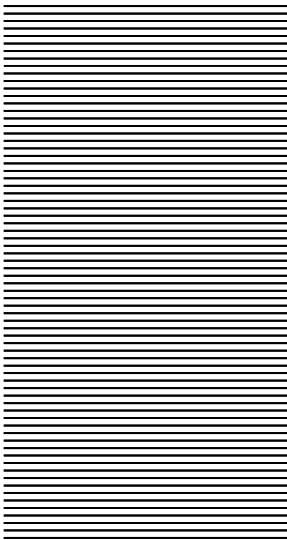
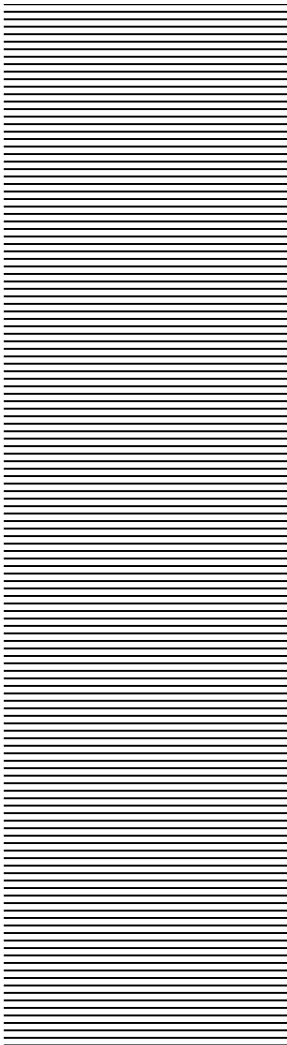


SCAD Soft



SCAD **Structure**



Decor
Analysis of wooden structural
members

User manual

UDC 737.30

Team of authors

E.Z. Kriksunov, M.A. Perelmuter, L.N. Skoruk, V.V. Fursov

DECOR. Analysis of wooden structural members.

USER MANUAL. Version 1.1.

The manual provides a description of functionality, controls, and recommendations for practical usage of the **DECOR** software application.

The software is oriented at structural design engineers who have basic computer skills.

SCAD Soft, 2006 ©

Table of contents

1.	Introduction	4
1.1	Structural assessment	4
1.2	The main window	5
1.3	Menus	6
1.4	Settings	7
1.5	Working with tables	8
2.	Information modes	10
2.1	Deflection and strain limits	10
2.2	Densities	11
2.3	Range of timber	11
2.4	Design strength	12
2.5	Wood	13
2.6	Limit slenderness	14
3.	Analysis	15
3.1	General management	15
3.1.1	Building cross-sections	15
3.1.2	Properties of wood	17
3.1.3	Service factors	17
3.1.4	Creating a report	17
3.2	Geometrical characteristics	18
3.3	Effective length	19
3.4	Joint resistance	19
3.5	Section resistance	21
3.5.1	Limitations of the current version	23
3.6	Continuous girders	24
3.6.1	Calculation	26
3.6.2	Limitations of the current version	27
3.7	Suspended spans	27
3.7.1	Limitations of the current version	27
3.8	Beams	27
3.8.1	Limitations of the current version	29
3.9	Columns	29
3.9.1	Limitations of the current version	31
3.10	Trusses	31
3.11	Truss element	34
4.	APPENDIX	36
4.1	Formula calculator	36
4.2	Converter of measurement units	36

1. Introduction

The **DECOR** software application does an analysis and checks of wooden structural members and joints for compliance with regulations of SNiP II-25-80 “Wooden constructions”. The application also provides reference data most often used when designing wooden constructions.

Principles of control, methods for preparing data and documenting the analysis results are exactly the same as those in the similar modes of computer-aided design software included in the **SCAD Office®** software system. All the applications are based on multi-tab windows and dialog boxes. To switch to a tab, click its title 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:

$$F_j(S, R) \leq 1, \quad (j = 1, \dots, n),$$

where F_j is a function of principal variables that implements j -th check/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 a particular analysis criterion as

$$\max_j K_j \leq 1,$$

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).

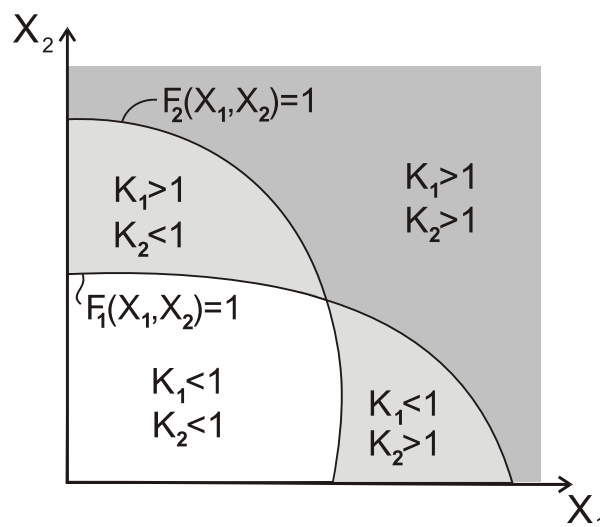
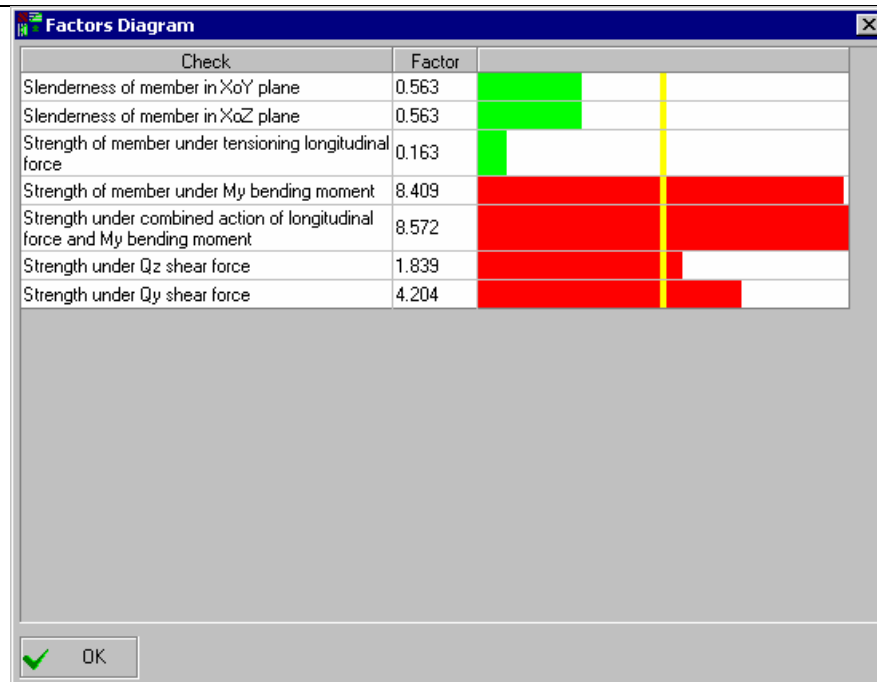


Fig. 1. A geometric illustration of the validity area for two variables

All values of the K_j factors obtained by analysis are available for reviewing in the **Criteria Diagram** dialog box (Fig. 2) or in the full report of the respective analysis. Appropriate dialog boxes display the value of $K_{\max}$ — a 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.

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. For example, it is hardly of any sense to improve the design strength if it is the stability that has a critical importance.

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

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 from a “map” of buttons, where design codes can be chosen, and settings of the whole application can be customized.



Fig. 3. The main window

Introduction

A particular set of design regulations can be selected from an appropriate list. The set of regulations that has been selected is displayed in the bottom left corner of the active mode's window. In this first version of the application, only the analysis based on SNiP II-25-80 is implemented.

Working modes are activated using their particular buttons; they can be classified into reference/information modes (the **Information** group) or analytic modes (the **Analysis** modes).

Here is the set of reference/information modes:

Deflection and strain limits — helps browse limit values of the deflections in wooden structural members and limit values of strains in joints, as shown in Tables 15, 16 of SNiP II-25-80;

Densities — provides data concerning the density of wood from Appendix 3 of SNiP II-25-80;

Range of timber — provides data concerning the sizes of softwood timber according to GOST 24454-80;

Design strength — calculates the design strength for a type of stress and strain distribution selected by the user;

Wood — provides data concerning the maximum allowed humidity of wood, the types and brands of adhesives;

Limit slenderness — provides information about the limit slenderness ratios of structural members.

The **Analysis** group of modes performs the following actions:

Geometrical characteristics — calculates the geometrical characteristics of a cross-section;

Effective length — implements recommendations of Sec. 4.21 of SNiP II-25-80 for calculation of effective length values;

Joint resistance — calculates limitation utilization factors for scarf/notch/cog joints and dowel pin joints;

Section resistance — calculates limitation utilization factors for any cross-section type suggested by the application under action of any forces; this mode also builds curves of interaction for any admissible combinations of stress couples;

Continuous girders — analyzes a continuous girder for limit states of first and second group;

Suspended spans — analyzes a suspended span together with its supporting cantilevers under a uniformly distributed load for limit states of first and second group;

Beams — analyzes a single-span beam with various conditions of support for limit states of first and second group;

Columns — this mode performs a structural appraisal of columns or stanchions;

Truss element — this mode does a structural appraisal of a separate element of a truss;

Trusses — this mode works with structural designs most frequently used in practice; it implements all kinds of appraisals of strength and stability of truss elements. The work begins with determining values of design stresses caused by given vertical external loads.

When you invoke any of the above listed modes, a multiple-tab dialog box appears where you enter whatever data are needed and review the results of the analysis that follows.

1.3 Menus

The drop-down menus are used to customize the application's settings, to invoke a desired working mode, or to use a service tool. There are five menus: **File**, **Modes**, **Settings**, **Tools**, **Help**.

The items of the **File** menu are used to do the following:

Menu — to switch from any working mode to the main window;

Exit — to finish the current session.

The **Modes** menu helps launch any of the implemented working modes, whether for information or analysis (it duplicates the respective buttons of the main window).

The **Settings** mode calls up the **Application Settings** dialog box where you set up whatever preferences or settings exist in the application's environment (there is a button in the main window that does the same).

The **Tools** menu items call the standard Windows calculator, a formula calculator, and a measurement unit converter utility.

The **Help** menu provides helpful information on how to use the **DECOR** software, how to use the Windows help system, and basic information about the application (No. of its version and the date of its last modification).

Auxiliary tools and operations on getting help are described in the appendix.

1.4 Settings

The **Application Settings** dialog box (Fig. 4) is called up by the **Settings** menu item or by a button under the same name located at the bottom of the main window. It contains three tabs: **Units of Measurement**, **Report and Languages**, **Visualization**.

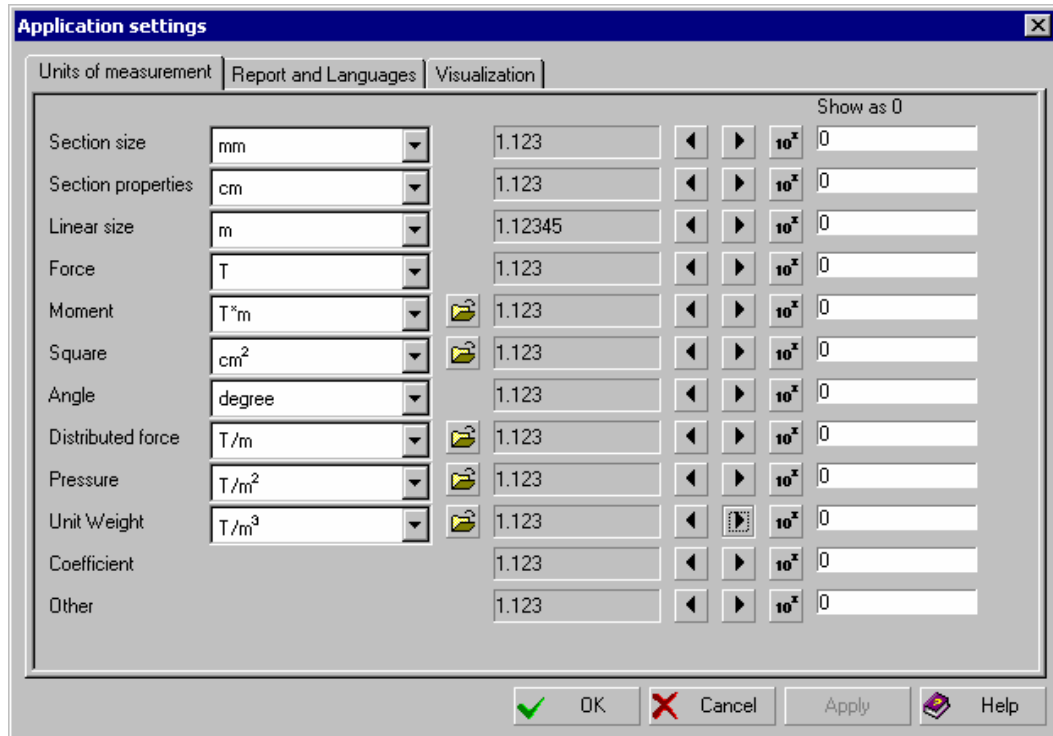


Fig. 4. The **Units of Measurement** tab of the **Application Settings** dialog box

The **Units of Measurement** tab is used to set up physical units of measurement which will be used to enter source data and review results of the analysis. The units of measurement can be altered at any time while working with the application. To set up simple units such as linear sizes or forces, use drop-down lists. In cases when the units are complex, the drop-down lists display the current ones, and the settings are made in the **Set up Units of Measurement** dialog boxes (Fig. 5). To open one of those dialog boxes, click the  button to the right of the respective list. To specify the desired units, choose appropriate items from the drop-down lists available in the dialog box and then click **OK** to exit the dialog.

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.

