construction of Russia. Moscow, USSR State Comittee for Construction and Architecture Press, 1989, 80 p.
6. TSN 102-00. Ferroconcrete constructions with reinforcement of classes A500S and A400S: Local building regulations for Moscow.
7. Manual of engineering of concrete and ferroconcrete constructions made of heavy-weight or light-weight concrete, with no prestressing of reinforcement (in addition to SNiP 2.03.01-84*) / Central Res. Inst. of Industrial Buildings and Res. Inst. for Ferroconcrete of USSR State Comm. for Constr. and Archit. Moscow, "Stroyizdat", 1986, 188 p.
8. SNiP 52-01-2003. Concrete and ferroconcrete constructions. Guidelines. Moscow, 2004, 24 p.
9. SP 52-101-03. Concrete and ferroconcrete constructions with no prestressing of reinforcement. Moscow, 2003, 126 p.
10. Manual of engineering of concrete and ferroconcrete constructions made of heavy-weight concrete, with no prestressing of reinforcement (in addition to SP 52-101-03). Moscow, 214 p.
11. GOST 26633-91. Heavy-weight and fine-grained types of concrete. Specifications.
12. GOST 23279-85. Welded reinforcement mats for ferroconcrete constructions and products. General specifications.
13. Murashev V.I., Sigalov E.B., Baikov V.N. Ferroconcrete constructions. Moscow, "Gosstroyizdat", 1962. 660 p.
14. Recommendations on engineering of steel fixings in ferroconcrete constructions, Res. Inst. for Ferroconcrete, USSR State Committee for Construction and Architecture. Moscow, "Stroyizdat", 1984, 87 p.
15. Manual of engineering of concrete and ferroconcrete constructions made of heavy-weight concrete (no prestressing) / Head Design Inst. of Leningrad, Department of Industrial Engineering of USSR State Comm. for Constr. and Archit., Central Res. Inst. of Industrial Buildings and Res. Inst. for Ferroconcrete of USSR State Comm. for Constr. and Archit. Moscow, "Stroyizdat", 1978, 175 p.
16. Manual of engineering of concrete and ferroconcrete constructions made of heavy-weight concrete (no prestressing) / Central Res. Inst. of Industrial Buildings and Res. Inst. for Ferroconcrete of USSR State Comm. for Constr. and Archit. Moscow, "Stroyizdat", 1978, 321 p.
17. Manual of engineering of residential buildings.— Vol.3. Designs of residential buildings (to SNiP 2.08.01-85) / Centr. Resrch. Inst. for Resident. Engng., State Committee for Architecture. Moscow, "Stroyizdat", 1989. 304 p.
18. Instruction manual for analysis of statically indeterminate ferroconcrete constructions, with redistribution of stresses taken into account / Res. Inst. for Ferroconcrete.— Moscow, "Stroyizdat", 1961. 112 p.
19. Murashev V.I., Sigalov E.B., Baikov V.N. Ferroconcrete constructions. Moscow, "Gosstroyizdat", 1962. 660 p.
20. Manual of analysis of statically indeterminate ferroconcrete constructions. Moscow, "Stroyizdat", 1975. 192 p.