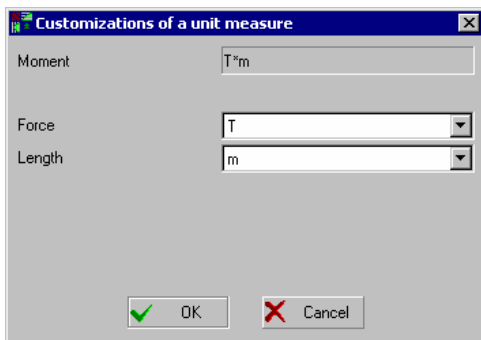


Fig. 5. The **Set up Units of Measurement** dialog box

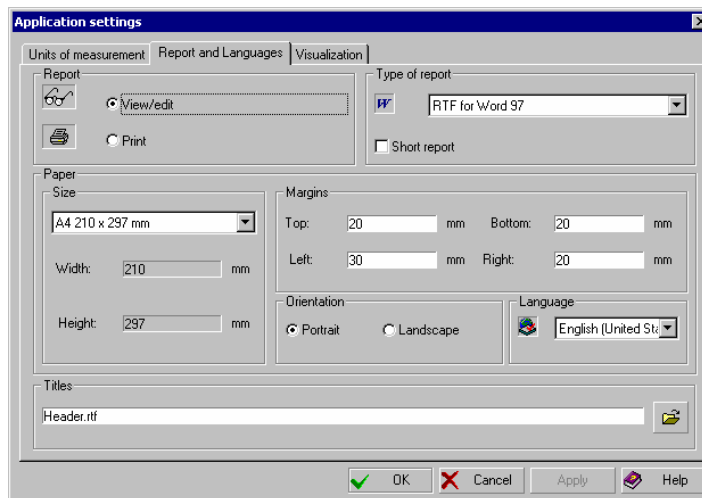


Fig. 6. The **Report and Languages** tab of the **Application Settings** dialog box

The **Visualization** tab is used to choose a font for the text messages displayed on the screen and printed out to 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 kind of analysis are provided as tables (Fig. 7). The following general rules are used for working with the tabular data:

- the data are entered in a 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 one or more selected rows;
- to select one or more successive rows, place 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.

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 in 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.

Decor - Section resistance

File Mode Settings Service Help

Общие параметры Efforts Effective length in the XoY plane Effective length in the XoZ plane Interaction curves

Section

Change force plane

☐ Efforts is taken from the calculation under the deformed scheme

	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
2	-129.577	-1.067	0.175	1.243	1.624
3	-128.797	-1.06	0.171	1.242	1.624
4	480.336	7.559	-7.006	1.549	2.209
5	-687.696	-9.221	7.255	0.243	0.185
6	480.186	7.585	-7.029	1.392	2.092
7	-687.546	-9.247	7.278	0.4	0.301
8	-159.842	-0.422	-0.419	2.409	2.687
9	-47.517	-1.24	0.668	-0.617	-0.294
10	468.716	7.455	-6.987	1.677	2.393
11	-676.076	-9.117	7.237	0.115	1.788e-004
12	-160.275	-0.426	-0.417	2.409	2.687
13	-47.084	-1.236	0.666	-0.617	-0.294
14	-130.914	-0.969	0.149	0.251	1.532
15	-128.51	-0.966	0.171	0.349	1.624

Efforts sign

Menu

Calculate

Report

Help

Add

Delete

Fig. 7. An example table of stresses

2. Information modes

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

2.1 Deflection and strain limits

This mode (Fig. 8) presents data concerning allowable limit values of the deflections in wooden structural members and limit values of strains/deformations in joints, as listed in Tables 15, 16 of SNiP II-25-80.

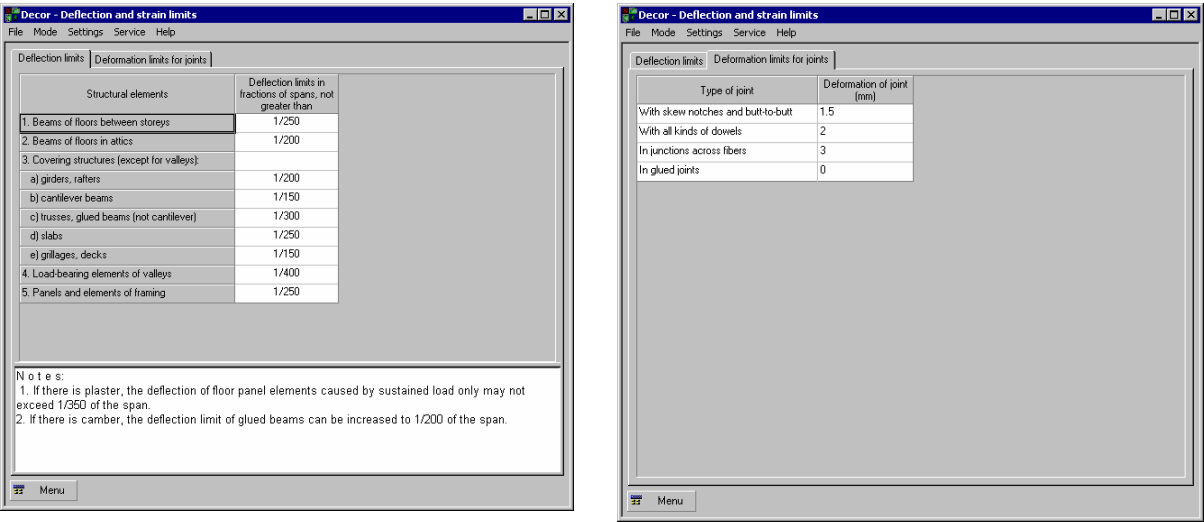


Fig. 8. Dialog boxes of the Deflection and strain limits mode

2.2 Densities

Information provided by this mode (Fig. 9) consists of the densities of wood and comes from Appendix 3 of SNiP II-25-80.

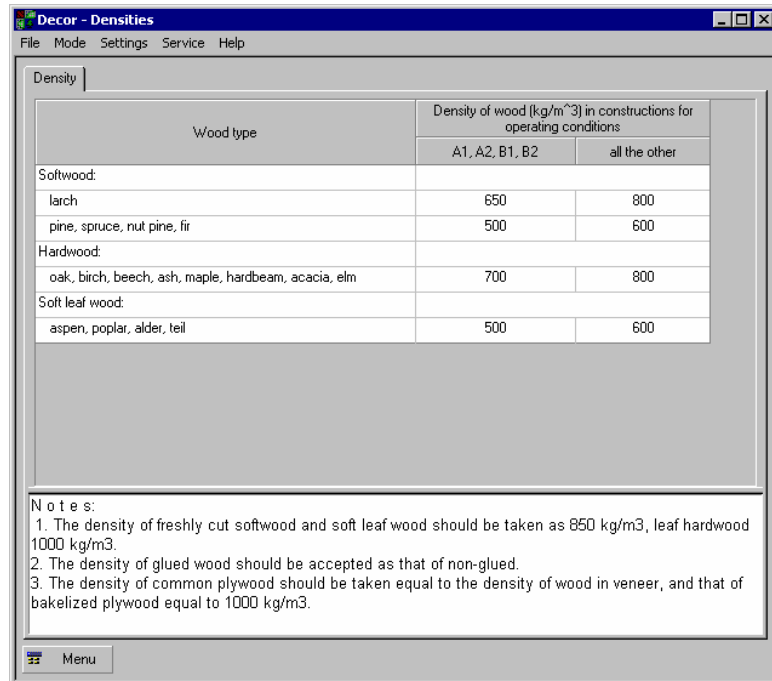


Fig. 9. The **Densities** dialog box

2.3 Range of timber

This mode provides information concerning the dimensions of edged and unedged softwood timber in compliance with GOST 24454-80 [2] (Fig. 10). Also, the **Timber for Glued Constructions** tab (Fig. 11) lets the user specify dimensions of timber before planing and then have the dimensions after planing calculated in compliance with GOST 7307-75* [3].

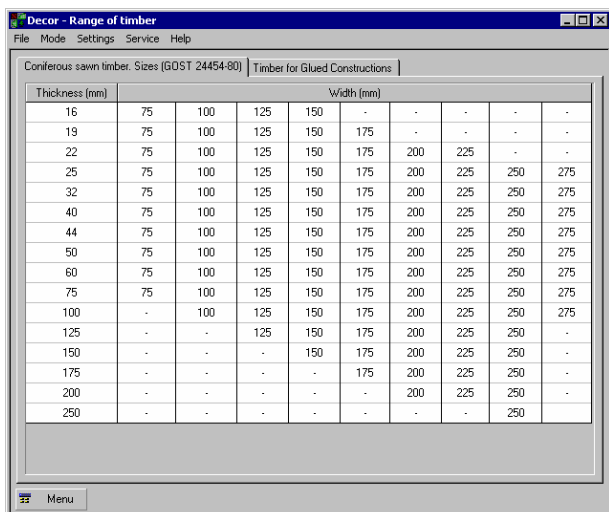


Fig. 10. The **Range of Timber** dialog box.
The **Softwood Timber** tab

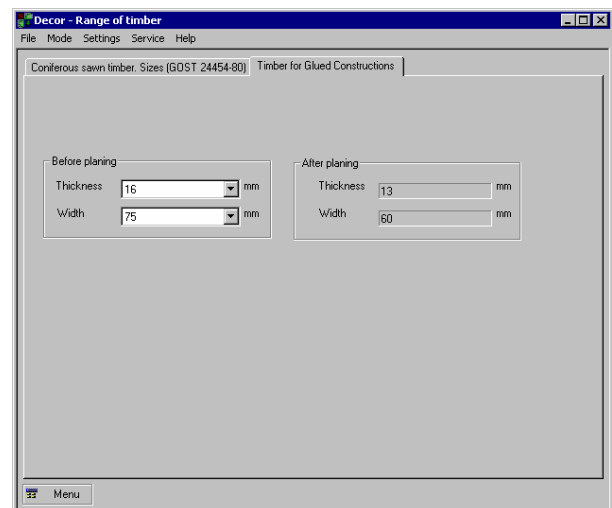


Fig. 11. The **Range of Timber** dialog box.
The **Timber for Glued Constructions** tab

2.4 Design strength

To determine the design strength of a member in compliance with Sec. 3.1, 3.2 of SNiP II-25-80, set the following parameters using appropriate lists of the **Design Strength** dialog box (Fig. 12):

- type of the stress and strain distribution;
- characteristics of the member;
- kind of wood;
- grade of wood.

Fig. 12. The **Design Strength** dialog box

Now use radio buttons to specify the type of section (glued or non-glued) and set the service factors (m_a , m_T , m_d , ...) by choosing their values from lists, or entering the values directly in the list fields if the factors are to differ from those recommended by the design codes (the lists contain all values presented in the respective tables of SNiP).

The value of the design strength will be displayed in the field under the respective name after you click the **Calculate** button.

2.5 Wood

This mode (Fig. 13) presents data concerning the maximum allowable humidity of wood in compliance with Table 1 of SNiP II-25-80. The rated humidity depends on given service conditions for the construction. Also, types and grades of glues required for glued wood are presented (in compliance with Table 2 of SNiP II-25-80).

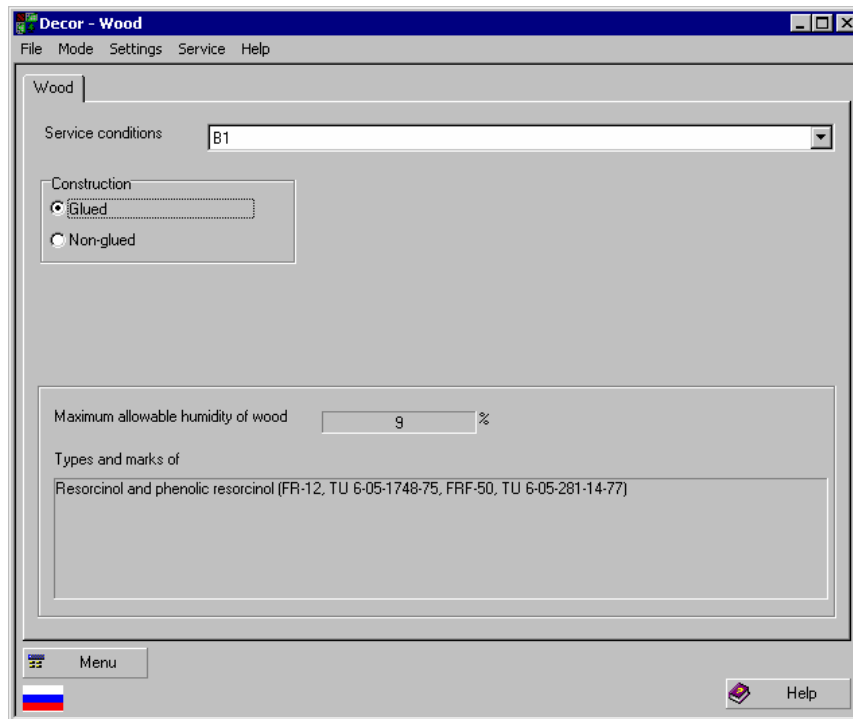


Fig. 13. The **Wood** dialog box

2.6 Limit slenderness

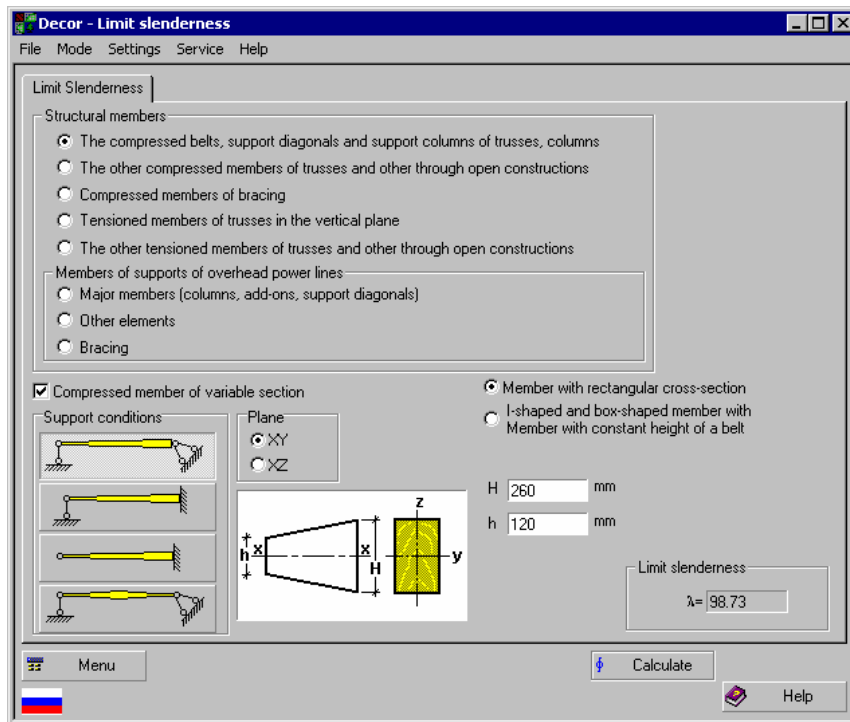


Fig. 14. The **Limit Slenderness** dialog box

The limit slenderness is calculated for various kinds of structural members on the basis of data from Table 14 of SNiP II-25-80. The dialog box of the mode (Fig. 14) contains: two groups of radio buttons which define the type of members (members of supports of overhead power lines make up a separate group); a group of radio buttons for choosing the section's plane; radio buttons to select the section's shape (rectangular, double tee, or box).

If the *Compressed member of variable section* checkbox is enabled, you should use the buttons below it to set the conditions of support. The limit slenderness value appears in the field of the respective name after you click the **Calculate** button (this value complies with Table 1 of SNiP and Table 1 of Appendix 4 of SNiP II-25-80).

3. Analysis

3.1 General management

Most modes intended for analysis include a number of common controls and techniques of management. These include groups of controls for specifying cross-sections, properties of wood, service factors and the like. Groups of this kind are described below.

3.1.1 Building cross-sections

The **DECOR** application provides the capability of analyzing wooden structural members of three section shapes: rectangular, double tee, and round. To choose a desired section shape and specify its sizes, use the **Section** group (Fig. 15).

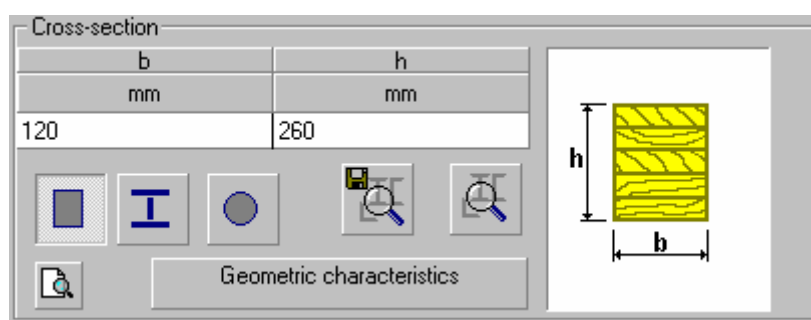


Fig. 15. Types of cross-sections

The button  provides access to a database of user-defined sections. The database is created gradually in the course of your working with the application. To save a newly created section in the database, use the button  clicking which opens the **User-defined Sections** dialog box. Specify a name for your section in this dialog (Fig. 16).

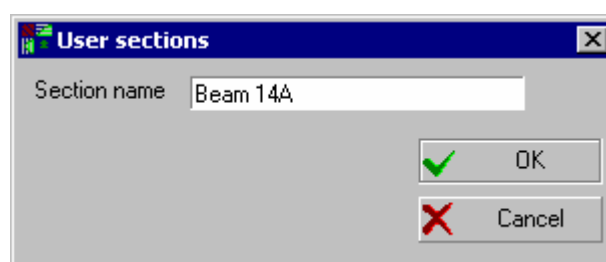


Fig. 16. The **User-defined Sections** dialog box

The application does not make any check of uniqueness of the names you use, so you have to do it yourself.

The geometry of a section specified can be checked by clicking the **Preview** button ; it opens a dialog box under the same name where a schematic of the section is presented (Fig. 17).

When you access the database of sections, a dialog box appears (Fig. 18) that contains a list of all sections stored in the database. The sections can be deleted from the database or renamed (using the respective buttons

Delete and **Rename**), previewed (**Preview** ) , or loaded into the application's environment for subsequent use. To do the latter action, select a line with the name of your desired section and click the **Apply** button.

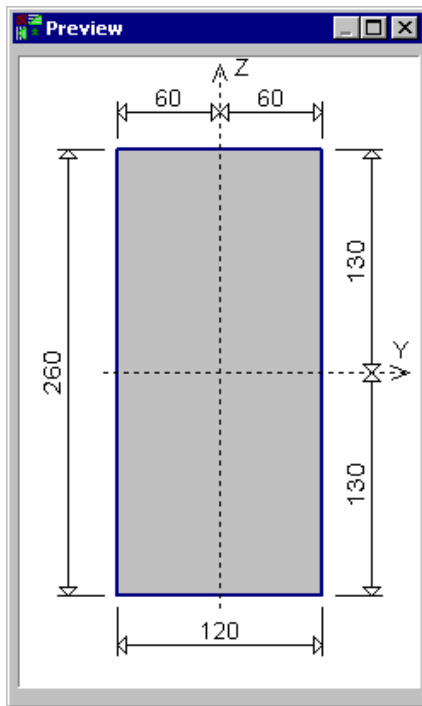


Fig. 17. The **Preview** dialog box

The **Geometric Characteristics** button located in the **Section** group calls up the **Section Properties** dialog box (Fig. 19) where geometric characteristics of the section (its area, its moments of inertia etc.) and a draft of the section with its sizes and axes are displayed.

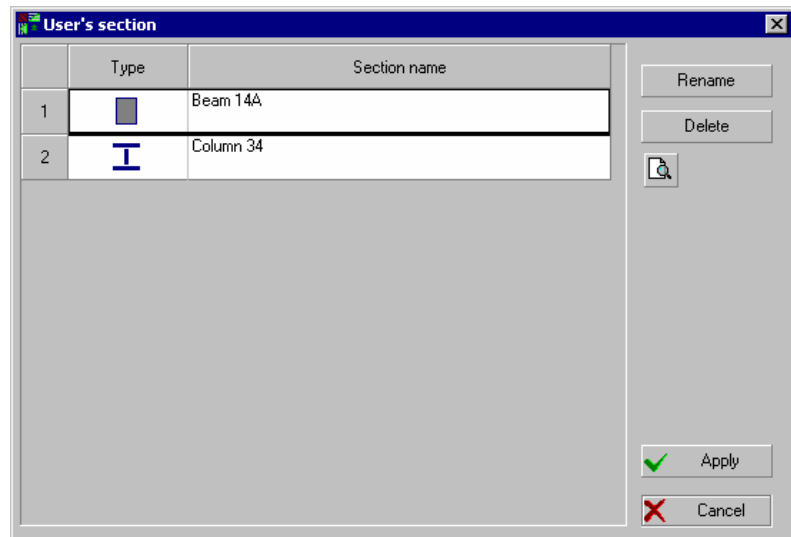


Fig. 18. A dialog box that lists user-defined cross-sections stored in the database

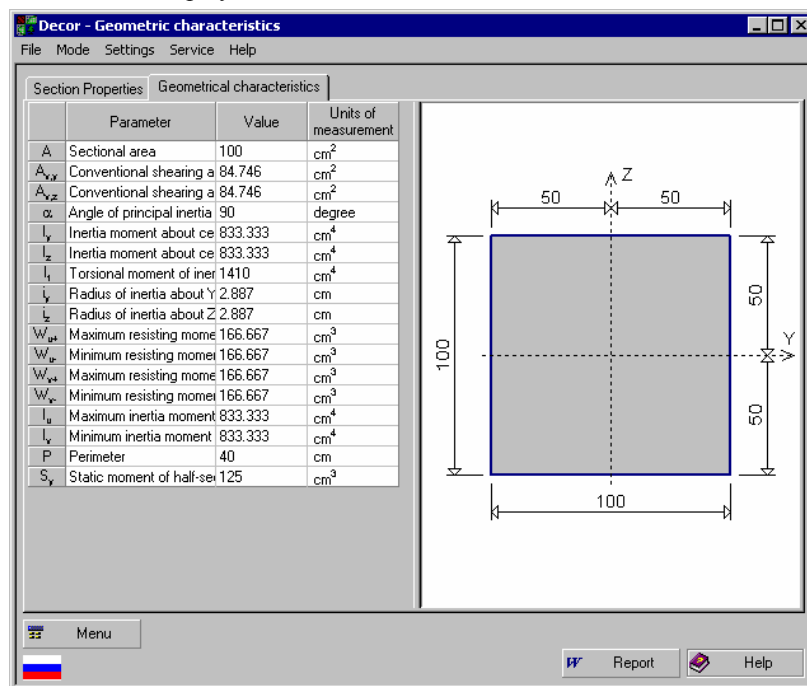


Fig. 19. The **Section Properties** dialog box

3.1.2 Properties of wood

Fig. 20. A group of controls used to specify properties of wood

The properties of wood are required by all analysis modes. To specify this information, you use, as a rule, a standard group of controls (Fig. 20). Properties like the kind of wood (pine, fir, oak, ...), the grade of wood, the density are selected from lists. The latter of the parameters is required only in modes where a self-weight load can be applied to a construction (such as a beam or a girder). Radio buttons are used to indicate whether a section is a glued one.

Most often, this group of controls contains a list for choosing or entering the safety factor for responsibility in compliance with GOST 27751-88 [4].

3.1.3 Service factors

To analyze a structural member or joint, one needs to know service factors (m_a , m_T , m_d , ...). To specify the service factors, choose appropriate values from lists (the lists contain all values defined in Sec. 3.2 and the respective tables of SNiP II-25-80) or enter ones directly in the list fields (Fig. 21).

Fig. 21. Service factors

Some of the factors may be absent in particular modes (or, sometimes, depending on the state of other controls). For example, non-glued sections do not require knowing the m_{sp} factor.

3.1.4 Creating a report

All working modes that do an analysis have a button entitled **Report**. Clicking this button starts the following actions, provided there are no mistakes in the source data:

- all analyzes/calculations are performed;
- a file is created in the RTF (Rich Text) format, that contains a list of source data and results of the analysis. Depending on what option is selected in the **Options** dialog box (whether **Full messages** or **Short messages**), 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 maximum) load-bearing ability utilization factors etc.;
- a Windows application is launched, associated with the RTF-type files. 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**.

3.2 Geometrical characteristics

This mode is used to calculate numerical geometrical characteristics of a section following rules described earlier (see Section 3.1.1). The source data are entered on the **Section Properties** tab (Fig. 22). The results are displayed on the **Geometrical Characteristics** tab in the form shown in Fig. 23.

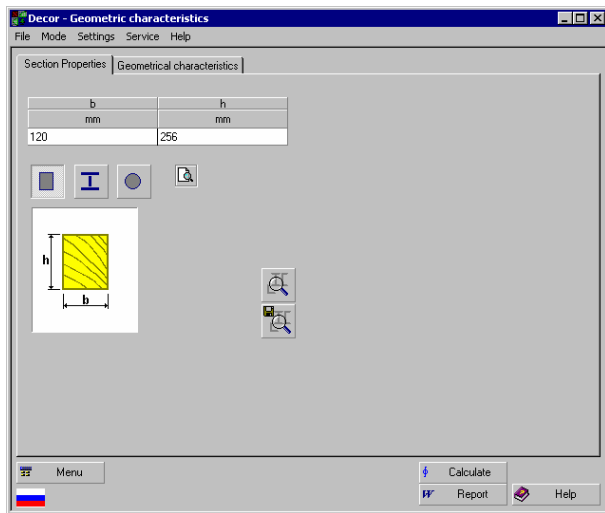


Fig. 22. The **Section Properties** tab of the **Geometrical Characteristics** mode

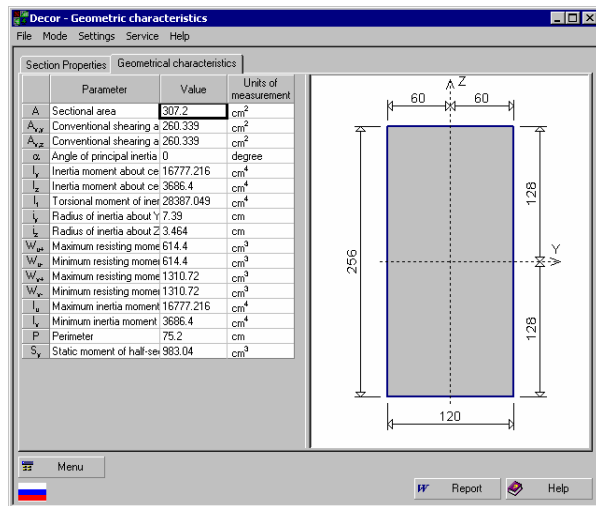


Fig. 23. The **Geometrical Characteristics** tab of the **Geometrical Characteristics** mode

3.3 Effective length

This mode implements recommendations of Sec. 4.21 of SNiP II-25-80 for calculation of effective length values. Launching this mode opens the **Construction Type** tab (Fig. 24) where you indicate what type of construction you want the effective length calculated for (two construction types are implemented: *Separately standing columns and stanchions* and *Intersection of a compressed member with an unloaded one*). Depending on the choice you make, the second tab of the mode changes its appearance (Fig. 25, 26).

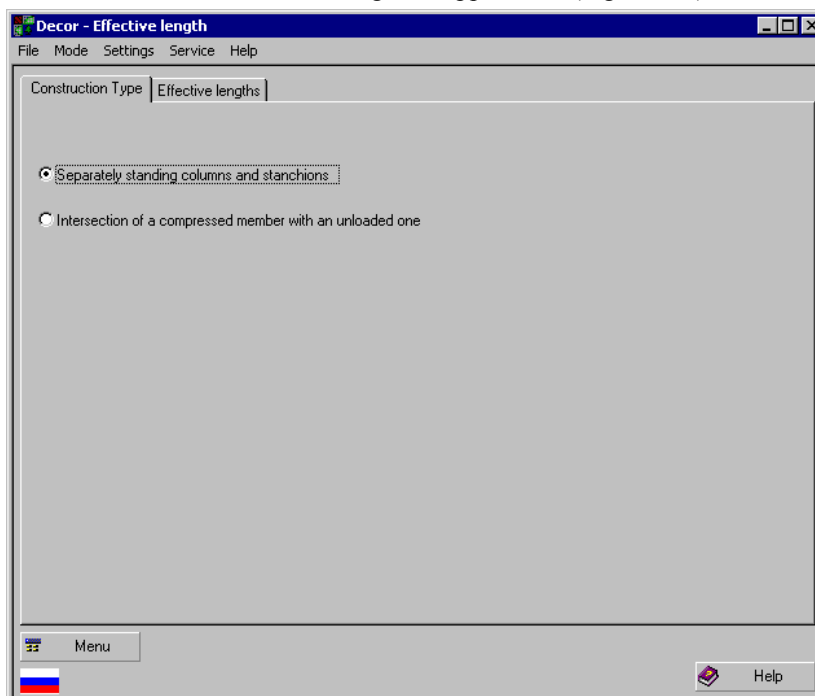


Fig. 24. The **Effective Length** dialog box. The **Construction Type** tab

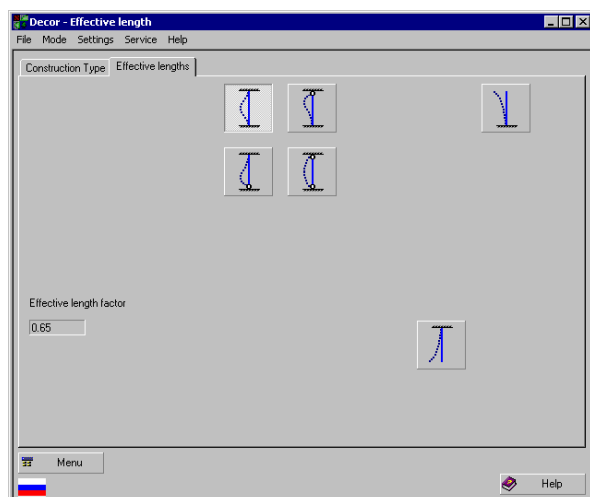


Fig. 25. The **Effective Length** dialog box. The **Effective Length** tab

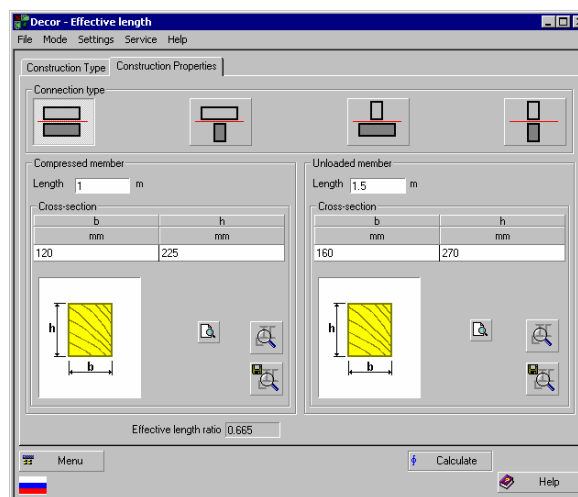


Fig. 26. The **Effective Length** dialog box. The **Construction Properties** tab

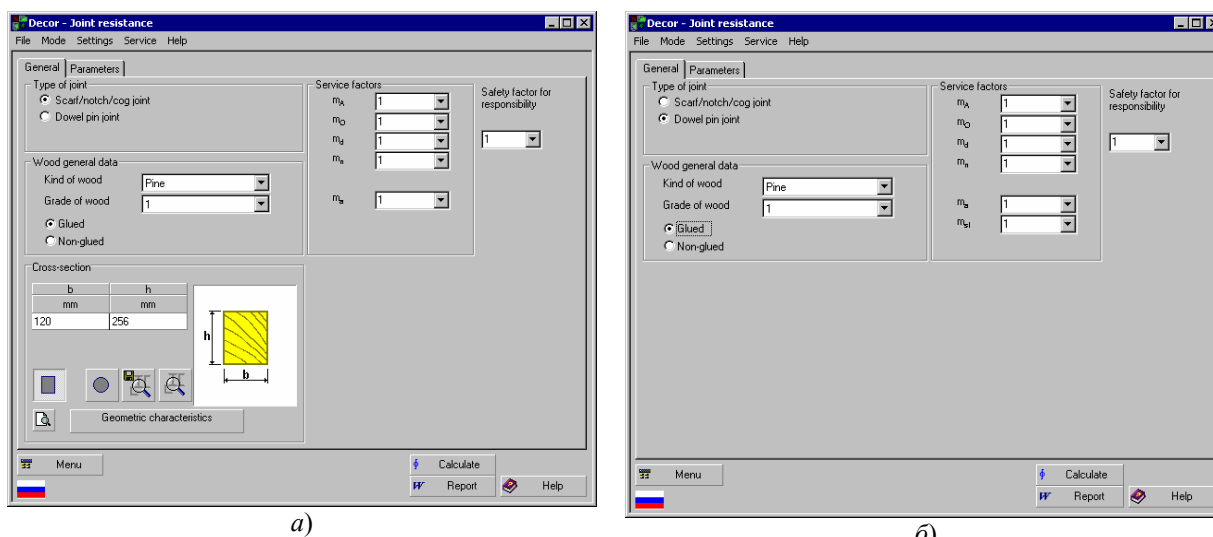
For *separately standing columns and stanchions*, the **Effective Length** tab has a group of buttons corresponding to particular fixation types. The result of the analysis as defined by Sec. 4.21 of SNiP II-25-80 (the ratio of the effective length to the geometric length of the member) will be displayed in the **Effective length ratio** after you click the appropriate button.

For *intersection of a compressed member with an unloaded one*, you use the controls on the **Construction Properties** tab to specify sizes of the sections of the members, lengths of the members, and the way they are connected. The effective length ratio is calculated as soon as you click the **Calculate** button.

3.4 Joint resistance

This mode is used to calculate the limitation utilization factors for scarf/notch/cog joints and dowel pin joints. The mode consists of two tabs. The **General** tab (Fig. 27) helps choose a joint type — either a scarf/notch/cog joint

or a dowel pin joint. The same tab contains standard groups of controls for entering wood data and service factors. For the case of a scarf/notch/cog joint, information about the section also needs to be provided.



a)

b)

Fig. 27. The **Joint Resistance** dialog box. The **General** tab

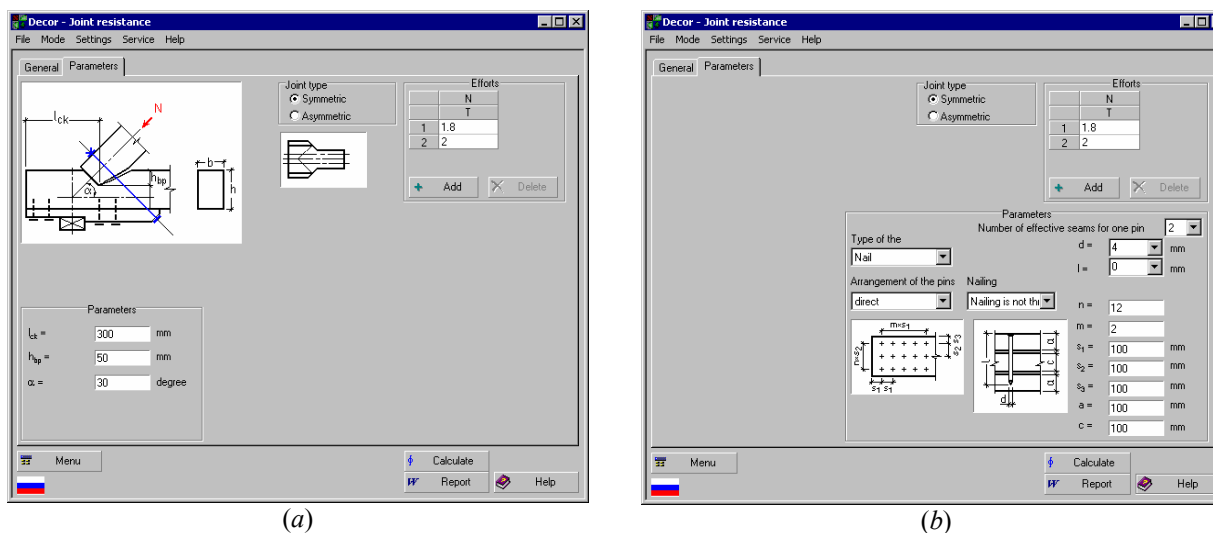
Depending on the type of a joint to be analyzed, the **Parameters** tab is used to specify the following data:

(a) for a scarf/notch/cog joint (Fig. 28,a):

- parameters of the joint;
- type of the joint (either symmetric or asymmetric);
- forces.

(b) for a dowel pin joint (Fig. 28,b):

- type of the pin (nail, steel, aluminum, fiberglass, oak);
- arrangement of the pins (direct, staggered, oblique);
- number of effective seams for one pin;
- diameter of the pin;
- number of the pins and their geometric arrangement;
- forces.



(a)

(b)

Fig. 28. The **Joint Resistance** dialog box. The **Parameters** tab

When specifying the forces, you can enter multiple values for the longitudinal force (which correspond to multiple mutually exclusive loadings).

The scarf/notch/cog joints are analyzed for:

- strength based on conditions of bearing — Sec. 5.2 of SNiP II-25-80;
- strength based on conditions of shearing — Sec. 5.2 of SNiP II-25-80.

The dowel pin joints are analyzed by the following criteria:

- load-bearing ability for bearing of an extreme member — Sec. 5.13-5.15 of SNiP II-25-80;
- load-bearing ability for bearing of a middle member — Sec. 5.13-5.15 of SNiP II-25-80;

- bending of a steel pin — Sec. 5.13-5.15 of SNiP II-25-80;
- bending of a nail — Sec. 5.13-5.15 of SNiP II-25-80;
- bending of a fiberglass pin — Sec. 5.13-5.15 of SNiP II-25-80;
- bending of an aluminum pin — Sec. 5.13-5.15 of SNiP II-25-80;
- load-bearing ability for bearing of a thicker member — Sec. 5.13-5.15 of SNiP II-25-80;
- load-bearing ability for bearing of a thinner member — Sec. 5.13-5.15 of SNiP II-25-80;
- bending of an oak pin — Sec. 5.13-5.15 of SNiP II-25-80.

3.5 Section resistance

This mode calculates the load-bearing ability of any cross-section that the application can work with. Generally, the analysis involves the action of a longitudinal force, bending moments, and shear forces in the principal planes of inertia. The mode implements the whole set of checks for strength, stability, and limit slenderness in compliance with Section 4 of SNiP II-25-80.

The section is analyzed by the following criteria:

- slenderness of the member in the XoY plane — Sec. 4.4 of SNiP II-25-80;
- slenderness of the member in the XoZ plane — Sec. 4.4 of SNiP II-25-80;
- strength of the member under a tensioning longitudinal force — Sec. 4.1 of SNiP II-25-80;
- strength of the member under a compressing longitudinal force — Sec. 4.2 of SNiP II-25-80;
- stability in the XoZ plane under a longitudinal force — Sec. 4.2 of SNiP II-25-80;
- stability in the XoY plane under a longitudinal force — Sec. 4.2 of SNiP II-25-80;
- strength of the member under bending moment M_y — Sec. 4.9 of SNiP II-25-80;
- strength of the member under bending moment M_z — Sec. 4.9 of SNiP II-25-80;
- strength under the combined action of M_y and M_z — Sec. 4.12 of SNiP II-25-80;
- strength under the combined action of a longitudinal force and bending moment M_z — Sec. 4.16, 4.17 of SNiP II-25-80;
- strength under the combined action of a longitudinal force and bending moment M_y — Sec. 4.16, 4.17 of SNiP II-25-80;
- strength under lateral force Q_z — Sec. 4.10 of SNiP II-25-80;
- strength under lateral force Q_y — Sec. 4.10 of SNiP II-25-80;
- stability of a planar mode of deformation — Sec. 4.14, 4.15, 4.18 of SNiP II-25-80.

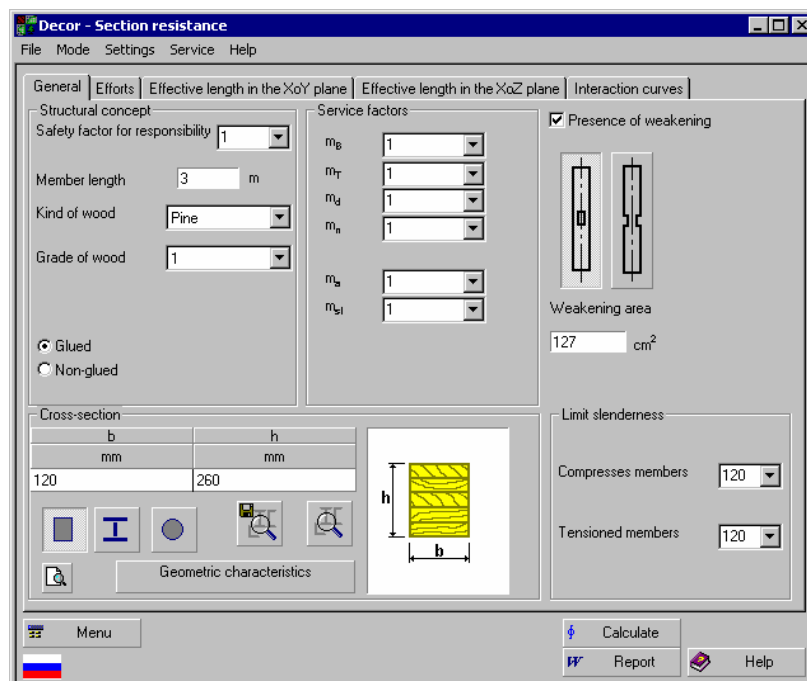


Fig. 29. The Section Resistance dialog box. The General tab

The dialog box of the mode consists of five tabs: **General**, **Stresses**, **Effective Length in XoY Plane**, **Effective Length in XoZ Plane**, **Interaction Curves**. The first four tabs are used to enter source data, and the fifth one displays results of the analysis.

The **General** tab (Fig. 29) is used to specify data about the cross-section (see Section 3.1.1), wood (see Section 3.1.2), service factors (see Section 3.1.3), and limit slenderness values. Also, the behavior of weakened

sections can be analyzed (the **Presence of weakening** checkbox). If the latter checkbox is enabled, a type of weakening and its area should be specified.

The **Effective Length in XoY (XoZ) Plane** tabs are replicas of the **Construction Properties** tab for **Separately standing columns and stanchions** from the **Effective Length** mode. They suggest 5 options of end fixations for a compressed bar member, which differ by combinations of boundary conditions (a free end, a hinge or simple support, a stiff clamp). Working with this dialog is described in Section 3.3. Unlike the **Effective Length**

mode, this dialog has the  button (Fig. 30) clicking which lets you enter any value of the effective length factor and confirm your choice with the **Apply** button. In all other cases this field does not let any data be entered.

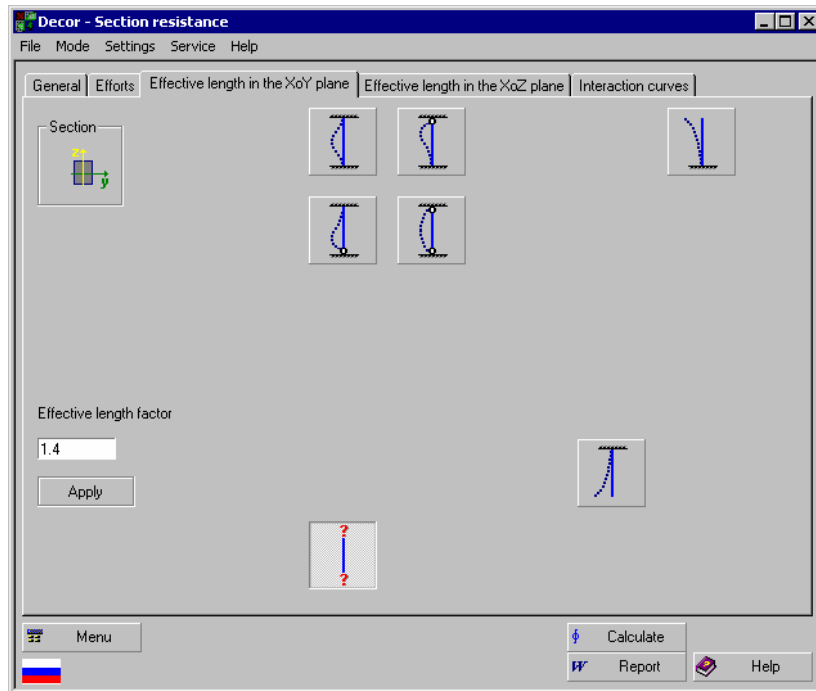


Fig. 30. The **Section Resistance** dialog box. The **Effective Length** tab

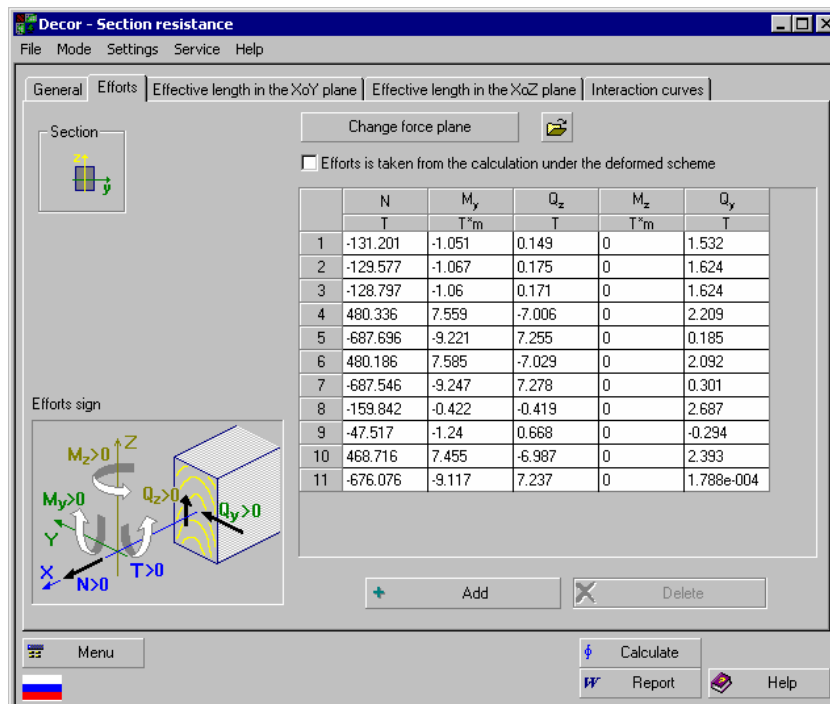


Fig. 31. The **Section Resistance** dialog box. The **Stresses** tab

The **Stresses** tab (Fig. 31) is used to enter stresses that act in the cross-section of the member. A schematic of the cross-section shown in it displays principal axes of inertia and positive directions of the stresses. The tab contains a table where you specify the stresses caused in the section by one or more loadings. The number of rows in

the table conforms to the number of the loadings. The table can be filled by importing data from **SCAD** which describe design stress combinations (DSC). The file with the .RSU extension is created in the **Element Information** mode of the **SCAD** software and imported into **DECOR** by clicking the  button above the table.

In this mode, the stresses acting in the section of the member are specified on the basis of a static analysis performed externally, therefore you need to indicate what design model has been used to calculate the stresses — either a second-order model (a nonlinear analysis where the deformed shape is used) or a first-order model (a linear analysis where the original shape is used). If the analysis has been nonlinear, the **Stresses are calculated by second-order analysis** checkbox should be turned on.

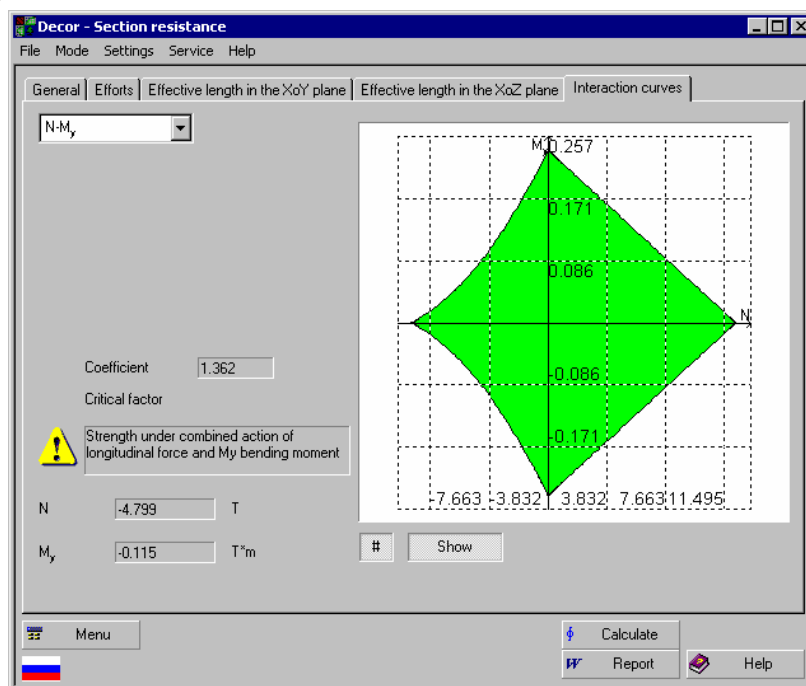


Fig. 32. The **Section Resistance** dialog box. The **Interaction Curves** tab

The **Section Resistance** mode suggests a capability of switching the plane of loading (the **Switch Plane of Loading** button). Clicking this button will replace M_y by M_z and Q_y by Q_z for each loading in the table of stresses.

The **Interaction Curves** tab (Fig. 32) is used to build curves which bound a load-bearing ability area for the section under various couples of stresses that may act upon it.

The interaction curves (Fig. 32) surround the coordinate origin by a closed curve that confines points of conditionally admissible couples of stresses. A couple of stresses is deemed admissible when $K_{\max} \leq 1$. All the other stresses are assumed to be zero.

The mouse pointer can be used to explore the stress variation area shown in the picture. Each position of the pointer conforms to a certain couple of numerical values of the stresses; the values are displayed in appropriate fields.

The limit slenderness factors do not depend on the stresses, therefore they are not calculated for the interaction curves.

At the same time, a maximum value of the limitation utilization factor that corresponds to those stresses and a type of check that has produced this maximum are displayed. If the pointer is placed onto a point where $K_{\max} > 1$, a warning sign  appears.

Clicking the right mouse button lets you browse a list of checks performed and values of analysis criteria for the set of stresses that conforms to the position of the pointer on the interaction curve.

3.5.1 Limitations of the current version

1. The design modulus of the weakened section is assumed to be equal to the gross modulus of section (we do not have any data about the shape of the weakening, therefore we cannot calculate the moment of inertia of the weakened section).
2. The k_F factor for the analysis of stability of a planar mode of deformation is assumed to be 1.13.
3. The power of n in (33) is assumed to be 1.
4. The analysis of stability of a planar mode of deformation for members in combined bending and compression is performed for a rectangular section only, because formula (33) makes use of the ϕ_M factor calculated by (23). Formula (23) is intended for rectangles only.
5. In the analysis of stability of a planar mode of deformation for beams of a variable rectangular cross-section, the k_{GM} factor is taken equal to 1 because, when the number of bracings from the plane is $n \geq 4$,

the SNIIP document (Sec. 4.14) requires that $k_{GM}=1$, and for other values of n it refers to Table 2 of Appendix 4 where nothing is said about what to do with an arbitrary curve of moments.

6. When analyzing the strength of eccentrically tensioned and tensioned-bent members by formula (28) of SNIIP II-25-80, the value of M_d is calculated by (29), (30).

3.6 Continuous girders

This mode is used to analyze continuous girders of a round or rectangular section. The dialog box of the mode (Fig. 33) consists of two tabs: **General** and **Loads**.

The **General** tab is used to specify the number of spans and their lengths. Here you also specify (according to rules described in Section 3.1) information about the section, wood, and service factors. The angle of slope of the roof can be specified here, too (to take into account the pitch component of the stresses). If the second limit state analysis is required, enable the **Limitation of deflection** checkbox and choose (or enter) in the respective drop-down list a maximum allowable ratio of the greatest deflection to the length of the span.

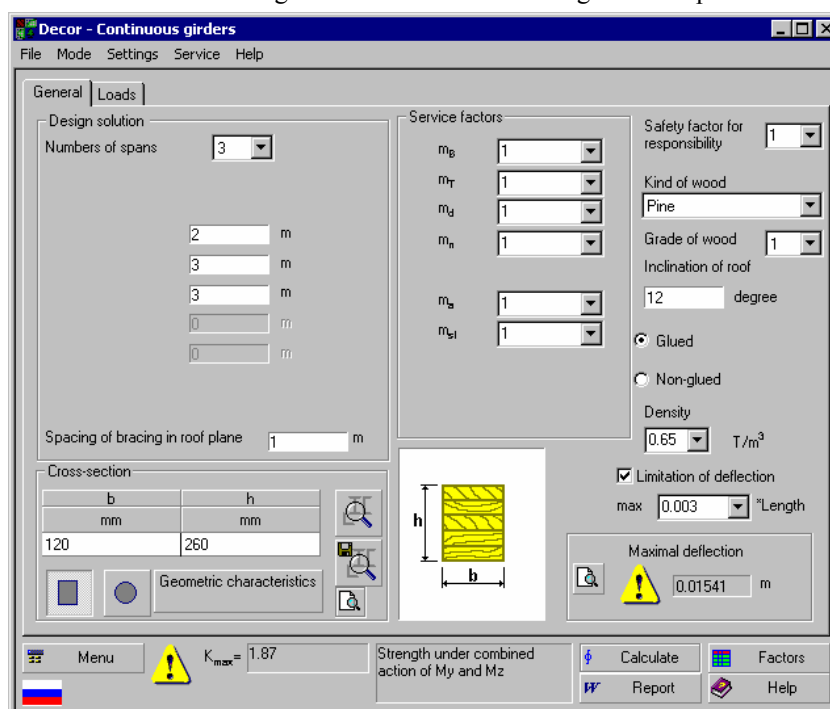


Fig. 33. The Continuous Girders dialog box. The **General** tab

The **Loads** tab (Fig. 34) is used to define loads applied to the girder. The application can do the analysis of multiple loading patterns, each one containing a few loads.

To enter a new loading (including the very first one), follow these steps:

- click the **Create** button in the **Loadings** group;
- choose a loading type (permanent, temporary but sustained, short-term, snow, or wind), which will define the combination factors used, according to SNIIP 2.01.07-85*, to multiply the loads from this loading by when including them in combinations of loads;
- click a button that depicts the desired type of load;
- enter numerical parameters of the load;
- click the **Add** button.

Multiple load components can be specified for each loading. *Design* values of the loads are to be specified.

Depending on the load type, its parameters 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;
- 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.

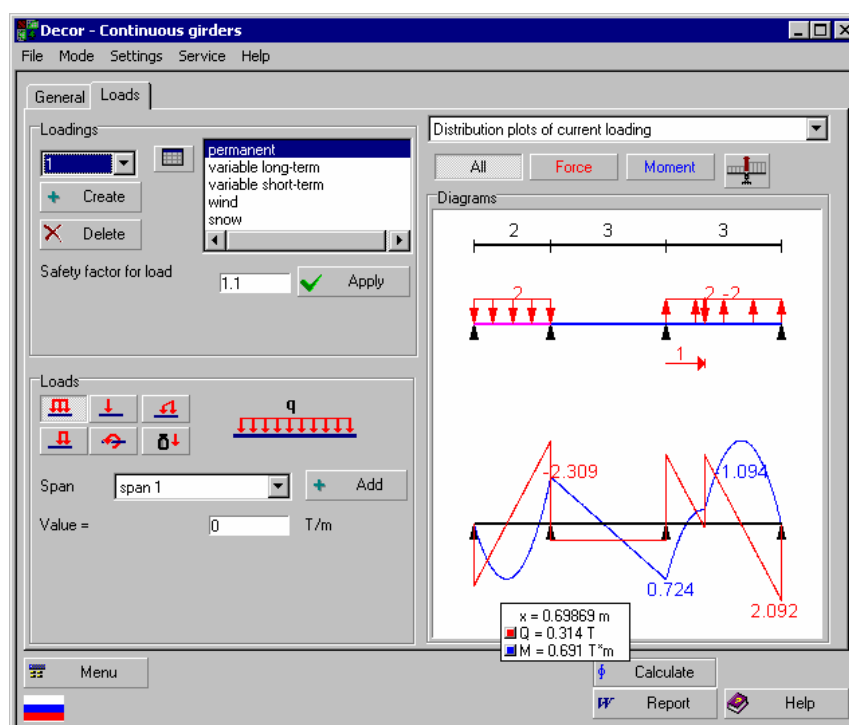


Fig. 34. The **Continuous Girders** dialog box. The **Loads** tab

For each load (except for self-weight) you need to specify a span to which it is applied (use the **Span** drop-down list to do it).

To delete a loading (not a particular load from it), use the **Delete** button.

To start specifying the next loading, click the **Create** button again; clicking it will increase the number of existing loadings by one. If you need to review or revise data for any of the previously defined loadings, just choose its No. in the **Loading** list.

Clicking the **Add** button will display the current state of the loading in the **Loads** area; a schematic will display the curves of the moments and shear forces together (Fig. 34).

Having defined all loadings, you can review the envelopes of the moments and their respective shear forces, or the envelopes of the shear forces and their respective bending moments. To choose a criterion (maximum/minimum moment, maximum/minimum shear force), use a drop-down list. The envelopes comply with requirements of SNiP 2.01.07-85* "Loads and actions" [5].

The **All**, **Force**, and **Moment** buttons let you choose a curve display mode — the moment and the shear force curves together, only the shear forces, or only the bending moments. Also, clicking the  button will display calculated values of the support reactions (Fig. 35).

Support reactions				
	Force in support 1	Force in support 2	Force in support 3	Force in support 4
	T	T	T	T
by criterion M_{max}	0	2.396	-0.216	0
by criterion M_{min}	0	2.396	-0.216	0
by criterion Q_{max}	1.784	2.396	-0.216	-0.036
by criterion Q_{min}	1.784	2.396	-0.216	-0.036

Fig. 35. The **Support Reactions** dialog box

As you move the mouse pointer across the curve display area, the screen shows values of the moment and the shear force in a particular section that conforms to the position of the pointer.

If you need a second limit state analysis, check whether the safety factor for load is specified correctly for each loading, because the calculation of the deflections is based on nominal values of the loads.

If you need to change the value of a load or delete a load remaining within one loading, use the table of loads (the  button in the **Loadings** group). The **Table of Loads** dialog box (Fig. 36) that opens by clicking this button will display the load's type, its value, and its position. Changes made to the parameters of the loads will be remembered after you exit the dialog by clicking the **OK** button.

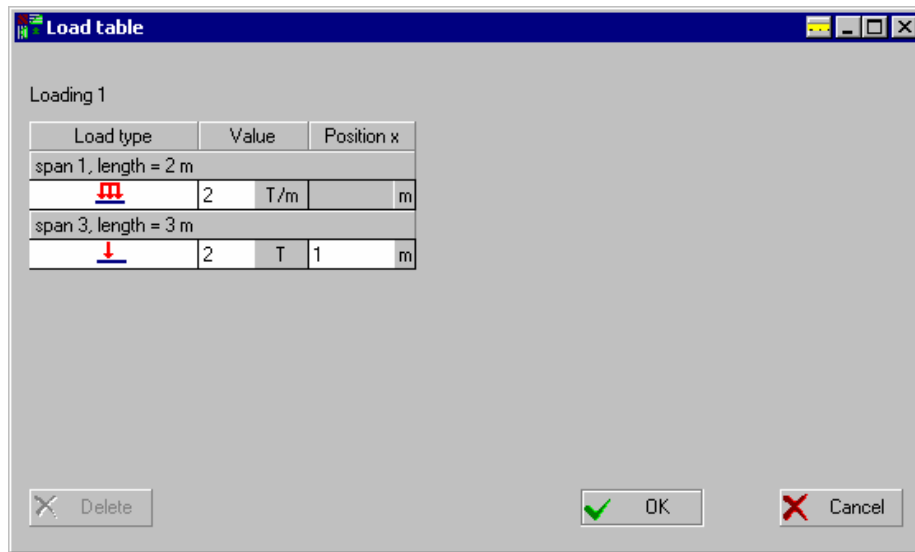


Fig. 36. The **Table of Loads** dialog box

3.6.1 Calculation

To perform the calculation of the load-bearing ability utilization factors, click the **Calculate** button. The following criteria are used to check the girders:

- strength of the member under bending moment M_y — Sec. 4.9 of SNiP II-25-80;
- strength of the member under bending moment M_z — Sec. 4.9 of SNiP II-25-80;
- strength of the member under shear force Q_z — Sec. 4.10 of SNiP II-25-80;
- strength of the member under shear force Q_y — Sec. 4.10 of SNiP II-25-80;
- strength under combined action of M_y and M_z — Sec. 4.12 of SNiP II-25-80;
- stability of a planar mode of deformation — Sec. 4.14, 4.15, 4.18 of SNiP II-25-80;
- deflection — Sec. 4.33 of SNiP II-25-80.

If the **Limitation of deflection** checkbox is enabled on the **General** tab, an envelope of the deflections will be calculated. The maximum value of the deflection will be displayed in the **Maximum deflection** group on the

General tab. Using the  button in the same group, you can call up a dialog box where curves of deflections are displayed (Fig. 37).

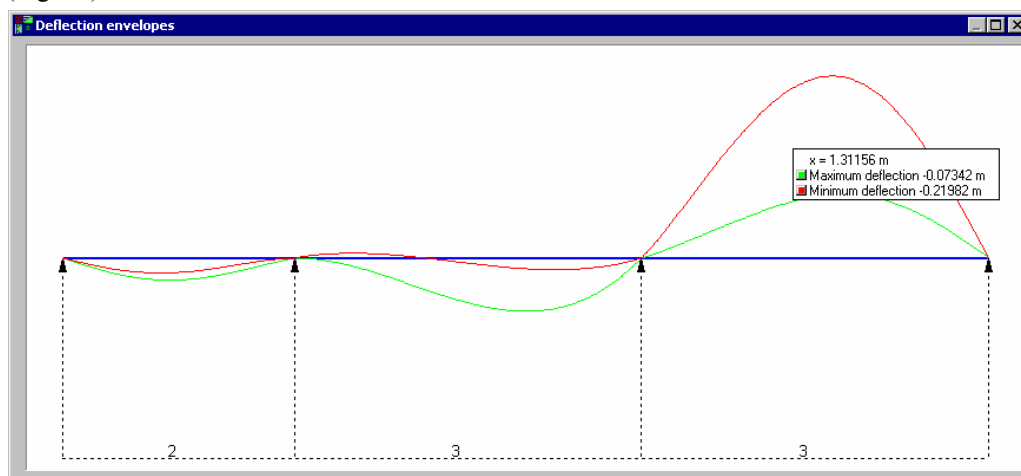


Fig. 37. The **Deflection Envelope** dialog box

Note that if the pitch of the roof is not zero, girders are bent in two planes, therefore the deformed axis of a girder is a spatial curve. Seeing that the pitch of the roof is usually not too big, the **Deflection Envelope** displays the deflections approximately, without allowing for the pitch. However, both the extreme value of the deflection and the respective factor are calculated precisely.

3.6.2 Limitations of the current version

Formula (50) is not used to calculate the maximum deflection because girders are assumed to have a constant height, and Table 3 of Appendix 4 requires that $k=1$, $c=0$ (the first row in the table is the most frequently used case).

3.7 Suspended spans

This mode is similar to the **Continuous Girders** mode. The differences are that the **General** tab (Fig. 38) requires only the length of a standard span specified (all standard spans are assumed to have the same length), and you can specify the location of hinges on the basis of either an equal-moment or equal-deflection model. If the **other** option is selected, you need to specify the distance from the support to the hinge.

Note that this mode can be used to analyze simply supported girders by specifying the zero distance between the support and the hinge.

Only two types of loads can be specified for the suspended spans: a uniformly distributed load on all spans, or a self-weight load.

The spans are checked for the following:

- strength of the member under bending moment M_y — Sec. 4.9 of SNiP II-25-80;
- strength of the member under bending moment M_z — Sec. 4.9 of SNiP II-25-80;
- strength of the member under shear force Q_z — Sec. 4.10 of SNiP II-25-80;
- strength of the member under shear force Q_y — Sec. 4.10 of SNiP II-25-80;
- strength under combined action of M_y and M_z — Sec. 4.12 of SNiP II-25-80;
- stability of a planar mode of deformation — Sec. 4.14, 4.15, 4.18 of SNiP II-25-80;
- deflection — Sec. 4.33 of SNiP II-25-80.

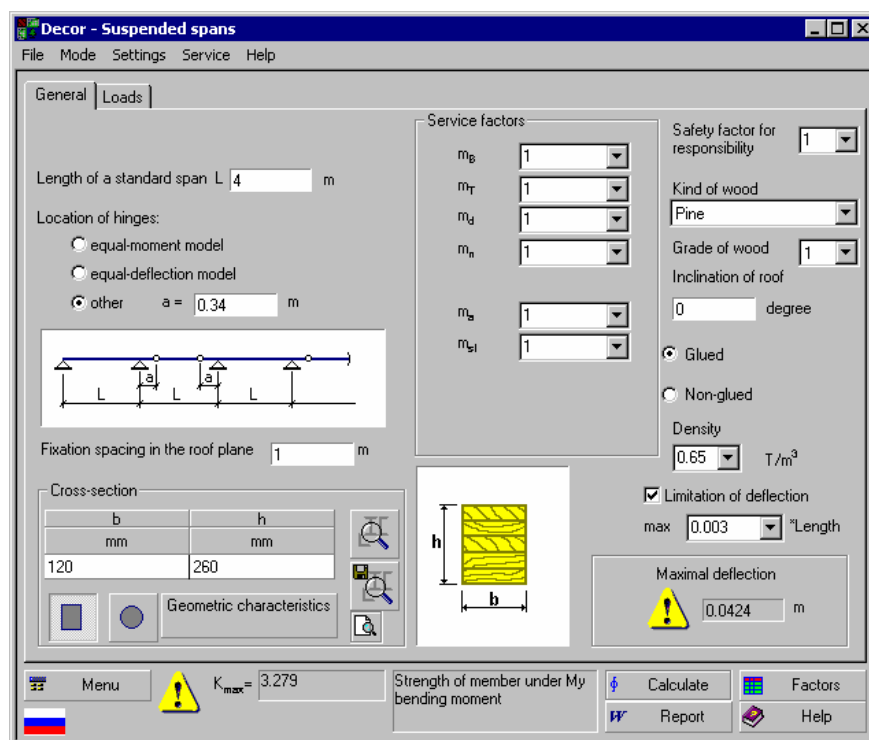


Fig. 38. The **Suspended Spans** dialog box. The **General** tab

3.7.1 Limitations of the current version

Formula (50) is not used to calculate the maximum deflection because spans are assumed to have a constant height, and Table 3 of Appendix 4 requires that $k=1$, $c=0$ (the first row in the table is the most frequently used case).

The analysis is based on the assumption that the extreme span has a recommended (see [7]) length (~ 0.85 of the standard span length). Maximum values of the bending moments, the shear forces, and the deflections cannot appear in the extreme span, so it is excluded from the analysis.

3.8 Beams

Use this multiple-tab dialog box (Fig. 39) to do a check of a usual or gable beam. The dialog consists of three tabs: **General**, **Fixations**, **Loads**.

The mode is similar to **Continuous Girders**. The differences are that only one length is specified for a beam and a gable beam can be analyzed if the section is rectangular (the height of the section in the middle of the span should be specified additionally).

The **Fixations** tab (Fig. 40) contains two groups of buttons to choose a system of fixations for the beam in and out of its plane of bending. The choice is independent in each group; just depress a button that depicts a desired method of fixation. If you choose the last option in the group that defines the fixations out of the bending plane, a field will open to enter the number of segments the span of the beam is to be divided into.

To check the correctness of the source data entered, use the table of constraints.

Fig. 39. The **Beams** dialog box. The **General** tab

	Left	Right
Displacement along Y		
Displacement along Z	Fixed	Fixed
Rotation about Y	Fixed	
Rotation about Z		

Fig. 40. The **Beams** dialog box. The **Fixations** tab

Specifying these loads is absolutely similar to specifying loads upon continuous girders (No. of a span the load is applied to is not required).

The beams are checked for the following:

- strength of the member under bending moment — Sec. 4.9 of SNiP II-25-80;
- strength under a shear force — Sec. 4.10 of SNiP II-25-80;
- stability of a planar mode of deformation — Sec. 4.14, 4.15, 4.18 of SNiP II-25-80;

- deflection — Sec. 4.33 of SNiP II-25-80.

3.8.1 Limitations of the current version

When calculating the maximum deflection in formula (50) for beams of a variable rectangular section, factors k and c are calculated according to row 4 of Table 3 of Appendix 4, i.e.:

$$k=0.15+0.85\beta;$$

$$c=15.4+3.8\beta.$$

Row 5 of Table 3 of Appendix 4 is used for beams of a variable double tee section, i.e.:

$$k=0.4+0.6\beta;$$

$$c=(45.3+6.9\beta)\gamma.$$

3.9 Columns

This mode is used to do checks of stanchions or columns. The whole set of checks of strength, stability, and limit slenderness is implemented in compliance with Section 4 of SNiP II-25-80. A planar pattern of loading is assumed.

The dialog box of the **Columns** mode consists of four tabs: **General**, **Stresses**, **Effective Length in XOY Plane**, **Effective Length in XOZ Plane**.

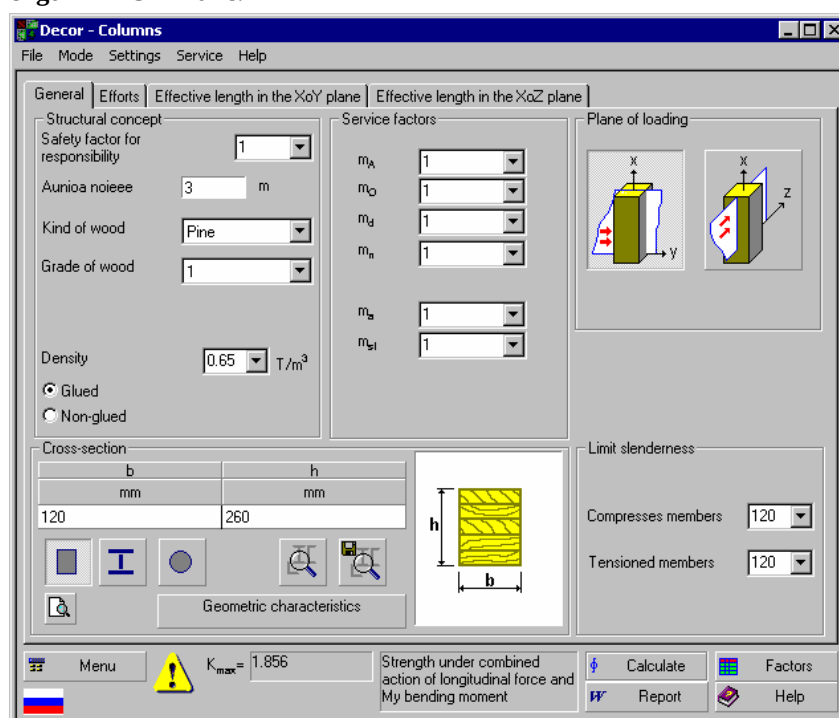


Fig. 41. The **Columns** dialog box. The **General** tab

The **General** tab (Fig. 41) contains an edit field for entering the column height and two buttons for choosing a plane of loading (an orientation of the deformation plane). The same tab is used to specify (according to standard rules from Section 3.1) data about the section, wood, and service factors. Also, you need to specify (by entering or choosing from a drop-down list) limit slenderness values (for compressed and tensioned members).

All stresses are specified together for each loading on the **Stresses** tab (Fig. 42).

Note that all loads act either in the XoY plane or in the XoZ plane (the X axis is oriented along the member's axis). To choose a plane, use the **General** tab.

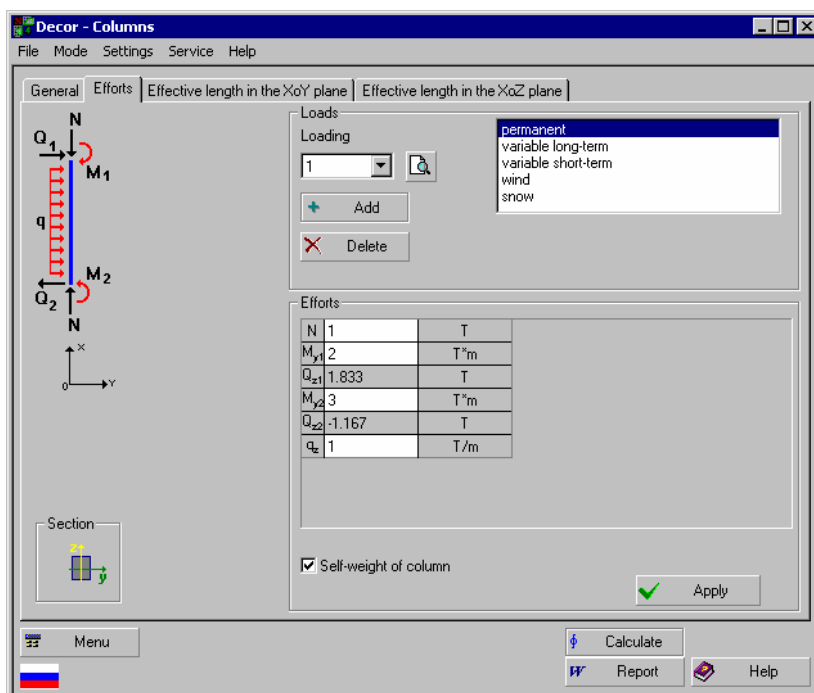


Fig. 42. The **Columns** dialog box. The **Stresses** tab

Clicking the  button opens the **Preview** dialog that displays curves of the N , M_y , and Q_z stresses (Fig. 43).

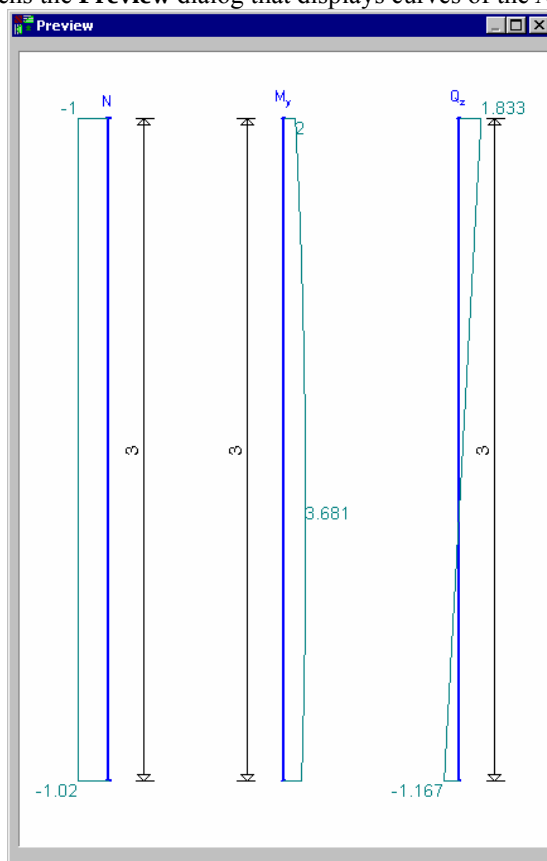


Fig. 43. The **Preview** dialog box

Tabs on which to specify the effective length are similar to their counterparts in the **Section Resistance** mode (see Section 3.5).

The columns and stanchions are checked for the following:

- slenderness in the XoY plane — Sec. 4.4 of SNiP II-25-80;
- slenderness in the XoZ plane — Sec. 4.4 of SNiP II-25-80;
- strength of the member under a tensioning longitudinal force — Sec. 4.1 of SNiP II-25-80;
- strength of the member under a compressing longitudinal force — Sec. 4.2 of SNiP II-25-80;
- stability in the XoZ plane under a longitudinal force — Sec. 4.2 of SNiP II-25-80;

- stability in the XoY plane under a longitudinal force — Sec. 4.2 of SNiP II-25-80;
- strength of the member under bending moment M_y — Sec. 4.9 of SNiP II-25-80;
- strength of the member under bending moment M_z — Sec. 4.9 of SNiP II-25-80;
- strength under combined action of M_y and M_z — Sec. 4.12 of SNiP II-25-80;
- strength under combined action of longitudinal force and bending moment M_z — Sec. 4.16, 4.17 of SNiP II-25-80;
- strength under combined action of longitudinal force and bending moment M_y — Sec. 4.16, 4.17 of SNiP II-25-80;
- strength under shear force Q_z — Sec. 4.10 of SNiP II-25-80;
- strength under shear force Q_y — Sec. 4.10 of SNiP II-25-80;
- stability of a planar mode of deformation — Sec. 4.14, 4.15, 4.18 of SNiP II-25-80.

3.9.1 Limitations of the current version

1. The k_F factor for the analysis of stability of a planar mode of deformation is assumed to be 1.13.
2. The power of n in (33) is assumed to be 1.
3. The analysis of stability of a planar mode of deformation for members in combined bending and compression is performed for a rectangular section only, because formula (33) makes use of the φ_M factor calculated by (23). Formula (23) is intended for rectangles only.
4. In the analysis of stability of a planar mode of deformation for beams of a variable rectangular cross-section, the k_{GM} factor is taken equal to 1 because, when the number of bracings from the plane is $n \geq 4$, the SNiP document (Sec. 4.14) requires that $k_{GM}=1$, and for other values of n it refers to Table 2 of Appendix 4 where nothing is said about what to do with an arbitrary curve of moments.
5. When analyzing the strength of eccentrically tensioned and tensioned-bent members by formula (28) of SNiP II-25-80, the value of M_d is calculated by formulas (29), (30).

3.10 Trusses

This mode implements all required checks of truss elements for strength and stability; it also checks the slenderness. The work begins with calculating design stress values caused by given external loads for structural designs most frequently used in practice.

This mode is oriented at the analysis of wooden trusses. To analyze combined metal-and-wood trusses, the following trick can be used. First, you need to specify a configuration of the truss and a load upon it. All admissible trusses are statically determinate, therefore the stresses in their members do not depend on the material. Every wooden member can be checked for compliance with SNiP II-25-80 using **DECOR**, and steel members can be checked using the **KRISTALL** software (see [6]).

Source data for the mode are specified on four tabs: **General**, **Materials**, **Sections**, and **Loads**.

The **General** tab (Fig. 44) contains a group of buttons to choose a configuration of the truss. All trusses are statically determinate and are assumed to be fixed in extreme nodes of their lower chord in a statically determinate way according to the beam pattern.

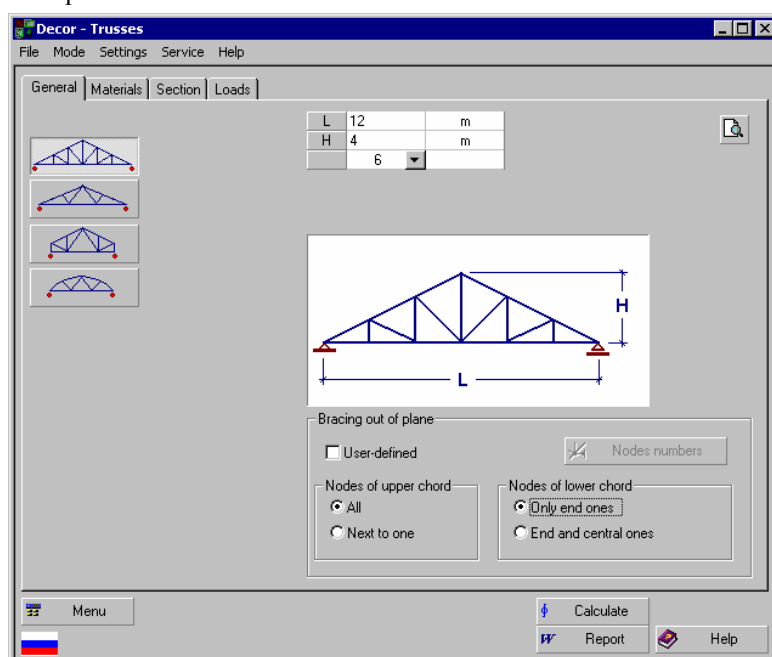


Fig. 44. The Trusses dialog box. The General tab

For the selected configuration, you need to specify the span of the truss, its height on the support and in the midspan, and the number of panels in the lower chord. Having entered all required geometric sizes, you can review the truss you have defined in the **Preview** dialog box (Fig. 45) called up by the  button.

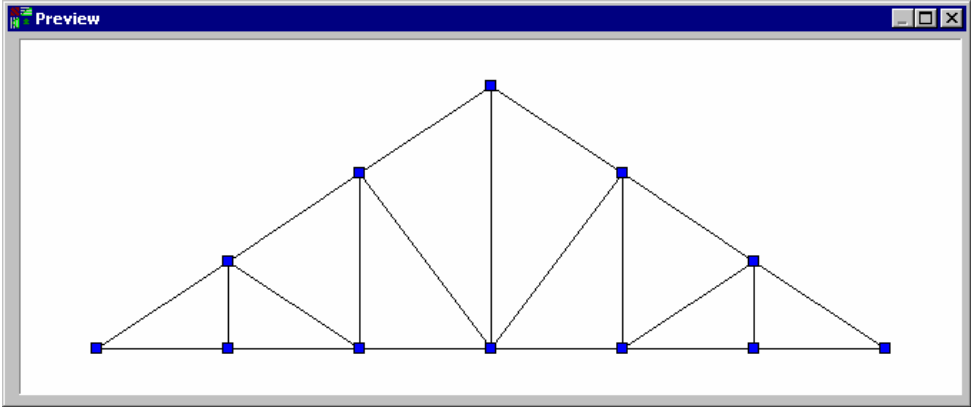


Fig. 45. The **Preview** dialog box

The **Bracing out of plane** group has checkboxes and radio buttons to indicate the method of bracing of lower and upper chord's panel points out of the plane of the truss (the bracing is assumed to be statically determinate in the truss's plane — a hinged immobile support for the left bearing node and a hinged mobile support for the right one).

A nonstandard arrangement of the bracing out of the plane can be defined. To do it, enable the **User-defined** checkbox and click the  button to open a dialog box with a table (Fig. 46). Points where there is bracing out of the plane are displayed in green on the schematic of the design model.

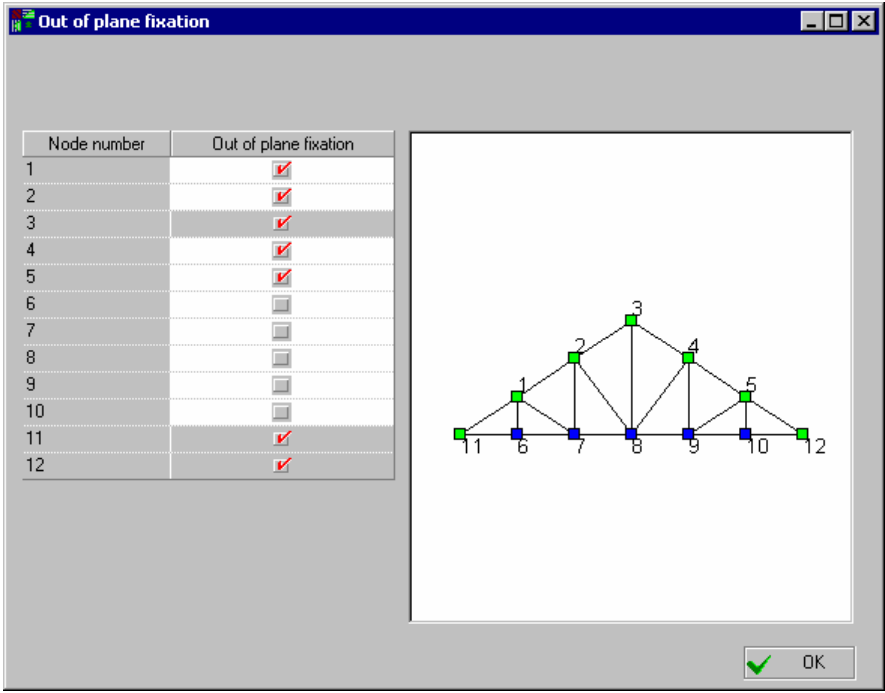


Fig. 46. The **Bracing out of Plane** dialog box

The **Materials** tab (Fig. 47) is used to specify, following standard rules (see Section **Error! Reference source not found.**), data concerning wood and service factors.

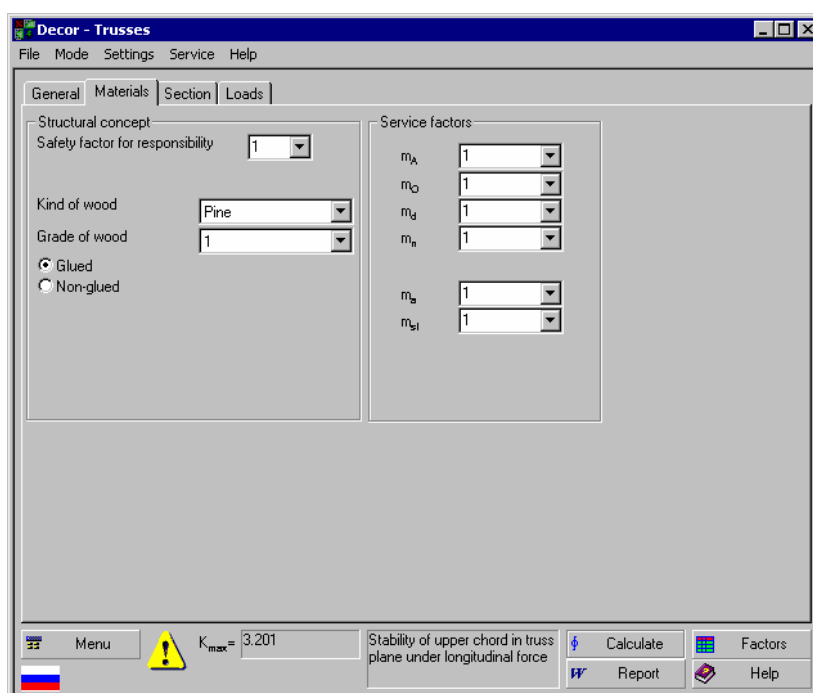


Fig. 47. The **Trusses** dialog box. The **Materials** tab

The **Sections** tab (Fig. 48) is used to specify sizes of cross-sections for every group of the truss members (the upper chord, the lower chord, diagonals, stanchions, supporting diagonals).

The **Loads** tab (Fig. 49) is much similar to the respective tab of the **Continuous Girders** mode described earlier (see Section 3.6); however, it has its peculiar features. First, only a uniformly distributed load or a concentrated force is allowed; second, the location of the concentrated force is defined by No. of its application node. As for the distributed loads, the application makes it possible to specify simultaneous loads on a group of members selected by the user: the upper/lower chord or the left/right half of a chord.

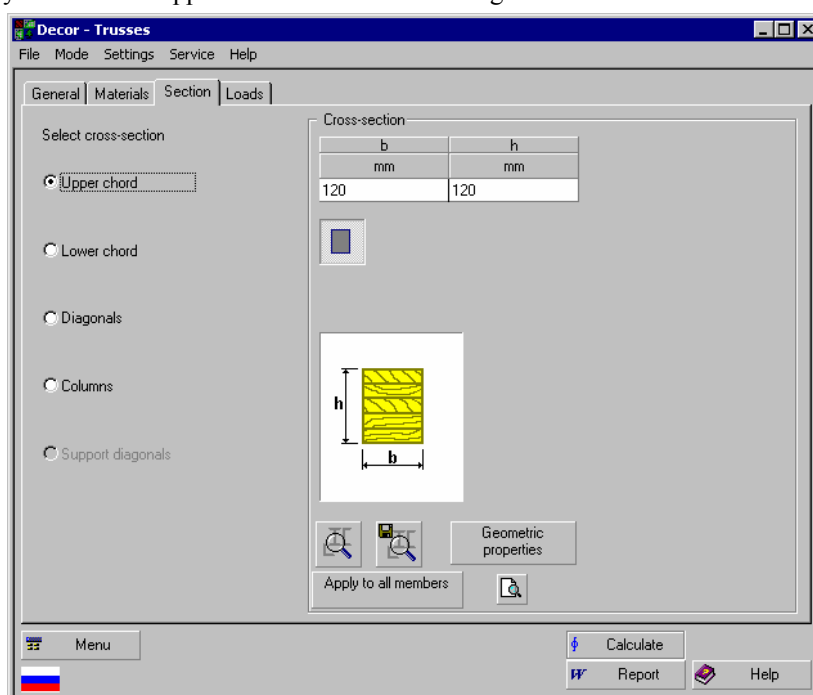


Fig. 48. The **Trusses** dialog box. The **Sections** tab

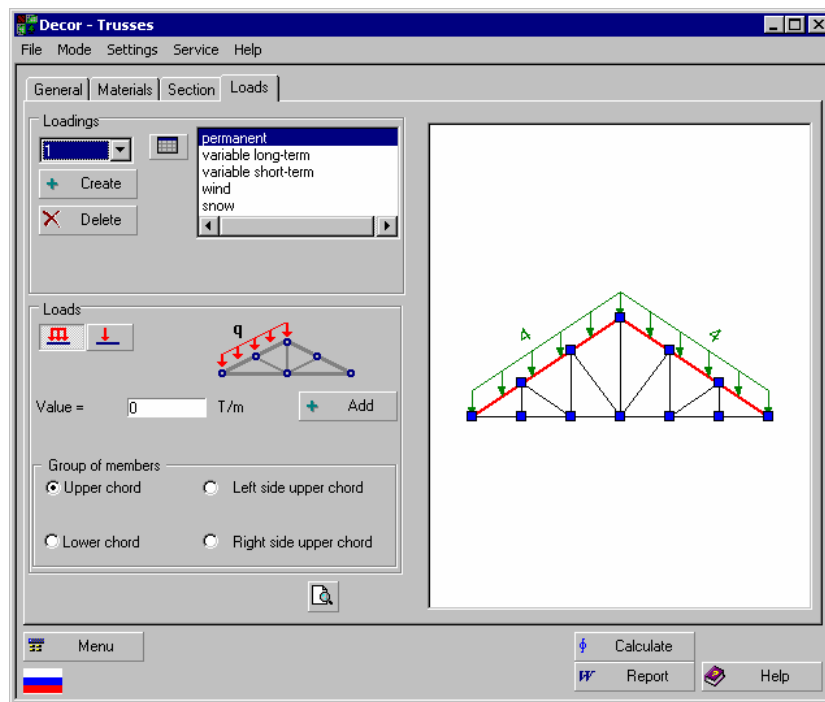


Fig. 49. The **Trusses** dialog box. The **Loads** tab

Stresses in the truss's elements for the current loading are displayed as distribution diagrams on the **Preview**

tab opened by clicking the  button (Fig. 50).

The trusses are checked for the following:

- slenderness out of the truss's plane — Sec. 4.4 of SNiP II-25-80;
- slenderness in the truss's plane — Sec. 4.4 of SNiP II-25-80;
- strength of the element under a tensioning longitudinal force — Sec. 4.1 of SNiP II-25-80;
- strength of the element under a compressing longitudinal force — Sec. 4.2 of SNiP II-25-80;
- stability in the truss's plane under a longitudinal force — Sec. 4.2 of SNiP II-25-80;
- stability out of the the truss's plane under a longitudinal force — Sec. 4.2 of SNiP II-25-80.

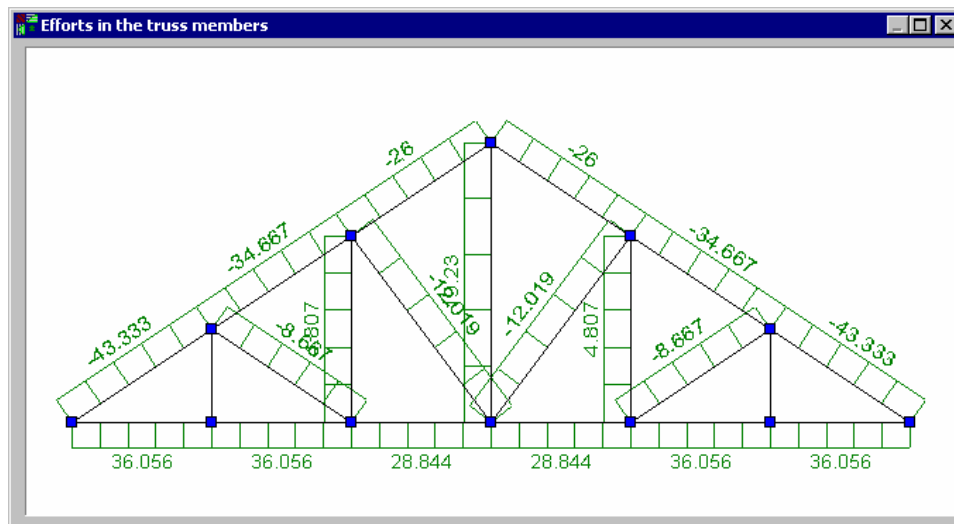


Fig. 50. The **Stresses in Truss Elements** dialog box

3.11 Truss element

The **Truss** mode described above suggests a most frequently used but not complete set of truss constructions. To analyze elements of any arbitrary truss, the application suggests a special mode entitled **Truss Element** capable of analyzing the load-bearing ability of a truss element (the static analysis is assumed to have been done earlier, and the stresses are assumed to be already known).

The **General** tab (Fig. 51) is used to specify data concerning the section of the element, its type (chord element, web element, ...), wood, and service factors. Also, there is a table to specify stresses (longitudinal forces) for any desired number of loadings.

Decor - Truss element

File Mode Settings Service Help

General

Structural concept

Safety factor for responsibility: 1

Member length: 3 m

Kind of wood: Pine

Grade of wood: 1

☒ Glued
☐ Non-glued

Member type: Chord element

Out of plane fixation spacing: 1 m

Service factors

m_A : 1
 m_O : 1
 m_d : 1
 m_n : 1
 m_b : 1
 m_{el} : 1

Cross-section

b	h
mm	mm
120	260

Geometric characteristics

Efforts

	N	T
1	-131.201	
2	-129.577	
3	-128.797	
4	480.336	
5	-687.696	
6	480.186	
7	-687.546	

Add Delete

Menu $K_{max} = 15.445$ Stability in truss plane under longitudinal force Calculate Factors Report Help

Fig. 51. The **Truss Element** dialog box

The truss element is checked for the following:

- slenderness out of the truss's plane — Sec. 4.4 of SNiP II-25-80;
- slenderness in the truss's plane — Sec. 4.4 of SNiP II-25-80;
- strength of the element under a tensioning longitudinal force — Sec. 4.1 SNiP II-25-80;
- strength of the element under a compressing longitudinal force — Sec. 4.2 of SNiP II-25-80;
- stability in the truss's plane under a longitudinal force — Sec. 4.2 of SNiP II-25-80;
- stability out of the the truss's plane under a longitudinal force — Sec. 4.2 of SNiP II-25-80.

4. APPENDIX

4.1 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. 52) 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, $2.5 \cdot 2.5 \cdot 2.5$ can be written also as 2.5^3 .

The following mathematical functions can be used in the formulas:

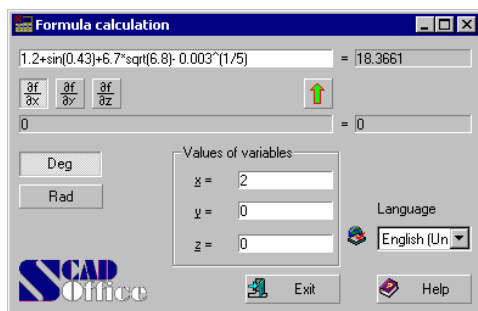


Fig. 52. The dialog box of the calculator

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

$$1.2+\sin(x)+6.7*\sqrt{6.8}-y^{(1/5)}.$$

The 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.

4.2 Converter of measurement units



This 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. 53). To do the action, select a tab of respective measures (**Length**, **Area** etc.).

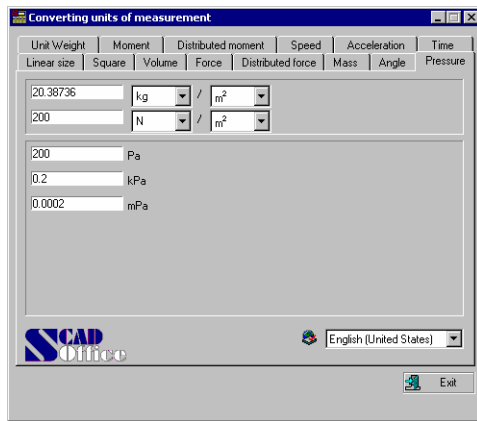


Fig. 53. *The Convert Units of Measurement dialog box*

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

To convert between 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 the results of this conversion in the edit field of the other line.

References

1. SNiP II-25-80. Wooden constructions/State Committee of Russia for Construction. Moscow, 2001, 30 p.
2. GOST 24454-80. Softwood timber. Dimensions.
3. GOST 7307-75*. Parts made of wood and wooden materials. Allowances for machining.
4. GOST 27751-88. Structural safety of constructions and foundations. Principles of analysis.
5. SNiP 2.01.07-85*. Loads and actions/ State Committee of Russia for Construction. Moscow, 2001, 44 p.
6. V. Karpilovsky, E. Kriksunov, A. Maliarenko, M. Mikitarenko, A. Perelmuter, M. Perelmuter, V. Fedorovsky, *SCAD Office. Implementation of SNiP in computer-aided design software*. Moscow, ASV Press, 2004, 288 p.
7. I.M. Grin, V.V. Fursov, D.M. Babushkin, P.G. Galushko, V.I. Grin, *Engineering and analysis of wooden constructions. Reference manual*. Kiev, "Budivelnik" Press, 1988, 240 p.